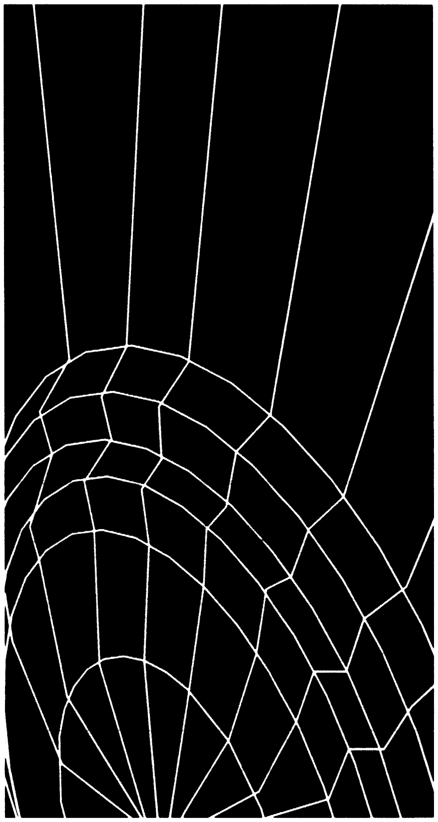
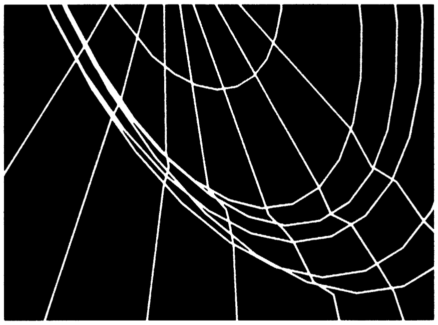
MARC
Analysis
Research
Corporation



MARC



Volume B

Element Library

Version K7

MARC
Analysis
Research
Corporation

Copyright © 1997 MARC Analysis Research Corporation
Printed in U. S. A.
This notice shall be marked on any reproduction of this data, in whole or in part.

MARC Analysis Research Corporation
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306 USA

Phone: (650) 329-6800
FAX: (650) 323-5892

Document Title: **MARC Volume B: Element Library, Version K7**
Part Number: RF-3002-07
Revision Date: August, 1997

Proprietary Notice

MARC Analysis Research Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

ALTHOUGH DUE CARE HAS BEEN TAKEN TO PRESENT ACCURATE INFORMATION, MARC ANALYSIS RESEARCH CORPORATION DISCLAIMS ALL WARRANTIES WITH RESPECT TO THE CONTENTS OF THIS DOCUMENT (INCLUDING, WITHOUT LIMITATION, WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE) EITHER EXPRESSED OR IMPLIED. MARC ANALYSIS RESEARCH CORPORATION SHALL NOT BE LIABLE FOR DAMAGES RESULTING FROM ANY ERROR CONTAINED HEREIN, INCLUDING, BUT NOT LIMITED TO, FOR ANY SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF, OR IN CONNECTION WITH, THE USE OF THIS DOCUMENT.

This software product and its documentation set are copyrighted and all rights are reserved by MARC Analysis Research Corporation. Usage of this product is only allowed under the terms set forth in the MARC Analysis Research Corporation License Agreement. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MARC Analysis Research Corporation is prohibited.

Restricted Rights Notice

This computer software is commercial computer software submitted with "restricted rights." Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c)(i)(ii) or the Rights in technical Data and Computer Software clause at DFARS 252.227-7013, NASA FAR Supp. Clause 1852.227-86, or FAR 52.227-19. Unpublished rights reserved under the Copyright Laws of the United States.

Trademarks

All products mentioned are the trademarks, service marks, or registered trademarks of their respective holders.

Contents

Chapter 1 Introduction

Chapter 2 MARC Element Classifications

Chapter 3 Element Library

Figures

Figure **Page**

Figure **Page**

Figure **Page**

Figure **Page**

Figures

Tables

1

Introduction

Library Elements

MARC contains an extensive element library. These elements provide coverage of plane stress and plane strain structures, axisymmetric structures (shell type or solid body, or any combination of the two), plate, beam and arbitrary shell structures, and full three-dimensional solid structures. A short description of each element and a summary of the data necessary for use of the elements is included in this section. Note that many elements serve the same purpose. Where possible, we have indicated the preferred element that should be used. In general, we have found that the lower-order quadrilateral elements give significantly better results than triangular elements in two dimensions. In problems involving thermal dependence and/or nonlinear behavior, the higher-order isoparametric elements should be used. When using the contact capability, lower-order elements are advantageous. The CENTROID parameter should only be used for linear analyses. Plate analysis can be performed by degenerating one of the shell elements. The elements with midside nodes require a large bandwidth for the solution of the master stiffness matrix. As a general comment, you should be aware that the sophisticated elements in MARC enables problems to be solved with many fewer elements when compared to solutions with conventional constant stress elements. This requires you to exercise your analytical skills and judgement. In return, you will be rewarded by a more accurate stress and displacement picture.

In this and subsequent sections, reference will be made to the first and second nodes, etc., of an element in order to define either the direction or sequence of the nodes. This order of the nodes is that which is defined by the connectivity matrix for the structure and is input by the CONNECTIVITY model definition block of Volume C.

Notes: For all elements right-handed coordinate systems are used. For all two-dimensional elements, right-handed rotation is counterclockwise in the plane. In this chapter, nodes are numbered in the order that they appear in the connectivity matrix. These numbers are, of course, replaced by the appropriate node numbers for an actual structural model. For all shell elements, stress and stiffness states are calculated at eleven representative points through the thickness unless modified using the SHELL SECT parameter or through the COMPOSITE model definition option.

All shear strains are engineering values, not tensor values.

Incompressible and Nearly Incompressible Elements (Herrmann Formulation)

Certain elements in the program (Elements 32-35, 58-61, 63, 66, 74, 80-84, 118-120, and 128-130) allow the study of incompressible and nearly incompressible materials in plane strain, axisymmetric, and three-dimensional cases through the use of an augmented variational principle based on the Herrmann Formulation (see Volume A). The element is assumed to be elastic (or viscoelastic). However, the implementation is general enough for large displacements, creep and thermal strains to be taken into account. Note that although these elements only allow elastic and viscoelastic behavior, they can be used in conjunction with other elastic-plastic elements in the same mesh. These elements are also used in RIGID-PLASTIC flow analysis problems (see Volume A). The elements can also be used to advantage for compressible elastic materials, since their hybrid formulation usually gives more accurate stress prediction.

Reduced Integration Elements

For a number of isoparametric elements in the program (Elements 22, 53-61, 69-71, 73 and 74, 114-123), a reduced integration scheme is used to determine the stiffness matrix of the element. In such a reduced scheme, the integration is not exact, the contribution of the highest order terms in the deformation field is neglected. Reduced integration elements have specific advantages and disadvantages. The most obvious advantage is the reduced cost for element assembly. This is specifically significant for the three-dimensional elements (Elements 57, 61, 71, 117, 120, and 123). Another advantage is the improved accuracy which may be obtained with reduced integration elements for higher-order elements. The increase in accuracy is due to the fact that the higher-order deformation terms are coupled to the lower order terms. The coupling is strong if the elements are distorted. The higher-order terms cause strain gradients within the element which are not present in the exact solution. Hence, the stiffness is overestimated. Since the reduced integration scheme does not take the higher-order terms into account, this effect is not present in the reduced integration elements.

The same feature also forms the disadvantage of the element. Each of the reduced integration elements has some specific higher-order deformation mode(s) which do not give any contribution to the strain energy in the element. The planar elements have one such "breathing" or "hourglass" mode, shown in Figure 1-1. Whereas, the three-dimensional bricks have six breathing modes. Breathing modes may become dominant in meshes with a single array (8-node quads) or single stack (20-node bricks) of elements. In meshes using higher-order elements of this type, sufficient boundary conditions should be prescribed to suppress the breathing modes, or the exact integration element should be used. It can also be advantageous to combine reduced integration elements with an element with exact integration in the same mesh – this is always possible in MARC.

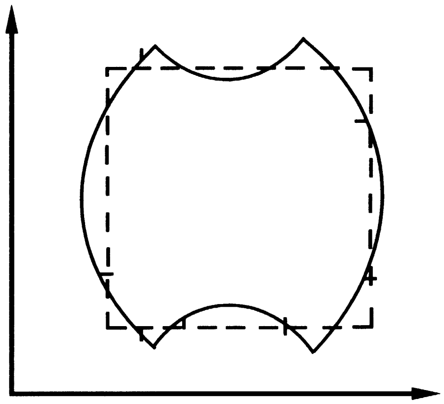


Figure 1-1 "Breathing" or "Hourglass" Mode

Reduced Integration with Hourglass Control

The lower-order reduced integration elements (114 to 123, and 140) are formulated with an additional contribution to the stiffness matrix based upon a consistent Hu-Washizu variational principle to eliminate the hourglass modes normally associated with reduced integration elements. These elements use modified shape functions based upon the natural coordinates of the element, similar to the assumed strain elements. They are very accurate for elastic bending problems. You should note that as there is only one integration point for each element, elastic-plastic behavior is only evaluated at the centroid which may result in a loss in accuracy if only a single element is used through the thickness. Additionally these elements do not lock when incompressible or nearly incompressible behavior is present. Unlike the assumed strain elements or the constant dilatation elements, no additional flags are required on the GEOMETRY option.

Assumed Strain Elements

For a number of linear elements in the program (Elements 3, 7, 11), a modified interpolation scheme can be used which improves the bending characteristics of the elements. This allows the ability to capture pure bending using a single element through the thickness. It is activated by use of the ASSUMED STRAIN parameter or by setting the third field of the GEOMETRY option to one. This can substantially improve the accuracy of the solution though the stiffness assembly computational costs will increase.

Constant Dilatational Elements

For a number of linear elements in the program (Elements 7, 10, 11, 19 and 20), an optional integration scheme can be used which imposes a constant dilatational strain constraint on the element. This is often useful for inelastic analysis where incompressible or nearly incompressible behavior occurs. This option should always be used in metal forming analysis. It is activated by use of the CONSTANT DILATATION parameter or by setting the second field of the GEOMETRY option to one.

Solution Procedures for Large Strain Analysis

There are two procedures; the total Lagrange and the updated Lagrange in MARC for the solution of large strain problems, each of which influences the choice of element technology, and the resultant quantities. Large strain analysis is broadly divided between elastic analyses, using either the Mooney, Ogden, or Foam model, and plasticity analyses using the von Mises yield criteria. Any analysis may have a mixture of material types and may use multiple solution procedures.

Large Strain Elasticity

The default procedure is the total Lagrange method, and is applicable for the Mooney, Ogden, and Foam model. When using the Mooney or Ogden material models for plane strain, generalized plane strain, axisymmetric, axisymmetric with twist, or three-dimensional solids in the total Lagrange frame work, Herrmann elements must be used to satisfy the incompressibility constraint. For plane stress, membrane or shell analysis with these models, conventional displacement elements should be used. The elements will thin, to satisfy the incompressibility constraint. When using the Foam material model, conventional displacement elements should be used. Using this procedure, the output includes the second Piola-Kirchhoff stress, the Cauchy (true) stress and the Green-Lagrange strain.

The updated Lagrange procedure can be used with the Mooney or Ogden material model, by using the ELASTICITY parameter. This procedure is not yet available for plane stress, membrane, or shell elements. When this procedure is used for continuum models, the elements should be conventional displacement based elements. A three-field variational principal is used to insure that incompressibility is satisfied. For more details, see Volume A. Using this procedure, the output includes the Cauchy (true) stress and the logarithmic strain.

Large Strain Plasticity

The updated Lagrange procedure should be used for large strain plasticity problems. There are two methods used for implementing the elastic-plastic kinematics; the additive decomposition and the multiplicative decomposition procedure. With the dominance of plasticity in the solution, the deformation becomes (approximately) incompressible. Both decomposition procedures uses conventional displacement elements for plane stress, plane strain, axisymmetric, or three-dimensional solids. When using the additive decomposition procedure, you should use either:

A. The "cross triangle" approach where four triangles form the edges and the diagonals of a quadrilateral element;

B. The modified volume strain integration (constant dilatation) approach, which is optionally available for element types 7, 10, 11, 19 and 20.

When using the multiplicative decomposition procedure, a three-field variational procedure is used to satisfy the nearly incompressible condition for plane strain, axisymmetric, and three-dimensional solids. This can be used for all lower and higher order displacement elements. This is summarized in the table below.

Output

When using the total Lagrange procedure or the updated Lagrange procedure, the strain and stress output is in the global x-y-z directions for continuum elements. For beam or shell elements, the output is in the local, element coordinate system based upon the original coordinate orientation for the total Lagrange procedure. For beams or shells, the output is in a co-rotational system attached to the deformed element if the updated Lagrange procedure is used.

Table 1-1 Large Strain Elasticity Element Selection

	Truss, Beam	**Membrane**	**Shell**	**Plane Stress**	**Plane Strain Axisymmetric 3D Solid**	**Strain Measure**	**Stress Measure**
Total Lagrange	conv.	conv.	conv.	conv.	Herrmann	Green-Lagrange	2nd Piola-Kirchhoff
Updated Lagrange	N/A	N/A	N/A	N/A	conv	Logarithmic	Cauchy

Table 1-2 Large Strain Plasticity Element Selection

	Truss, Beam	**Membrane**	**Shell**	**Plane Stress**	**Plane Strain Axisymmetric 3D Solid**	**Strain Measure**	**Stress Measure**
Updated Lagrange Additive Decomposition	conv.	conv.	conv.	conv.	conv. use Constant Dilatation	Logarithmic	Cauchy
Updated Lagrange Multiplicative Decomposition	N/A	N/A	N/A	N/A	conv 3-field variation	Logarithmic	Cauchy

Note: conv. stands for conventional displacement formulation.

Shell Layer Convention

The shell elements in MARC are numerically integrated through the thickness. The number of layers may be defined for homogeneous shells through the SHELL SECT parameter. The default is eleven layers. For problems involving homogeneous materials, Simpson's rule is used to perform the integration. For inhomogeneous materials, the number of layers is defined through the COMPOSITE model definition option. In such cases, the trapezoidal rule is used for numerical integration through the thickness.

The layer number convention is such that layer one lies on the side of the positive normal to the shell, and the last layer is on the side of the negative normal. The normal to the element is based upon both the coordinates of the nodal positions and upon the connectivity of the element. The definition of the normal direction can be defined for five different groups of elements.

Beams in Plane - Elements 16 and 45

For element 16, the s-direction is defined in the coordinate block; for element 45, the s-direction points from node 1 to node 3. The normal direction is obtained by a rotation of 90° from the direction of increasing s in the x-y plane.

Axisymmetric Shells - Elements 1, 15, 87, 88, 89, and 90

For element 1 or 88, the s-direction points from node 1 to node 2. For element 15, the s-direction is defined in the coordinate block. For elements 87, 89, and 90, the s-direction points from node 1 to node 3. The normal direction is obtained by a rotation of 90° from the direction of increasing s in the z-r plane.

Curvilinear Coordinate Shell Elements 4, 8, and 24

The normal to the surface $\underline{n}$ is $\underline{a}_1$ x $\underline{a}_2$, where $\underline{a}_1$ is a base vector tangent to the positive θ^1 line and $\underline{a}_2$ is a base vector tangent to the positive θ^2 line.

Triangular Shell Element 49 and 138

A set of basis vectors tangent to the surface is created first. The first V_1 is in the plane of the three nodes from node 1 to node 2. The second V_2 lies in the plane, perpendicular to V_1. The normal n is then formed as V_1 x V_2.

Shell Elements 22, 72, 75, 85, 86, 139, and 140

A set of base vectors tangent to the surface is first created. The first $\underline{t}_1$ is tangent to the first isoparametric coordinate direction. The second $\underline{t}_2$ is tangent to the second isoparametric coordinate direction. In the simple case of a rectangular element, t_1 would be in the direction from node 1 to node 2 and t_2 would be in the direction from node 2 to node 3. In the nontrivial (nonrectangular) case, a new set of vectors $\underline{V}_1$ and $\underline{V}_2$ would be created which are an orthogonal projection of $\underline{t}_1$ and $\underline{t}_2$. The normal is then formed as $\underline{n} = \underline{V}_1 \times \underline{V}_2$. Note that the vector $\underline{m} = \underline{t}_1 \times \underline{t}_2$ would be in the same general direction as $\underline{n}$. That is, $\underline{n} \cdot \underline{m} > 0$.

Shell Elements 50, 85, 86, 87, and 88

For heat transfer shell elements 50, 85, 86, 87, and 88, multiple degrees of freedom are used to capture the temperature variation through the thickness directions. The first degree of freedom is a temperature on the top surface; i.e., the positive normal side, while the second degree of freedom is on the bottom surface; i.e., the negative normal side. If a quadratic variation is selected, the third degree of freedom represents the temperature at the midsurface.

Transverse Shear for Thick Beams and Shells

Conventional finite element implementation of Mindlin shell theory results in the transverse shear distribution being constant through the thickness of the element. An extension has been made for the thick beam type 45 and the thick shells 22 and 75, or 140 such that a "parabolic" distribution of the transverse shear stress is obtained. In subsequent versions, a more "parabolic" distribution of transverse shear may be used. It is based upon a strength of materials beam equilibrium approach. For beam 45, the distribution is exact and the user no longer needs to correct his Poisson's ratio for the shear factor. For thick shells 22, 75, and 140 the new formulation is approximate since it is derived by assuming that the stresses in two perpendicular directions are uncoupled. To activate the parabolic shear distribution and calculation of interlaminar shear, include the TSHEAR parameter.

Beam Elements

When using beam elements, it is necessary to define four attributes of each element. The four attributes are defined in the following sections and summarized in Table 1-3 and Table 1-3.

Table 1-3 2D Beams

Element Type	Length	Beam Orientation	Cross-section Definition	Cross-section Orientation
5	$\lvert x_2 - x_1 \rvert$	1 to 2	Rectangular via GEOMETRY	Global z
16	User Defined s	Increasing s	Rectangular via GEOMETRY	Global z
14	$\lvert x_3 - x_1 \rvert$	1 to 3	Rectangular via GEOMETRY	Global z

Table 1-4 3D Beams

Element Type	Length	Beam Orientation	Cross-section Definition	Cross-section Orientation
13	User Defined s	Increasing s	Open section via BEAM SECT	via COORDINATES allows twist
14	$\lvert x_2 - x_1 \rvert$	1 to 2	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
25	$\lvert x_2 - x_1 \rvert$	1 to 2	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
31	Bending radius or $\lvert x_2 - x_1 \rvert$	1 to 2	Circular Pipe or BEAM SECT	GEOMETRY
52	$\lvert x_2 - x_1 \rvert$	1 to 2	A, I_{xx}, I_{yy} on GEOMETRY or BEAM SECT	GEOMETRY or COORDINATES
76	$\lvert x_3 - x_1 \rvert$	1 to 3	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
77	$\lvert x_3 - x_1 \rvert$	1 to 3	Open section via BEAM SECT	GEOMETRY or COORDINATEs
78	$\lvert x_2 - x_1 \rvert$	1 to 2	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES

Table 1-4 3D Beams (Continued)

Element Type	Length	Beam Orientation	Cross-section Definition	Cross-section Orientation
79	$\|x_2 - x_1\|$	1 to 2	Open section via BEAM SECT	GEOMETRY or COORDINATES
98	$\|x_2 - x_1\|$	1 to 2	A, I_{xx}, I_{yy} on GEOMETRY or BEAM SECT	GEOMETRY or COORDINATES
Note: Elements 31, 52, and 98 are elastic elements.				

Length of Element

The length of a beam element is generally the distance between the last node and the first node of the element. The exceptions are for the curved beams 13, 16, and 31. For element types 13 and 16, the user provides s as the length of the beam. If element 13 is used as a pipe bend, the length will depend on the bending radius.

Orientation of Beam Axis

The orientation of the beam (local z-axis) is generally from the first node to the last node. The exceptions are for the curved beams 13 and 16. For these elements, the axis is in the direction of increasing s (see Length of Element above).

Cross-section Properties

The cross-section properties fall into four different groups.

1. For two dimensional beams (elements 5, 16, and 95), only rectangular beams are possible. The height and thickness are given through the GEOMETRY option. The stress strain law is integrated through the height of the beam.
2. For elastic beam elements 52 or 98, the area and moment of inertias can be specified directly through the GEOMETRY option. For the elastic pipe-bend element 31, the radius and thickness are defined in the GEOMETRY option. The BEAM SECT option can also be used to define the properties for these elements.
3. For closed-section elements (14, 25, 76, and 78), the default cross section is a circular cross section (see Figure 1-2). When using the default circular section, 16 points are used to integrate the material behavior through the cross section. General closed section elements can be defined using the BEAM SECT option.

4. For open section beam elements (13, 77, and 79), the BEAM SECT must be used to define the cross section.

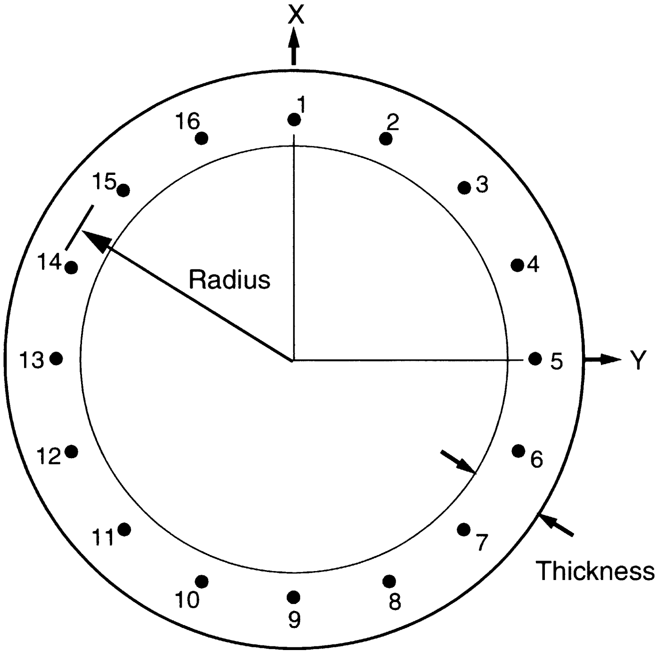
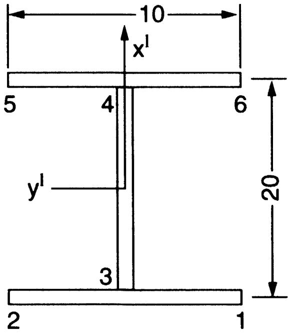


Figure 1-2 Default Cross Section

Definition of the Section

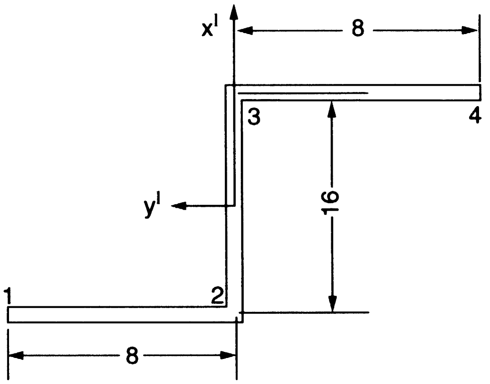
In any problem, any number of different beam sections may be included. Each section is defined by data cards following the BEAM SECT parameter of the MARC input given in Volume C. The sections will be numbered 1, 2, 3, etc. in the order they are input. Then, a particular section is obtained for an element by setting EGEOM2 (GEOMETRY option, ***2nd data line***, second field) to the floating point value of the section number; i.e., 1, 2, or 3. For elements 14, 25, 76, or 78 if EGEOM1 is nonzero, the default circular section is assumed. The beam section must be defined for open section beams. The section is defined using input data as shown in Figure 1-3.

Section 1

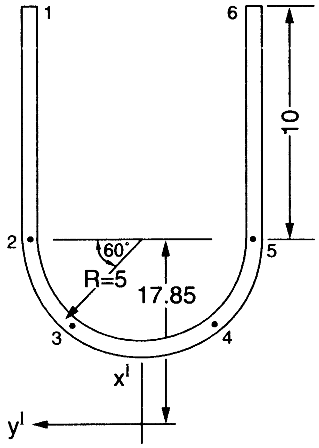


Branch	Divisions	Thickness
1-2	8	1.0
2-3	4	0.0
3-4	8	0.3
4-5	4	0.0
5-6	8	1.0

Section 2



Branch	Divisions	Thickness
1-2	10	1.0
2-3	10	1.0
3-4	10	1.0



Section 3

Branch	Divisions	Thickness
1-2	8	0.5
2-3	4	0.5
3-4	8	0.5
4-5	4	0.5
5-6	8	0.5

Figure 1-3 Section Definition Examples

The rules and conventions for defining a section are as follows:

A. The section is defined in an x^1 - y^1 coordinate system, with x^1 the first director at a point of the beam (see Figure 1-4).

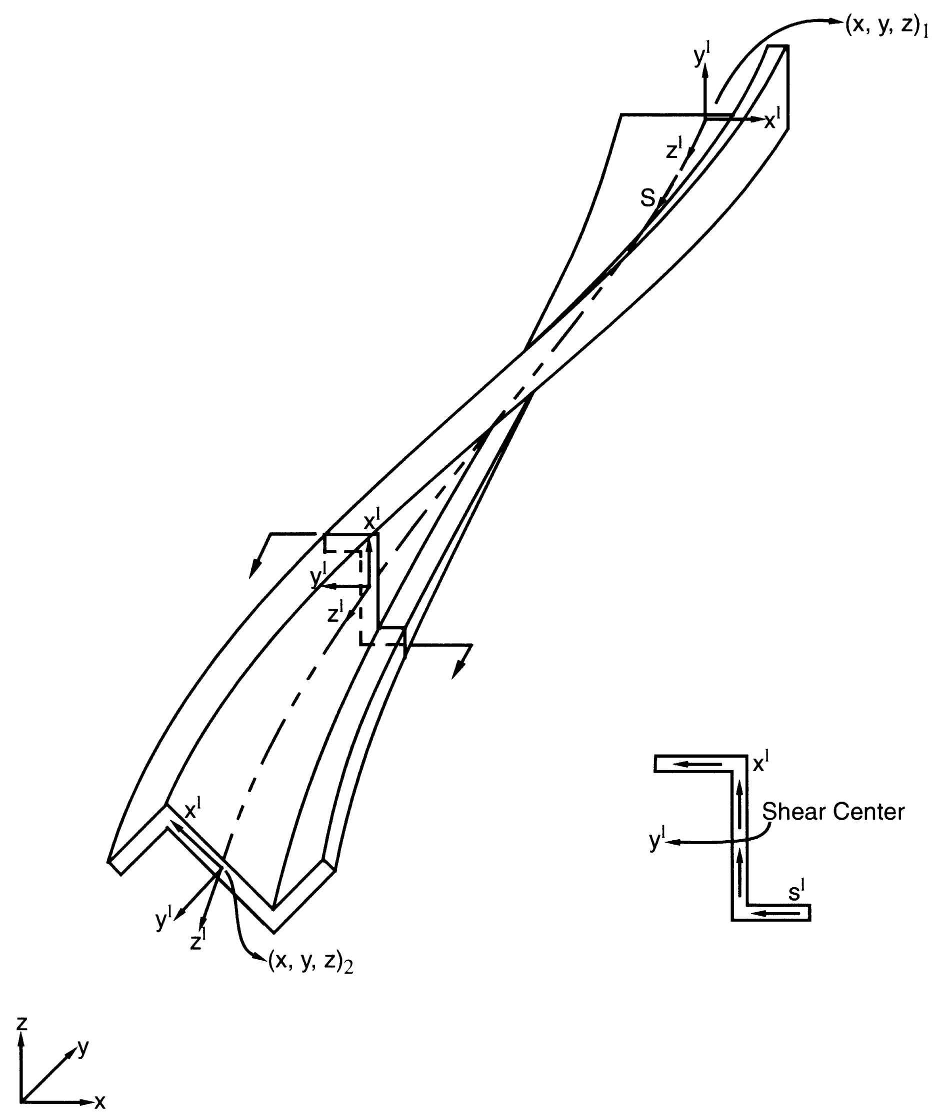


Figure 1-4 Beam Element Type 13 Including Twist, (x^l, y^l, z^l)

B. The section is input as a series of branches. Each branch may have a different geometry, but the branches must form a complete transverse of the section in the input sequence. Thus, the end point of a branch is always the start of the next branch. It is legitimate (in fact, it is often necessary) for the traverse of the section to double back on itself. This is achieved by specifying a branch with zero thickness.

C. Each branch is divided (by the user) into segments. The stress points of the section, that is, the points used for numerical integration of section stiffness and also for output of stress, will be the segment division points. Then end points of any branch will always be stress points, and *there must always be an even number of divisions* (nonzero) in any branch. A maximum of 31 stress points (30 divisions) may be used in a complete section, not counting branches of zero thickness.

D. Within a branch, the thickness will vary linearly between the values given for thickness at the end points of the branch. The thickness may be discontinuous between branches. If the thickness at the end of the branch is given as an exact zero, the branch is assumed to be of constant thickness, equal to the thickness given at the beginning of the branch.

E. The shape of any branch will be interpolated as a cubic based on the values of x^1 and y^1 and their directions with respect to distance along the branch, which are input at the two ends of the branch. If both $\frac{dx^1}{ds}$ and $\frac{dy^1}{ds}$ are given as exact zeros at both ends of the branch, the branch is assumed to be straight as a default condition. Notice that x^1 and y^1 for the beginning of the branch are given only for the first branch of a section, since the beginning point of any other branch must be the same as the end point of the previous branch. Notice also that the section can have a discontinuous slope at the branch ends.

F. Any stress points separated by a distance less than t/10, where t is a thickness at one of these points, will be merged into one point.

Example

As an example of defining beam sections, the three sections shown in Figure 1-3 may be defined by data shown in Table 1-5. The corresponding output for the I-Section is shown in Table 1-6 and Table 1-7. The program provides the user with the location of each stress point on the section, the thickness at that point, the weighting associated with each point (for numerical integration of section stiffness) and the warping function (sectorial area) at each point. Notice the use of zero thickness branches in the traverse of the I-Section.

Standard (Default) Circular Section

For elements 14, 25, 76, and 78, the default cross section is that of a circular pipe. The positions of the 16 numerical integration points are shown in Figure 1-2. The contributions of the 16 points to the section quantities are obtained by numerical integration using Simpson's rule.

Table 1-5 Beam Section Definition Data

12345678901234567890123456789012345678901234567890123456789012345678901234567890							
Xb	**Yb**	**dXb/ds**	**dYb/ds**	**Xe**	**Ye**	**dXe/ds**	**dYe/ds**
s	**tb**	**te**					
beam sect							
i-section							
5 8	4 8	4 8					
-10.0	-5.0	0.0	1.0	-10.0	5.0	0.0	1.0
10.0	1.0	1.0					
		0.0	-1.0	-10.0	0.0	0.0	-1.0
5.0	0.0	0.0					
		1.0	0.0	10.0	0.0	1.0	0.0
20.0		0.5					
		0.0	1.0	10.0	5.0	0.0	1.0
5.0	0.0	0.0					
		0.0	-1.0	10.0	-5.0	0.0	-1.0
10.0	1.0	1.0					
last							

Table 1-6 I-Section, Branch Definition Output Listing

I-Section										
Number of Branches 5			**Intervals Per Branch 8 4 8 4 8**							
Branch Definition										
Branch	**x1**	**y1**	**x1p**	**y1p**	**x2**	**y2**	**x2p**	**y2p**	**p1**	**t1**	**t2**
1	-10.000	-5.000	0.000	1.000	-10.000	5.000	0.000	1.000	10.000	1.000	1.000
2	-10.000	5.000	0.000	-1.000	-10.000	0.000	0.000	-1.000	5.000	0.000	0.000
3	-10.000	0.000	1.000	0.000	10.000	0.000	1.000	0.000	20.000	0.500	0.500
4	10.000	0.000	0.000	1.000	10.000	5.000	0.000	1.000	5.000	0.000	0.000
5	10.000	5.000	0.000	-1.000	10.000	-5.000	0.000	-1.000	10.000	1.000	1.000

Table 1-7 Beam Section Point Coordinates and Weights

Section 1 (Open)					
Point Number	**Coordinates in Section**		**Thickness**	**Warping Ftn.**	**Weight**
1	-10.00000	-5.00000	1.00000	-50.00000	0.41667
2	-10.00000	-3.75000	1.00000	-37.50000	1.66667
3	-10.00000	-2.50000	1.00000	-25.00000	0.83333
4	-10.00000	-1.25000	1.00000	-12.50000	1.66667
5	-10.00000	0.00000	0.75000	0.00000	1.25000
6	-10.00000	1.25000	1.00000	12.50000	1.66667
7	-10.00000	2.50000	1.00000	25.00000	0.83333
8	-10.00000	3.75000	1.00000	37.50000	1.66667
9	-10.00000	5.00000	1.00000	50.00000	0.41667
10	-7.50000	0.00000	0.50000	0.00000	1.66667
11	-5.00000	0.00000	0.50000	0.00000	0.83333
12	-2.50000	0.00000	0.50000	0.00000	1.66667
13	0.00000	0.00000	0.50000	0.00000	0.83333
14	2.50000	0.00000	0.50000	0.00000	1.66667
15	5.00000	0.00000	0.50000	0.00000	0.83333
16	7.50000	0.00000	0.50000	0.00000	1.66667
17	10.00000	0.00000	0.75000	0.00000	1.25000
18	10.00000	5.00000	1.00000	-50.00000	0.41667
19	10.00000	3.75000	1.00000	-37.50000	1.66667
20	10.00000	2.50000	1.00000	-25.00000	0.83333
21	10.00000	1.25000	1.00000	-12.50000	1.66667
22	10.00000	-1.25000	1.00000	12.50000	1.66667
23	10.00000	-2.50000	1.00000	25.00000	0.83333
24	10.00000	-3.75000	1.00000	37.50000	1.66667
25	10.00000	-5.00000	1.00000	50.00000	0.41667

Cross-section Orientation

The cross-section axis orientation is important both in defining the beam section and in interpreting the results. The moments given in the output and on the post file are with respect to the local axes defined here. The definition falls into the following three groups:

1. For two-dimensional beam elements (5, 16, and 45), there is no choice. The local x-axis is in the global z-direction.
2. For element type 13, the direction cosines of the local axis is given in the coordinates block. Different values can be prescribed at both nodes of the beam. This allows you to prescribe a twist to the element.
3. For all other beam elements, the local axis can be given in either of two ways:
   - a. The coordinates of a point (point 3) is chosen to define the local x-z plane (see Figure 1-5).
   - b. The direction cosines of a vector in the x-y plane is defined. The local x-axis is then the component of this vector which is perpendicular to the z-axis (see Figure 1-6).

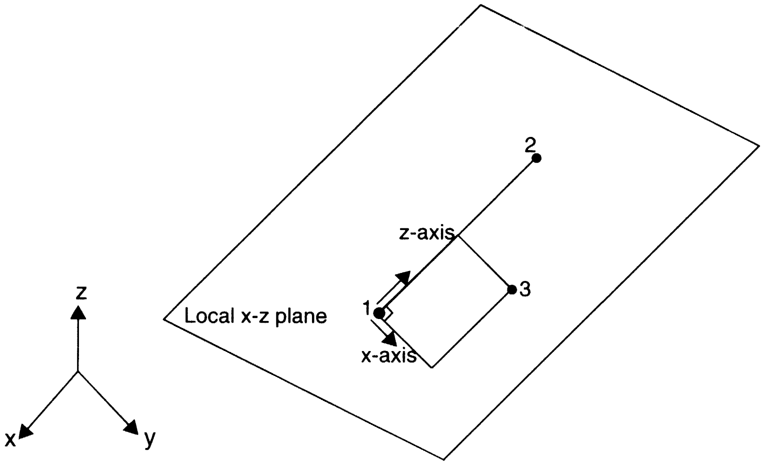


Figure 1-5 Local Axis Defined by Giving a Point

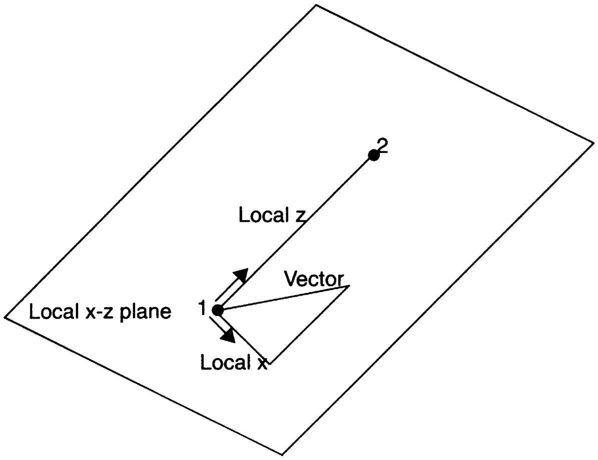


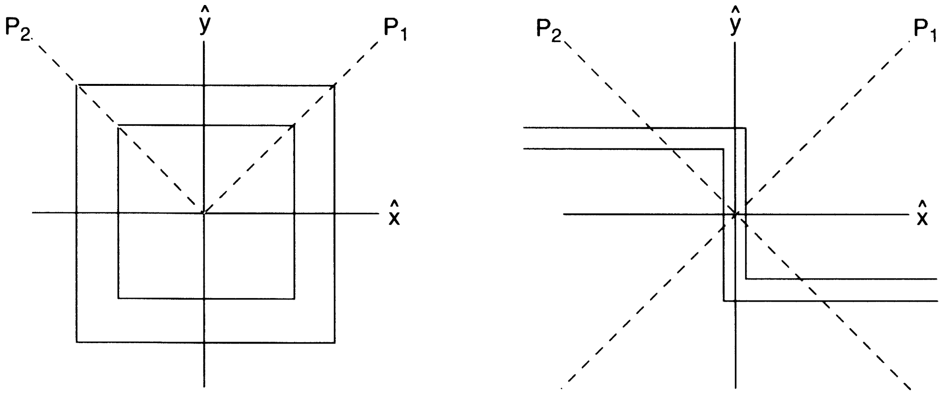
Figure 1-6 Local Axis Defined by Giving a Vector

Location of the Local Cross-section Axis

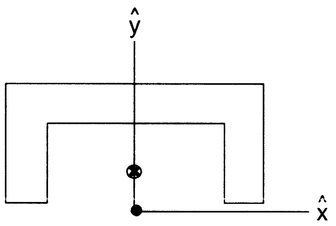
The origin of the local coordinate system is at the nodal point. The beam section is defined with respect to this point. If the origin is the shear center of the cross section, the element will behave in the classical manner, a bending moment parallel to a principle axis of the cross section will cause no twisting.

In the output, the location of the integration points is given with respect to the shear center (see Figure 1-7).

For the default circular cross section, the local origin is at the center, which is also the shear center.



Local Axis through Shear Center
Principle Axis – P_1, P_2



Local Axis not at the Shear Center

Figure 1-7 Location of Local Axis

Element Characteristics

In Chapter 3, the element description appears in the sequence in which they are developed in the program. However, it is easier to recognize the applicability of each element by grouping it in its own structural class. This is shown in Table 1-8. Table 1-9 lists the element library in the sequence that the elements were developed.

Each element type has unique characteristics governing its behavior. This includes the number of nodes, the number of direct and shear stress components, the number of integration points used for stiffness calculations, the number of degrees of freedom, and the number of coordinates. This information is summarized in Table 1-10. These values are also very useful when writing user subroutines (see Volume D).

The availability of the updated Lagrange procedure for different element types is given in Table 1-10. If an element cannot support the updated Lagrange method, the total Lagrange method is used for that element.

A summary of which elements are available for heat transfer, Joule Heating, Conrad-gap, channels, electrostatics, magnetostatics, and radiation view factors is presented in Table 1-11.

Table 1-8 Structural Classification of Elements

Element Structural Type	Element Number	Function	Remarks
Three-dimensional truss	9	Linear	2-node straight
	12	Linear	4-node straight gap and friction
	51	Analytic	2-node cable element
	64	Quadratic	3-node curved
	97	Linear	4-node straight double gap and friction
Two-dimensional beam column	5	Linear/cubic	2-node straight
	16	Cubic	2-node curved
	45	Cubic	3-node curved Timoshenko theory
Three-dimensional beam column	13	Cubic	2-node curved open section
	14	Linear/cubic	2-node straight closed section
	25	Cubic	2-node straight closed section
	31	Analytic	2-node elastic
	52	Linear/cubic	2-node straight elastic
	76	Linear/cubic	2 + 1-node straight closed section; use with Element 72
	77	Linear/cubic	2 + 1-node straight open section; use with Element 72
	78	Linear/cubic	2-node straight closed section; use with Element 75
	79	Linear/cubic	2-node straight open section
	98	Linear	2-node straight Timoshenko theory
Axisymmetric shell	1	Linear/cubic	2-node straight
	15	Cubic	2-node curved
	89	Quadratic	3-node curved thick shell theory
	90	Quadratic	3-node curved with arbitrary loading; thick shell theory

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Plane stress	3	Linear	4-node quadrilateral
	26	Quadratic	8-node quadrilateral
	53	Quadratic	8-node reduced integration quadrilateral
	114	Linear/Assumed strain	4-node quadrilateral, reduced integration, with hourglass control
	124	Quadratic	6-node triangle
Plane strain	6	Linear	3-node triangle
	11	Linear	4-node quadrilateral
	27	Quadratic	8-node quadrilateral
	54	Quadratic	8-node reduced integration quadrilateral
	91	Linear/special	6-node semi-infinite
	93	Quadratic/special	9-node semi-infinite
	115	Linear/Assumed strain	4-node quadrilateral, reduced integration, with hourglass control
	125	Quadratic	6-node triangle
Generalized plane strain	19	Linear	4 + 2-node quadrilateral
	29	Quadratic	8 + 2-node quadrilateral
	56	Quadratic	8 + 2-node reduced integration quadrilateral

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Axisymmetric solid	2	Linear	3-node triangle
	10	Linear	4-node quadrilateral
	20	Linear	4-node quadrilateral with twist
	28	Quadratic	8-node quadrilateral
	55	Quadratic	8-node reduced integration quadrilateral
	62	Quadratic	8-node quadrilateral with arbitrary loading
	67	Quadratic	8-node quadrilateral with twist
	73	Quadratic	8-node reduced integration quadrilateral and arbitrary loading
	92	Linear/special	6-node semi-infinite
	94	Quadratic/special	9-node semi-infinite
	95	Linear	4-node quadrilateral with bending
	96	Quadratic	8-node quadrilateral with bending
	116	Linear/Assumed strain	4-node quadrilateral, reduced integration, with hourglass control
	126	Quadratic	6-node triangle
Membrane three-dimensional	18	Linear	4-node quadrilateral
	30	Quadratic	8-node quadrilateral
Doubly-curved thin shell	4	Cubic	4-node curved quadrilateral
	8	Fractional cubic	3-node curved triangle
	24	Cubic patch	4 + 4-node curved quadrilateral
	49	Linear	3 + 3-node curved triangle discrete Kirchhoff
	72	Linear	4 + 4-node twisted quadrilateral discrete Kirchhoff
	138	Linear	3-node triangle discrete Kirchhoff
	139	Linear	4-node twisted quadrilateral discrete Kirchhoff

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Doubly-curved thick shell	22	Quadratic	8-node curved quadrilateral with reduced integration
	75	Linear	4-node twisted quadrilateral
	140	Linear	4-node twisted quadrilateral, reduced integration with hourglass control
Three-dimensional solid	7	Linear	8-node hexahedron
	21	Quadratic	20-node hexahedron
	57	Quadratic	20-node reduced integration hexahedron
	107	Linear/special	12-node semi-infinite
	108	Quadratic	27-node semi-infinite special
	117	Linear/Assumed strain	8-node hexahedron, reduced integration with hourglass control
	127	Quadratic	10-node tetrahedron
	134	Linear	4-node tetrahedron
Incompressible plane strain	32	Quadratic	8-node quadrilateral
	58	Quadratic	8-node reduced integration quadrilateral
	80	Linear	4 + 1-node quadrilateral
	118	Linear/Assumed strain	4 + 1-node quadrilateral, reduced integration with hourglass control
	128	Quadratic	6-node triangle
Incompressible generalized plane strain	34	Quadratic	8 + 2-node quadrilateral
	60	Quadratic	8 + 2-node with reduced integration quadrilateral
	81	Linear	4 + 3-node quadrilateral

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Incompressible axisymmetric	33	Quadratic	8-node quadrilateral
	59	Quadratic	8-node with reduced integration quadrilateral
	63	Quadratic	8-node quadrilateral with arbitrary loading
	66	Quadratic	8-node quadrilateral with twist
	74	Quadratic	8-node reduced integration quadrilateral with arbitrary loading
	82	Linear	4 + 1-node quadrilateral
	83	Linear	4 + 1-node quadrilateral with twist
	119	Linear/Assumed strain	4 + 1-node quadrilateral reduced integration, with hourglass control
	129	Quadratic	6-node triangle
Incompressible three-dimensional solid	35	Quadratic	20-node hexahedron
	61	Quadratic	20-node with reduced integration hexahedron
	84	Linear	8 + 1-node hexahedron Node 9
	120	Linear/Assumed strain	8 + 1-node hexahedron, reduced integration with hourglass control
	130	Quadratic	10-node tetrahedron
Pipe bend	17	Cubic	2-nodes in-section; 1-node out-of-section
	31	Special	2-node elastic

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Rebar Elements	23	Quadratic	20-node hexahedron
	46	Quadratic	8-node quadrilateral plane strain
	47	Quadratic	8 + 2-node quadrilateral generalized plane strain
	48	Quadratic	8-node quadrilateral axisymmetric
	142	Quadratic	8-node axisymmetric with twist
	143	Linear	4-node plane strain
	144	Linear	4-node axisymmetric
	145	Linear	4-node axisymmetric with twist
	146	Linear	8-node hexahedron
	147	Linear	4-node membrane
	148	Quadratic	8-node membrane
Three-dimensional shear panel	68	Linear	4-node quadrilateral
Heat conduction three-dimensional link	36	Linear	2-node straight
	65	Quadratic	3-node curved
Heat conduction planar	37	Linear	3-node triangle
	39	Linear	4-node quadrilateral
	41	Quadratic	8-node quadrilateral
	69	Quadratic	8-node reduced integration quadrilateral
	101	Linear/special	6-node semi-infinite
	103	Linear/special	9-node semi-infinite
	121	Linear	4-node quadrilateral, reduced integration, with hourglass control
	131	Quadratic	6-node triangular

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Heat conduction axisymmetric	38	Linear	3-node triangle
	40	Linear	4-node quadrilateral
	42	Quadratic	8-node quadrilateral
	70	Quadratic	8-node reduced integration quadrilateral
	102	Linear/special	6-node semi-infinite
	104	Quadratic/special	9-node semi-infinite
	122	Linear	4-node quadrilateral, reduced integration, with hourglass control
	132	Quadratic	6-node triangular
Heat conduction solids	43	Linear	8-node hexahedron
	44	Quadratic	20-node hexahedron
	71	Quadratic	20-node reduced integration hexahedron
	105	Linear/special	12-node semi-infinite
	106	Quadratic/special	27-node semi-infinite
	123	Linear	8-node hexahedron, reduced integration, with hourglass control
	133	Quadratic	10-node tetrahedron
	135	Linear	4-node tetrahedron
Heat conduction shell	50	Linear	3-node triangle
	85	Linear	4-node quadrilateral
	86	Quadratic	8-node quadrilateral
Heat conduction axisymmetric shell	87	Quadratic	3-node curved
	88	Linear	2-node straight
Magnetostatic three-dimensional solids	109	Linear	8-node hexahedron
	110	Linear/special	12-node semi-infinite

Table 1-8 Structural Classification of Elements (Continued)

Element Structural Type	Element Number	Function	Remarks
Electromagnetic Planar	111	Linear	4-node quadrilateral
Electromagnetic Axisymmetric	112	Linear	4-node quadrilateral
Electromagnetic Solid	113	Linear	8-node hexahedron

Table 1-9 Element Library

Element	Description	Code
1	Two-node Axisymmetric Shell Element	(1)
2	Axisymmetric Triangular Ring Element	(2)
3	Two-dimensional (Plane Stress) Four-node, Isoparametric Quadrilateral	(3)
4	Curved Quadrilateral Thin-shell Element	(4)
5	Beam-column	(5)
6	Two-dimensional Plane Strain, Constant Stress Triangle	(6)
7	Eight-node Isoparametric Three-dimensional Hexahedron	(7)
8	Three-node, Triangular Arbitrary Shell	(8)
9	Three-dimensional Truss Element	(9)
10	Axisymmetric Quadrilateral Element (Isoparametric)	(10)
11	Plane Strain Quadrilateral Element (Isoparametric)	(11)
12	Friction and Gap Element	(12)
13	Open-section Beam	(13)
14	Closed-section Beam	(14)
15	Isoparametric, Two-node Axisymmetric Shell	(15)
16	Isoparametric, Two-node Curved Beam	(16)
17	Pipe-bend Element	(17)
18	Four-node, Isoparametric Membrane	(18)
19	Generalized Plane Strain Quadrilateral	(19)
20	Axisymmetric Torsional Quadrilateral	(20)
21	Three-dimensional, 20-node brick	(21)
22	Curved Quadrilateral Thick-shell Element	(22)
23	Three-dimensional, 20-node Rebar Element	(23)
24	Curved Quadrilateral Shell Element	(24)
25	Closed-section Beam in Three Dimensions	(25)
26	Plane Stress, Eight-node Distorted Quadrilateral	(26)

Table 1-9 Element Library (Continued)

Element	Description	Code
27	Plane Strain, Eight-node Distorted Quadrilateral	(27)
28	Axisymmetric, Eight-node Distorted Quadrilateral	(28)
29	Generalized Plane Strain, Distorted Quadrilateral	(29)
30	Membrane, Eight-node Distorted Quadrilateral	(30)
31	Elastic Curved Pipe (Elbow)/Straight Beam Element	(31)
32	Plane Strain, Eight-node Distorted Quadrilateral, Herrmann or Mooney Material Formulation	(32)
33	Axisymmetric, Eight-node Distorted Quadrilateral, Herrmann or Mooney Material Formulation	(33)
34	Generalized Plane Strain, Eight-node Distorted Quadrilateral, Herrmann or Mooney Material Formulation	(34)
35	Three-dimensional, 20-node Brick, Herrmann or Mooney Material Formulation	(35)
36	Heat Transfer Element (three-dimensional link)	(36)
37	Heat Transfer Element (arbitrary planar triangle)	(37)
38	Heat Transfer Element (arbitrary axisymmetric triangle)	(38)
39	Heat Transfer Element (planar bilinear quadrilateral)	(39)
40	Heat Transfer Element (axisymmetric bilinear quadrilateral)	(40)
41	Heat Transfer Element (eight-node planar biquadratic quadrilateral)	(41)
42	Heat Transfer Element (eight-node axisymmetric biquadratic quadrilateral)	(42)
43	Heat Transfer Element (three-dimensional eight-node brick)	(43)
44	Heat Transfer Element (three-dimensional 20-node brick)	(44)
45	Curved Timoshenko Beam Element in a Plane	(45)
46	Plane Strain Rebar Element	(46)
47	Generalized Plane Strain Rebar Element	(47)
48	Axisymmetric Rebar Element	(48)
49	Triangular Shell Element	(49)
50	Triangular Heat Transfer Shell Element	(50)
51	Cable Element, Two-Node	(51)

Table 1-9 Element Library (Continued)

Element	Description	Code
52	Elastic Beam	(52)
53	Plane Stress, Eight-node Quadrilateral with Reduced Integration	(53)
54	Plane Strain, Eight-node Distorted Quadrilateral with Reduced Integration	(54)
55	Axisymmetric, Eight-node Distorted Quadrilateral with Reduced Integration	(55)
56	Generalized Plane Strain, Ten-node Distorted Quadrilateral with Reduced Integration	(56)
57	Three-dimensional, 20-node Brick with Reduced Integration	(57)
58	Plane Strain, Eight-node Distorted Quadrilateral for Incompressible Behavior with Reduced Integration	(58)
59	Axisymmetric, Eight-node Distorted Quadrilateral for Incompressible Behavior with Reduced Integration	(59)
60	Generalized Plane Strain, Ten-node Distorted Quadrilateral for Incompressible Behavior with Reduced Integration	(60)
61	Three-dimensional, 20-node Brick for Incompressible Behavior with Reduced Integration	(61)
62	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading	(62)
63	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading, Herrmann Formulation	(63)
64	Isoparametric, Three-node Truss Element	(64)
65	Heat Transfer Element, Three-node Link	(65)
66	8-node Axisymmetric with Twist, Herrmann Formulation	(66)
67	8-node Axisymmetric with Twist	(67)
68	Elastic, Four-node Shear Panel	(68)
69	Heat Transfer Element (Eight-node planar, biquadratic quadrilateral with Reduced Integration)	(69)
70	Heat Transfer Element (Eight-node, biquadratic quadrilateral with Reduced Integration)	(70)
71	Heat Transfer Element (three-dimensional 20-node brick with Reduced Integration)	(71)
72	Bilinear Constrained Shell	(72)
73	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading, with Reduced Integration	(73)
74	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading, Herrmann Formulation, with Reduced Integration	(74)
75	Bilinear Thick Shell	(75)

Table 1-9 Element Library (Continued)

Element	Description	Code
76	Thin-walled Beam in Three Dimensions Without Warping	(76)
77	Thin-walled Beam in Three Dimensions Including Warping	(77)
78	Thin-walled Beam in Three Dimensions Without Warping	(78)
79	Thin-walled Beam in Three Dimensions Including Warping	(79)
80	Incompressible Arbitrary Quadrilateral Plane Strain	(80)
81	Incompressible Generalized Plane Strain Quadrilateral	(81)
82	Incompressible Arbitrary Quadrilateral Axisymmetric Ring	(82)
83	Incompressible Axisymmetric Torsional Quadrilateral	(83)
84	Incompressible Three-dimensional Arbitrarily Distorted Cube	(84)
85	Bilinear Heat Transfer Shell	(85)
86	Curved Quadrilateral Heat Transfer Shell	(86)
87	Curved Axisymmetric Heat Transfer Shell	(87)
88	Linear Axisymmetric Heat Transfer Shell	(88)
89	Thick Curved Axisymmetric Shell	(89)
90	Thick Curved Axisymmetric Shell for Arbitrary Loading	(90)
91	Linear Plane Strain Semi-infinite Element	(91)
92	Linear Axisymmetric Semi-infinite Element	(92)
93	Quadratic Plane Strain Semi-infinite Element	(93)
94	Quadratic Axisymmetric Semi-infinite Element	(94)
95	Axisymmetric Quadrilateral with Bending Element	(95)
96	Axisymmetric Eight-node Distorted Quadrilateral with Bending	(96)
97	Double Friction and Gap Element Used with 95 or 96	(97)
98	Elastic Beam with Transverse Shear	(98)
101	Six-node Planar Semi-infinite Heat Transfer	(101)
102	Six-node Axisymmetric Semi-infinite Heat Transfer	(102)
103	Nine-node Planar Semi-infinite Heat Transfer	(103)

Table 1-9 Element Library (Continued)

Element	Description	Code
104	Nine-node Axisymmetric Semi-infinite Heat Transfer	(104)
105	Twelve-node Three-dimensional Semi-infinite Heat Transfer	(105)
106	Seven-node Three-dimensional Semi-infinite Heat Transfer	(106)
107	Twelve-node Three-dimensional Semi-infinite	(107)
108	Twenty-node Three-dimensional Semi-infinite	(108)
109	Eight-node Three-dimensional Magnetostatics	(109)
110	Twelve-node Three-dimensional Semi-infinite Magnetostatics	(110)
111	Four-node Quadrilateral Planar Electromagnetic	(111)
112	Four-node Quadrilateral Axisymmetric Electromagnetic	(112)
113	Eight-node Three-Dimensional Brick, Electromagnetic	(113)
114	Four-node Quadrilateral Plane Stress, Reduced Integration with Hourglass Control	(114)
115	Four-node Quadrilateral Plane Strain, Reduced Integration with Hourglass Control	(115)
116	Four-node Quadrilateral Axisymmetric, Reduced Integration with Hourglass Control	(116)
117	Eight-node Three-dimensional Brick, Reduced Integration with Hourglass Control	(117)
118	Incompressible 4+1-node, Quadrilateral, Plane Strain, Reduced Integration with Hourglass Control	(118)
119	Incompressible 4+1-node, Quadrilateral, Axisymmetric, Reduced Integration with Hourglass Control	(119)
120	Incompressible 8+1-node, Three-Dimensional Brick, Reduced Integration with Hourglass Control	(120)
121	Four-node, Heat Transfer Planar, Reduced Integration with Hourglass Control	(121)
122	Four-node, Heat Transfer Axisymmetric, Reduced Integration with Hourglass Control	(122)
123	Eight-node, Heat Transfer Three-dimensional Brick, Reduced Integration with Hourglass Control	(123)
124	Six-node, Plane Stress Triangle	(124)
125	Six-node, Plane Strain Triangle	(125)
126	Six-node, Axisymmetric Triangle	(126)

Table 1-9 Element Library (Continued)

Element	Description	Code
127	Ten-node, Tetrahedron	(127)
128	Incompressible, Six-Node Triangle	(128)
129	Incompressible, Six-Node Triangle	(129)
130	Incompressible, Ten-Node Tetrahedral	(130)
131	Six-node, Heat Transfer Planar	(131)
132	Six-node, Heat Transfer Axisymmetric	(132)
133	Ten-node, Heat Transfer Tetrahedral	(133)
134	Four-node, Tetrahedral	(134)
135	Four-node, Heat Transfer Tetrahedral	(135)
138	Three-node, Thin Shell	(138)
139	Four-node, Thin Shell	(139)
140	Four-Node, Thick Shell, Reduced Integration with Hourglass Control	(140)
142	Eight-node Axisymmetric Rebar with Twist	(142)
143	Four-node Plane Strain Rebar	(143)
144	Four-node Axisymmetric Rebar	(144)
145	Four-node Axisymmetric Rebar with Twist	(145)
146	Eight-node Brick Rebar	(146)
147	Four-node Membrane Rebar	(147)
148	Eight-node Membrane Rebar	(148)

Table 1-10 Summary of Element Properties

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
1	2	2	1	1	3	2	Yes
2	3	3	1	1	2	2	Yes
3	4	2	1	4	2	2	Yes
4	4	2	1	9	12	14	Yes
5	2	1	1	3	3	2	No
6	3	3	1	1	2	2	Yes
7	8	3	3	8	3	3	Yes
8	3	2	1	7	9	11	Yes
9	2	1	0	1	2/3	2/3	Yes
10	4	3	1	4	2	2	Yes
11	4	3	1	4	2	2	Yes
12	4	1	2	1	2/3	2/3	No
13	2	1	0	3	8	13	No
14	2	1	1	3	6	6	Yes
15	2	2	0	3	4	5	Yes
16	2	1	0	3	4	5	Yes
17	3	2	0	3	6	5	No
18	4	2	1	4	3	3	No
19	6	3	1	4	2	2	Yes
20	4	3	3	4	3	2	Yes
21	20	3	3	27	3	3	Yes
22	8	2	3	4	6	3	Yes
23	20	1	0	20	3	3	No
24	8	2	1	28	9	11	Yes
25	2	1	1	3	7	6	Yes
26	8	2	1	9	2	2	Yes
27	8	3	1	9	2	2	Yes
28	8	3	1	9	2	2	Yes
29	10	3	1	9	2	2	Yes

Table 1-10 Summary of Element Properties (Continued)

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
30	8	2	1	9	3	3	No
31	2	2	1	2	6	3	No
32	8	3	1	9	3	2	No
33	8	3	1	9	3	2	No
34	10	3	1	9	3	2	No
35	20	3	3	27	4	3	No
36	2	1	0	1	1	3	No
37	3	2	0	1	1	2	No
38	3	2	0	1	1	2	No
39	4	2	0	4	1	2	No
40	4	2	0	4	1	2	No
41	8	2	0	9	1	2	No
42	8	2	0	9	1	2	No
43	8	3	0	8	1	3	No
44	20	3	0	27	1	3	No
45	3	1	1	2	3	2	Yes
46	8	1	0	10	2	2	No
47	10	1	0	10	2	2	No
48	8	1	0	10	2	2	No
49	6	2	1	1	3	3	Yes
50	3	3	0	1	2	3	No
51	2	1	0	1	3	3	No
52	2	1	0	3	6	6	Yes
53	8	2	1	4	2	2	Yes
54	8	3	1	4	2	2	Yes
55	8	3	1	4	2	2	Yes
56	10	3	1	4	2	2	Yes
57	20	3	3	8	3	3	Yes
58	8	3	1	4	3	2	No

Table 1-10 Summary of Element Properties (Continued)

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
59	8	3	1	4	3	2	No
60	10	3	1	4	3	2	No
61	20	3	3	8	4	3	No
62	8	3	3	9	3	2	No
63	8	3	3	9	4	2	No
64	3	1	0	3	3	3	Yes
65	3	1	0	3	1	3	No
66	8	3	3	9	4	2	No
67	8	3	3	9	3	2	No
68	4	0	1	1	3	3	No
69	8	2	0	4	1	2	No
70	8	2	0	4	1	2	No
71	20	3	0	8	1	3	No
72	8	2	1	4	3	3	Yes
73	8	3	3	4	3	2	No
74	8	3	3	4	4	2	No
75	4	2	3	4	6	3	Yes
76	3	1	1	2	6	6	Yes
77	3	1	0	2	7	6	Yes
78	2	1	1	2	6	6	Yes
79	2	1	0	2	7	6	Yes
80	5	3	1	4	2	2	No
81	7	3	1	4	2	2	No
82	5	3	1	4	2	2	No
83	5	3	3	4	3	2	No
84	9	3	3	8	3	3	No
85	4	3	0	4	2/3	3	No
86	8	3	0	9	2/3	3	No
87	3	2	0	3	2/3	2	No

Table 1-10 Summary of Element Properties (Continued)

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
88	2	2	0	3	2/3	2	No
89	3	2	1	2	3	2	Yes
90	3	2	3	2	5	2	No
91	6	3	1	6	2	2	No
92	6	3	1	6	2	2	No
93	9	3	1	9	2	2	No
94	9	3	1	9	2	2	No
95	4	3	3	4	5	2	No
96	8	3	3	9	5	2	No
97	4	2	2	1	4	2	No
98	2	1	2	1	6	6	No
101	6	2	0	6	1	2	No
102	6	2	0	6	1	2	No
103	9	2	0	9	1	2	No
104	9	2	0	9	1	2	No
105	12	3	0	12	1	3	No
106	27	3	0	27	1	3	No
107	12	3	3	12	3	3	No
108	27	3	3	27	3	3	No
109	8	3	0	8	3	3	No
110	12	3	0	12	3	3	No
111	4	2	0	4	4	2	No
112	4	2	0	4	4	2	No
113	8	3	0	8	4	3	No
114	4	2	1	1	2	2	Yes
115	4	3	1	1	2	2	Yes
116	4	3	1	1	2	2	Yes
117	8	3	3	1	3	3	Yes
118	5	3	1	1	2	2	No

Table 1-10 Summary of Element Properties (Continued)

Element Type	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
119	5	3	1	1	2	2	No
120	9	3	3	1	3	3	No
121	4	2	0	1	1	2	No
122	4	2	0	1	1	2	No
123	8	3	0	1	1	3	No
124	6	2	1	3	2	2	Yes
125	6	3	1	3	2	2	Yes
126	6	3	1	3	2	2	Yes
127	10	3	3	4	3	3	Yes
128	6	3	1	3	3	2	No
129	6	3	1	3	3	2	No
130	10	3	3	4	4	3	No
131	6	2	0	3	1	2	No
132	6	2	0	3	1	2	No
133	10	3	0	4	1	3	No
134	4	3	3	1	3	3	Yes
135	4	3	0	1	1	3	No
138	3	2	1	1	6	3	Yes
139	4	2	1	4	6	3	Yes
140	4	2	3	1	6	3	Yes
142	8	1	0	10	3	2	No
143	4	1	0	10	2	2	No
144	4	1	0	10	2	2	No
145	4	1	0	10	3	2	No
146	8	1	0	20	3	3	No
147	4	1	0	20	3	3	No
148	8	1	0	20	3	3	No

Table 1-11 Overview of MARC Heat Transfer Element Types

Element Number	Heat Transfer	Joule Heating	Gap	Channel	Electrostatic	Magnetostatic	Radiation Cavity
36	yes*	yes	no**	no	no	no	no
37	yes	yes	no	no	yes	yes	no
38	yes	yes	no	no	yes	yes	no
39	yes	yes	yes	yes	yes	yes	no
40	yes	yes	yes	yes	yes	yes	yes
41	yes	yes	yes	yes	yes	yes	no
42	yes	yes	yes	yes	yes	yes	yes
43	yes	yes	yes	yes	yes	no	no
44	yes	yes	yes	yes	yes	no	no
50	yes	no	no	no	yes	no	no
65	yes	yes	no	no	no	no	no
69	yes	yes	yes	yes	yes	yes	no
70	yes	yes	yes	yes	yes	yes	yes
71	yes	yes	yes	yes	yes	no	no
85	yes	no	no	no	yes	no	no
86	yes	no	no	no	yes	no	no
87	yes	no	no	no	yes	no	no
88	yes	no	no	no	yes	no	no
101	yes	no	no	no	yes	yes	no
102	yes	no	no	no	yes	yes	no
103	yes	no	no	no	yes	yes	no
104	yes	no	no	no	yes	yes	no
105	yes	no	no	no	yes	no	no
106	yes	no	no	no	yes	no	no
109	no	no	no	no	no	yes	no

Table 1-11 Overview of MARC Heat Transfer Element Types (Continued)

Element Number	Heat Transfer	Joule Heating	Gap	Channel	Electrostatic	Magnetostatic	Radiation Cavity
110	no	no	no	no	no	yes	no
121	yes	yes	yes	yes	yes	yes	no
122	yes	yes	yes	yes	yes	yes	yes
123	yes	yes	yes	yes	yes	no	no
131	yes	yes	no	no	yes	yes	no
132	yes	yes	no	no	yes	yes	no
133	yes	yes	no	no	yes	no	no
135	yes	yes	no	no	yes	no	no

*yes – capabilities are available.**no – capabilities are not available

Follow Force Stiffness Contribution

When activating the FOLLOW FOR parameter, the distributed loads are calculated based upon the current deformed configuration. It is possible to activate an additional contribution which goes into the stiffness matrix. This improves the convergence. This capability is available for element types 3, 7, 10, 11, 18, 72, 75, 80, 82, 84, 114, 115, 116, 117, 118, 119, 120, 139, and 140.

Explicit Dynamics

The explicit dynamics formulation IDYN=5 model is restricted to the following elements:

2, 3, 5, 6, 7, 9, 11, 18, 19, 20, 52, 64, 75, 98, 114, 115, 116, 117, 118, 119, 120, 138, 139, and 140

When using this formulation, the mass matrix is defined semi-analytically; i.e., no numerical integration is performed. In addition, a quick method is used to calculate the stability limit associated with each element. For these reasons, this capability has been limited to the elements mentioned above.

Adaptive Mesh Refinement

The MARC program has a capability to perform adaptive mesh refinement to improve the accuracy of the solution. This capability is invoked by using the ADAPTIVE parameter and model definition option. The adaptive meshing is available for the following 2D and 3D elements:

> 2, 3, 6, 7, 10, 11, 18, 19, 20, 37, 38, 39, 40, 43, 75, 80, 81, 82, 83, 84, 111, 112, 113, 114, 115, 116, 117, 118, 119, 120, 121, 122, 123, 139, and 140

The adaptive procedure works by subdividing the existing element, based upon the error criteria specified. In two-dimensional analyses, triangles are divided into four triangles and quadrilaterals are divided into four quadrilaterals, while in three-dimensional analyses, tetrahedrals are subdivided into eight tetrahedrals and hexahedrals are subdivided into eight hexahedrals.

2

MARC Element Classifications

MARC contains a large number of element types. Functionally, these are used to model trusses, beams, plates, shells, plane stress, plane strain and general three-dimensional continua. Where appropriate, there are corresponding heat transfer elements which are found in Table 1-9. In Chapter 2, these elements are separated into classes based on their respective node number, interpolation functions, and integration schemes. In Chapter 3, these elements are detailed in numerical order.

Class 1

Elements 1 and 5

This class consists of two-noded elements with linear interpolation along the length and cubic interpolation normal to the length.

There is a local coordinate system $\hat{x}$, which is in the direction from node 1 to node 2 as shown below, associated with the element.

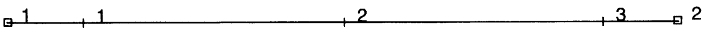
The displacement assumptions are as follows:

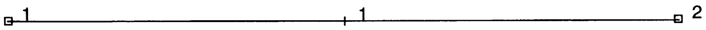
$$u^T = a_1 + a\,\hat{x}$$

$$u^N = a_3 + a_4\hat{x} + a_5\hat{x}^2 + a_6\hat{x}^3$$

where u^T and u^N are displacements tangential and normal to $\hat{x}$ respectively.

There are three degrees of freedom associated with each of the nodes, two global displacements and one rotation. The integration along the length of the element is performed with one or three Gaussian integration points, whose spacing is shown below.





This element uses the same integration points along the length to form equivalent nodal loads and the mass matrix.

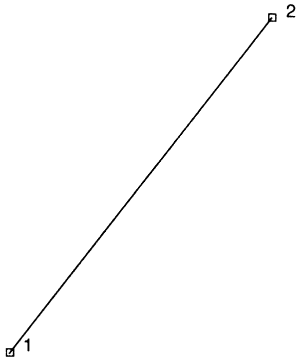
The stress-strain relation is integrated through the thickness by Simpson's rule, using the number of points defined in the shell section parameter card (11 by default). The first layer is on the positive normal side which is obtained by rotating the $\hat{x}$ axis 90 degrees counterclockwise.

The elements in class 11 (elements 15 and 16) are preferable elements to use.

Class 2

Element 9 and 36

This class consists of two-noded elements with linear interpolation along the length as shown below.

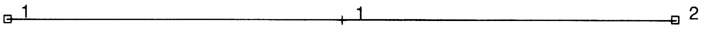


There is only stiffness associated along the length of the element. The interpolation function can be expressed as follows:

$$\psi = \psi_1 \xi + \psi_2 (1 - \xi)$$

where ψ_1, ψ_2 are the values of the function at the end nodes and ξ is the normalized coordinate $(0 \le \xi \le 1)$.

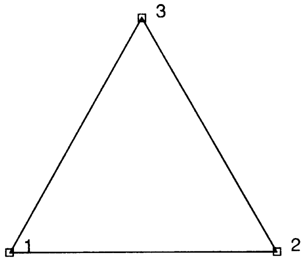
Element 9 has three degrees of freedom per node, and element 36 has one degree of freedom per node. There is a single integration point at the centroid the element as shown below.



Class 3

Elements 2, 6, 37, 38, and 50

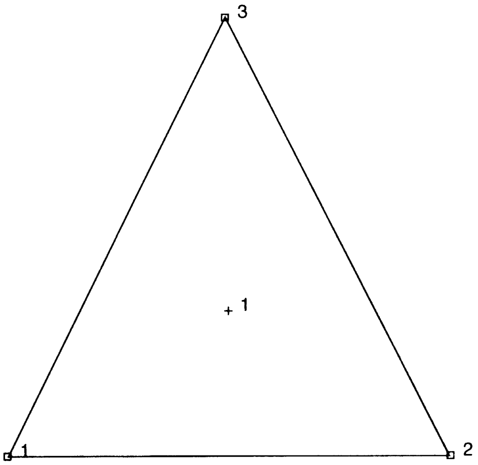
This class consists of three-noded triangular elements with linear interpolation functions as shown below.



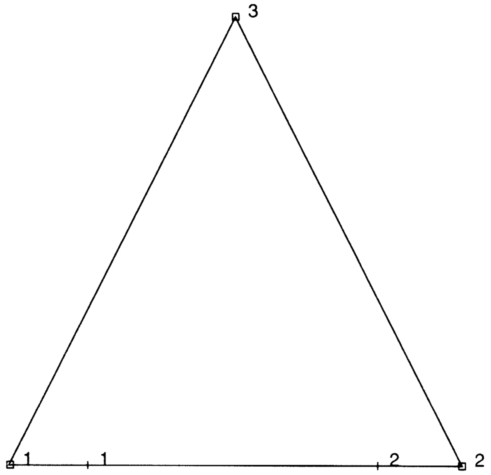
The node numbering must be counterclockwise in the plane of the element. The function is assumed to be expressed in the form

$$\psi = a_0 + a_1 x + a_2 y$$

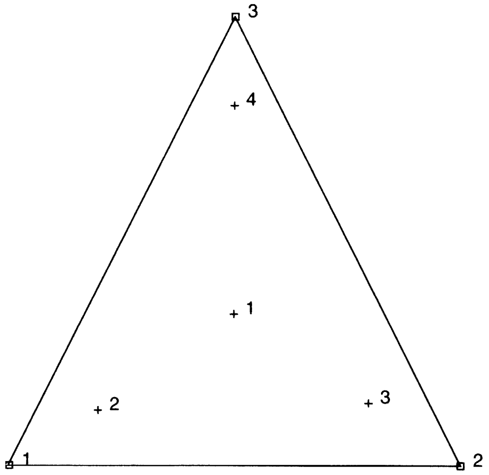
For elements 2 and 6, there are two degrees of freedom. For elements 37 and 38, there is one degree of freedom. For heat transfer shell element 50, there are either two or three degrees of freedom. There is a single integration point at the centroid of the element as shown below.



For distributed surface pressures (flux), two integration points, as shown below, are used.



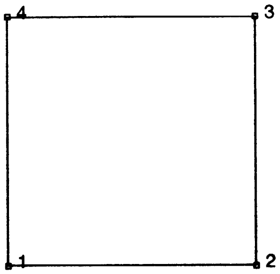
For volumetric forces (flux), four integration points, as shown below, are used.



Class 4

Elements 3, 10, 11, 18, 19, 20, 39, 40, 80, 81, 82, 83, 111, and 112

This class consists of four-noded isoparametric elements with bilinear interpolation. The element node numbering must be given in counterclockwise direction following the right-hand rule as shown below.



The element is formed by mapping from the x-y (z-r) plane to the ξ, η plane.

Both the mapping and the function assumption take the form:

$$x = a_0 + a_1\xi + a_2\eta + a_3\xi\eta$$
$$\psi = b_0 + b_1\xi + b_2\eta + b_3\xi\eta$$

Either the coordinate or function can be expressed in terms of the nodal quantities by the interpolation functions.

$$x = \sum_{i=1}^{4} x_i\phi_i \qquad \text{where}$$

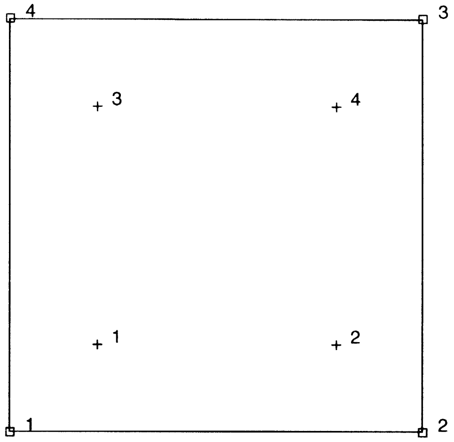
$$\phi_1 = \frac{1}{4}(1-\xi)(1-\eta)$$

$$\phi_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

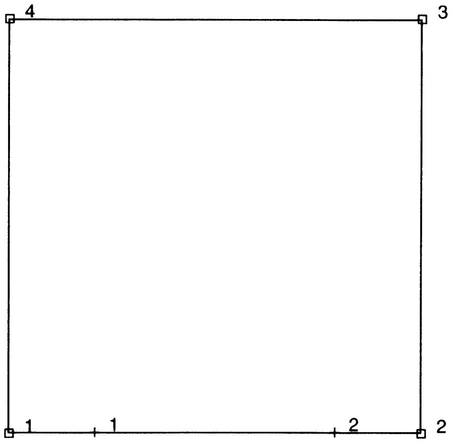
$$\phi_3 = \frac{1}{4}(1+\xi)(1+\eta)$$

$$\phi_4 = \frac{1}{4}(1-\xi)(1+\eta)$$

There are three degrees of freedom associated with each node for elements 18 and 20. There are two degrees of freedom per node for elements 3, 10, 11, and 19. There is one degree of freedom for elements 39 and 40. These elements use four-point Gaussian integration as shown below.



For distributed surface pressures, two-point Gaussian integration, as shown below, is used.



For elements 10, 11, 19, and 20, an optional integration scheme may be used which imposes a constant dilatational strain on the element. This is equivalent to a selective integration where the four Gaussian points are used for the deviatoric contribution of strain and the centroid for the dilatation contribution. This is flagged using the second parameter of the GEOMETRY option.

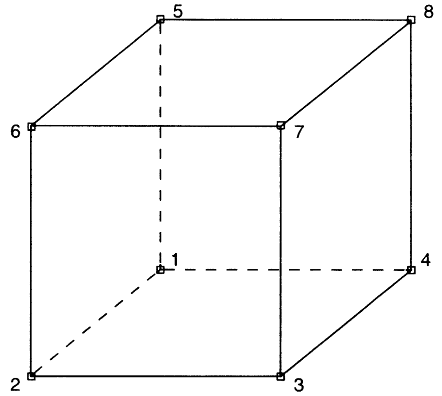
For elements 3 and 11, an optional assumed strain interpolation formulation is available. This significantly improves the behavior of this element in bending. This is flagged using the third parameter of the GEOMETRY option.

For elements 80, 81, 82, and 83, there is one extra node with a single degree of freedom (pressure). These elements use a mixed formulation for incompressible analysis.

Class 5

Elements 7, 43, 84, and 113

This class consists of eight-noded, isoparametric, three-dimensional brick elements with trilinear interpolation. The node numbering must follow the rules as shown and given below.



Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is on the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4. The element is based on the following type of displacement assumption and mapping from the x-y-z space into a cube in the ξ, η, ζ space.

$$x = a_0 + a_1 \xi + a_2 \eta + a_3 \xi + a_4 \xi \eta + a_5 \zeta \xi + a_6 \xi \zeta + a_7 \xi \eta \zeta$$

$$\psi = b_0 + b_1 \xi + b_2 \eta + b_3 \xi + b_4 \xi \eta + b_5 \eta \zeta + b_6 \xi \zeta + b_7 \xi \eta \zeta$$

Either the coordinate or function can be expressed in terms of the nodal quantities by the integration functions.

$$x = \sum_{i=1}^{8} x_i \phi_i$$

where

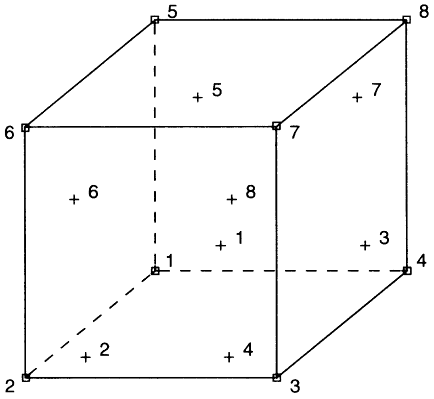
$$\phi_1 = \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta) \qquad \phi_5 = \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta)$$

$$\phi_2 = \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta) \qquad \phi_6 = \frac{1}{8}(1+\xi)(1-\eta)(1+\zeta)$$

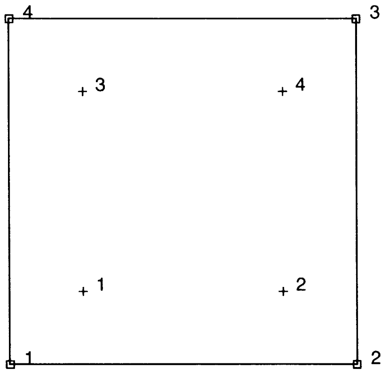
$$\phi_3 = \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta) \qquad \phi_7 = \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta)$$

$$\varphi_4 = \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta) \qquad \phi_8 = \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta)$$

There are three degrees of freedom associated with each node for element 7 and one degree of freedom for each node for element 43. These elements use eight-point Gaussian integration as shown below.



For distributed pressure (flux) loads, four-point Gaussian integration, as shown below, is used.



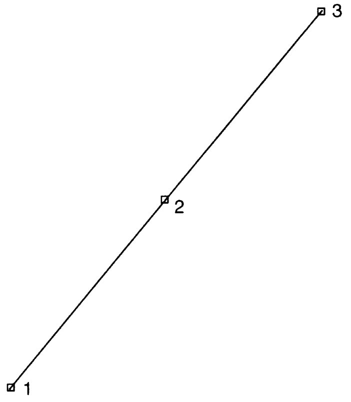
An optional integration scheme may be used for element 7 which imposes a constant dilatational strain on the element. This is equivalent to a selective integration where the eight Gaussian points are used for the deviatoric contribution of strain and the centroid for the dilatation contribution. This is flagged using the second parameter of the GEOMETRY option. For element 7, an optional assumed strain interpolation formulation is available. This significantly improves the behavior of this element in bending. This is flagged using the third parameter of the GEOMETRY option.

For element 84, there is one extra node with a single degree of freedom (pressure). This element uses a mixed formulation for incompressible analysis.

Class 6

Elements 64 and 65

This class consists of three-noded isoparametric links with quadratic interpolation along the length as shown below.



The node numbering is as shown. The interpolation function is such that element is parabolic. The element is based on the following displacement assumption, and mapping from the x-y-z space into a straight line.

$$x = a_0 + a_1 \zeta + a_2 \zeta^2$$

Either the coordinates or function can be expressed in terms of the nodal quantities by the interpolation functions

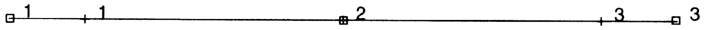
$$x = \sum_{i=1}^{3} x_i \phi_i \qquad \text{where}$$

$$\phi_1 = \frac{1}{2} \xi (\xi - 1)$$

$$\phi_2 = \frac{1}{2} \xi (1 + \xi)$$

$$\phi_3 = (1 - \xi)(1 + \xi)$$

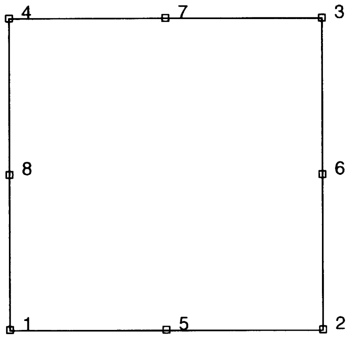
There are three degrees of freedom associated with each node for element 64. There is one degree of freedom per node for element 65. These elements use three-point Gaussian integration as shown below.



Class 7

Elements 26, 27, 28, 29, 30, 32, 33, 34, 41, 42, 47, 48, 62, 63, 66, and 67

This class consists of eight-noded isoparametric distorted quadrilateral elements with biquadratic interpolation and full integration. The node numbering must be counterclockwise in the plane as shown below.



The four corner nodes come first and then the midside nodes with node 5 on the 1-2 edge, etc. The interpolation function is such that each edge has parabolic variation along itself. The element is based on the following displacement assumption, and mapping from the x-y (z-r) space to a square in the ζ, η space.

$$x = a_0 + a_1 \xi + a_2 \eta + a_3 \xi^2 + a_4 \xi \eta + a_5 \eta^2 + a_6 \xi^2 \eta + a_7 \xi \eta^2$$

Either the coordinates or function can be expressed in terms of the nodal quantities by the interpolation functions.

$$x = \sum_{i=1}^{8} x_i \phi_i$$

where

$$\phi_1 = \frac{1}{4}(1-\xi)(1-\eta)(-1-\xi-\eta)$$

$$\phi_2 = \frac{1}{4}(1+\xi)(1-\eta)(-1+\xi-\eta)$$

$$\phi_3 = \frac{1}{4}(1+\xi)(1+\eta)(-1+\xi+\eta)$$

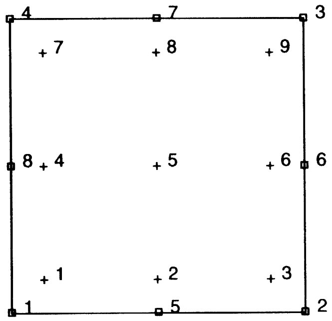
$$\phi_5 = \frac{1}{2}(1-\xi^2)(1-\eta)$$

$$\phi_6 = \frac{1}{2}(1+\xi)(1-\eta^2)$$

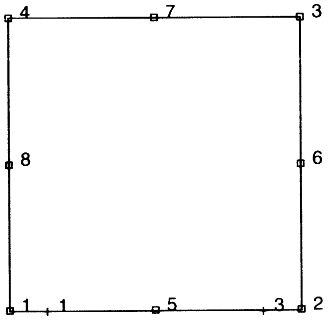
$$\phi_7 = \frac{1}{2}(1-\xi^2)(1+\eta)$$

$$\phi_4 = \frac{1}{4}(1-\xi)(1+\eta)(-1-\xi+\eta) \qquad \phi_8 = \frac{1}{2}(1-\xi)(1-\eta^2)$$

These elements use nine-point Gaussian integration as shown below for the calculation of the stiffness and mass matrix and evaluation of equivalent volumetric loads.



For distributed surface pressures (flux), three-point integration is used as shown below.



Elements 29 and 34 have two additional nodes which are used to formulate a generalized plane strain condition. These additional nodes are shared between all the elements in the mesh.

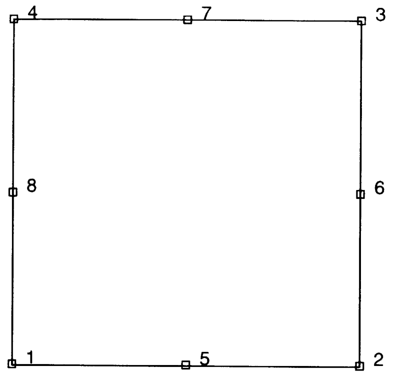
Elements 62 and 63 have been modified to be used in conjunction with the Fourier option. This allows nonaxisymmetric loads to be applied on an axisymmetric object. These elements can only be used for linear elastic static behavior.

Elements 32, 33, 34, and 63 have been modified so that they may be used for problems concerning incompressible or nearly incompressible materials. These elements use an extension of the Herrmann variational principle. They have an additional degree of freedom at the corner nodes, which represents the hydrostatic pressure. This function is interpolated linearly through the element.

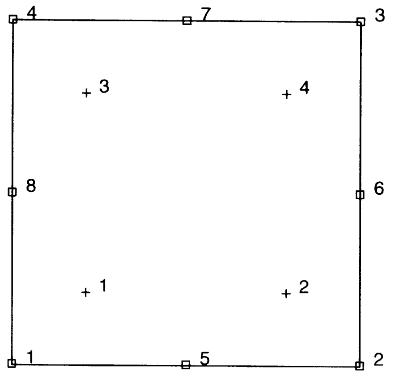
Class 8

Elements 53, 54, 55, 56, 58, 59, 60, 69, 70, 73, and 74

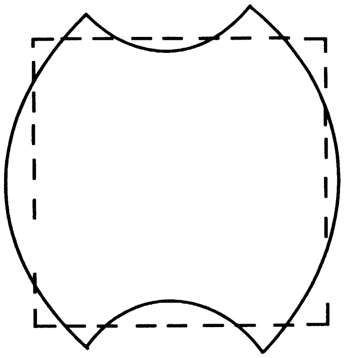
This class consists of eight-noded isoparametric distorted quadrilateral elements with biquadratic interpolation and reduced integration (see illustration below).



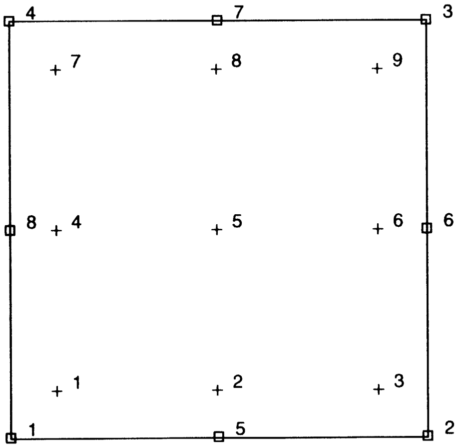
These elements have the same node numbering and interpolation functions as the elements in Class 7. The only difference is that these elements use a reduced integration technique in calculation of the stiffness matrix. The stiffness matrix is calculated using four-point Gaussian integration as shown below.



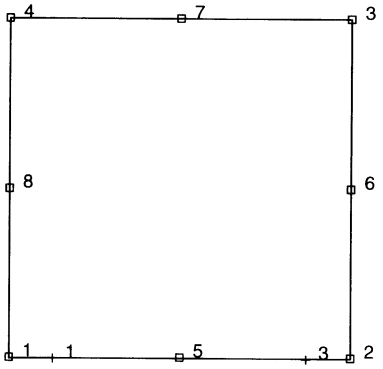
Using this method, quadratic functions as used in the interpolation functions are not integrated exactly. The contributions of the higher order terms in the deformation field are neglected. This results in singular modes; that is, deformations which do not contribute to the strain energy in the element. These elements have one such mode, as shown below.



The integration of the mass matrix and the consistent volumetric loads is done using nine-point Gaussian integration as shown below.



The surface distributed loads is integrated using three-point Gaussian integration as shown below.



Elements 56 and 60 have two additional nodes which are used to formulate a generalized plane strain condition. These additional nodes are shared between all the elements in the mesh.

Elements 73 and 74 have been modified to be used in conjunction with the Fourier option. This allows nonaxisymmetric loads to be applied on an axisymmetric object. These elements can only be used for linear elastic static behavior.

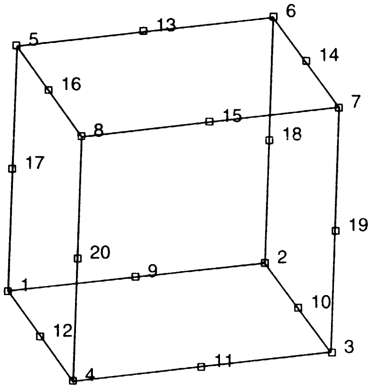
Elements 58, 59, 60, and 74 have been modified so that they may be used for problems concerning incompressible or nearly incompressible materials. These elements use an extension of the Herrmann principle. They have an additional degree of freedom at the corner nodes which represents the hydrostatic pressure. This function is interpolated linearly through the element.

Class 9

Elements 21, 35, and 44

This class consists of 20-node isoparametric distorted three-dimensional brick elements with full integration. The convention for the node numbering is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in counterclockwise order when viewed from inside the element. Nodes 5, 6, 7, 8 are corners of the opposite face; that is node 5 shares an edge with 1, node 6 with 2, etc. Nodes 9, 10, 11, 12 are the midside nodes of the 1, 2, 3, 4 face; node 9 between 1 and 2, node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midside nodes on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge, node 18 of the 2-6 edge, etc. This is shown below. Each edge forms a parabola.

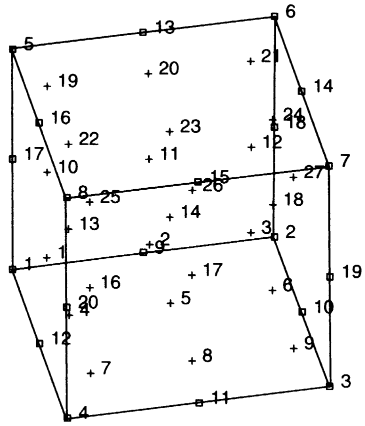


The element is based on the following displacement assumption, and mapping from the x-y-z space to a cube in ξ, η, ζ space.

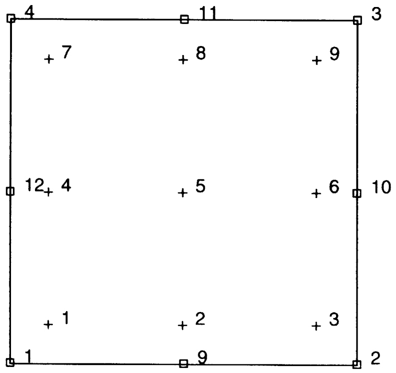
$$\begin{aligned} x = {} & a_0 + a_1 \xi + a_2 \eta + a_3 \zeta + a_4 \xi^2 + a_5 \xi \eta + a_6 \eta^2 + a_7 \eta \zeta + \\ & a_8 \zeta^2 + a_9 \xi \zeta + a_{10} \zeta^2 \eta + a_{11} \zeta \eta^2 + a_{12} \eta^2 \zeta + a_{13} \eta \zeta^2 + \\ & a_{14} \xi^2 \zeta + a_{15} \xi \zeta^2 + a_{16} \xi \eta \zeta + a_{17} \xi^2 \eta \zeta + a_{18} \xi \eta^2 \zeta + \\ & a_{19} \xi \eta \zeta^2 \end{aligned}$$

The resulting interpolation function is triquadratic.

These elements use 27-point Gaussian integration as shown below.



These integration points can be considered to be made up of three planes perpendicular to the ζ axis (i.e., parallel to the 1, 2, 3, 4, 9, 10, 11, and 12 face such that integration points 1-9 are closest to the 1, 2, 3, 4 face; integration points 19-27 are closest to the 5, 6, 7, 8 face; and integration points 10-18 are in between). Integration point 14 is in the centroid of the element. The calculation of the mass matrix of equivalent volumetric force uses the same integration scheme. Surface pressures are integrated using a nine-point Gaussian integration as shown below.

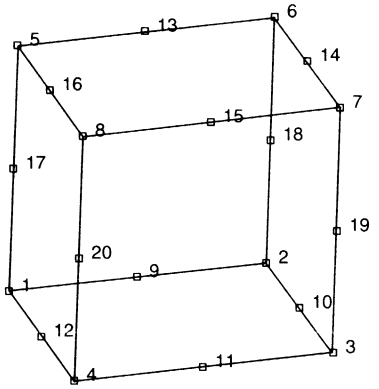


Element 35 has been modified so that it may be used for problems concerning incompressible or nearly incompressible materials. This element uses an extension of the Herrmann variational principle. It has an additional degree of freedom at the corner nodes which represents the hydrostatic pressure. This function is interpolated linearly throughout the element.

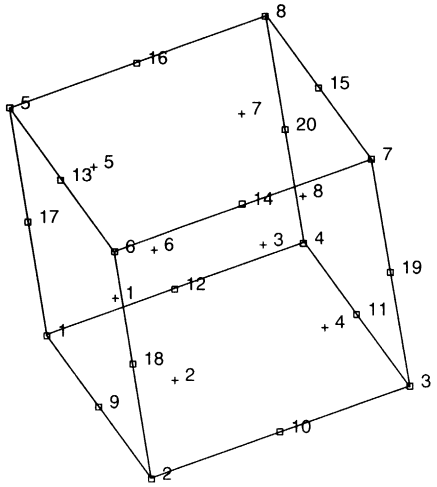
■ Class 10

Elements 57, 61, and 71

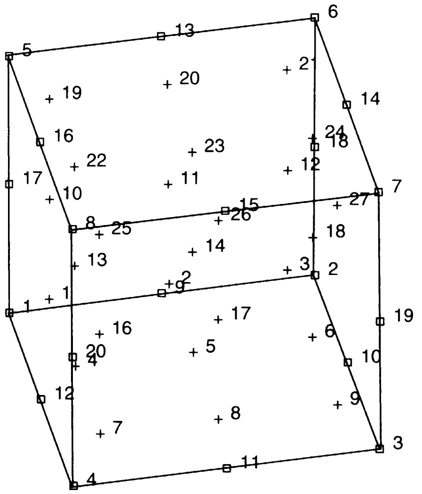
This class consists of 20-noded isoparametric, distorted, three-dimensional brick elements with reduced integration using triquadratic interpolation functions.These elements have the same node numbering and interpolation functions as the elements in Class 9. The element nodes are shown below.



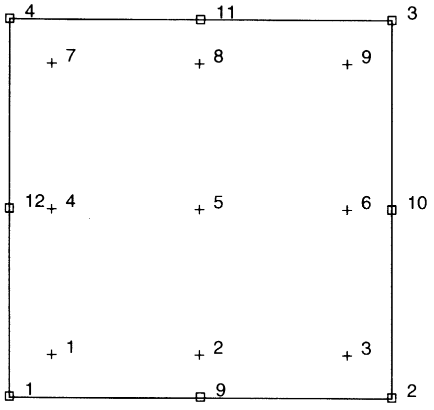
The only difference is that these elements use a reduced integration technique in calculation of the stiffness matrix. The stiffness matrix is calculated using eight-point Gaussian integration as shown below.



Using reduced integration, quadratic functions as used in the interpolating functions are not integrated exactly. The contribution of the higher order terms in the deformation field are neglected. This results in singular modes; that is, deformations which do not contribute to the strain energy in the element. These elements have six such modes. The integration of the mass matrix and the consistent volumetric loads is done using 27-point Gaussian integration as shown below.



The surface distributed loads are integrated using nine-point Gaussian integration as shown below.



Element 61 has been modified so that it may be used for problems concerning incompressible or nearly incompressible materials. This element uses an extension of the Herrmann variational principle. It has an additional degree of freedom at the corner nodes which represents the hydrostatic pressure. This function is interpolated linearly throughout the element.

Class 11

Elements 15, 16, and 17

This class consists of two-noded axisymmetric curved shell, curved beam and pipe bend with cubic interpolation functions. These elements are shown below.

They use a Hermite cubic interpolation function to express the nodal displacements. In this case,

$$u(\xi) = H_{01}(\xi)\,u_1 + H_{02}(\xi)\,u_2 + H_{11}(\xi)u'_1 + H_{12}(\xi)u'_2$$

where u is the value of the function at the nodes and u' is its first derivative. The Hermite polynomials are as follows:

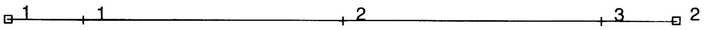
$$H_{01}(\xi) = (2+\xi)(1-\xi)^2/4$$

$$H_{02}(\xi) = (2-\xi)(1+\xi)^2/4$$

$$H_{11}(\xi) = (1+\xi)(1-\xi)^2/4$$

$$H_{12}(\xi) = -(1-\xi)(1+\xi)^2/4$$

These elements use three-point Gaussian integration as shown below.

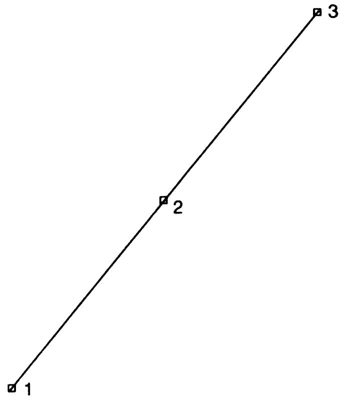


The stress-strain law is integrated through the thickness using Simpson's rule. The elements have two global displacement and two derivative degrees of freedom at each node.

Class 12

Element 45

This class contains a three-noded curved Timoshenko beam in a plane which allows transverse shear as well as axial straining as shown below.



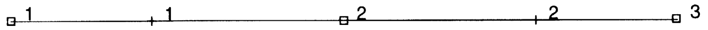
It uses quadratic interpolation for the beam axis, together with quadratic polynomial interpolation for the cross-section rotation. The center node (node 2) has been added to allow this quadratic representation of the cross-section rotation. Hence:

$$u\ (\xi) = \frac{1}{2}\ \xi\ (\xi - 1)\ u_1 + (1 - \xi)(1 + \xi)\ u_2 + \frac{1}{2}\ \xi\ (1 + \xi)\ u_3$$

and

$$\phi\ (\xi) = \frac{1}{2}\ \xi\ (\xi - 1)\ \phi_1 + (1 - \xi)(1 + \xi)\ \phi_2 + \frac{1}{2}\ \xi\ (1 + \xi)\ \phi_3$$

The interpolation of u, v, and ϕ are uncoupled. The calculation of the generalized strain terms leads to coupling in the shear calculation. This element is integrated using two-point Gaussian integration along the beam axis as shown below.

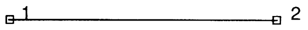


The stress-strain law is integrated through the thickness using Simpson's rule.

Class 13

Elements 13, 14, 25, and 52

This class consists of two-noded beam elements written in the global x-y-z space as shown below.



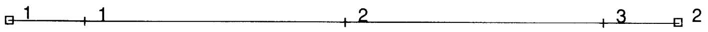
Each one of these elements has a different function summarized as follows:

Element 13	Open section thin-walled beam
Element 14	Closed section thin-walled beam
Element 25	Closed section thin-walled beam
Element 52	Elastic beam

The interpolation functions can be summarized as follows:

	Along the Axis	Normal to Axis	Twist
13	Cubic	Cubic	Cubic
14	Linear	Cubic	Linear
25	Cubic	Cubic	Linear
52	Linear	Cubic	Linear

These elements are integrated using three-point Gaussian integration along the beam axis as shown below.

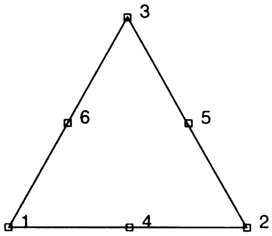


The stress-strain law is integrated using Simpson's rule through the cross section of the element. Elements 14 and 25 have an annular default cross section. The default values of the annular pipe may be set on the GEOMETRY option. General cross section geometry for beam elements may be defined using the BEAM SECT parameter.

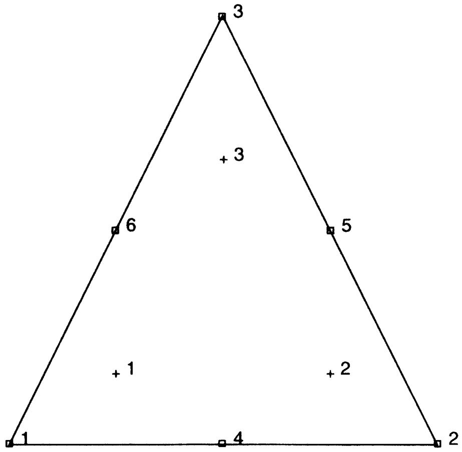
Class 14

Elements 124, 125, 126, 128, 129, 131, and 132

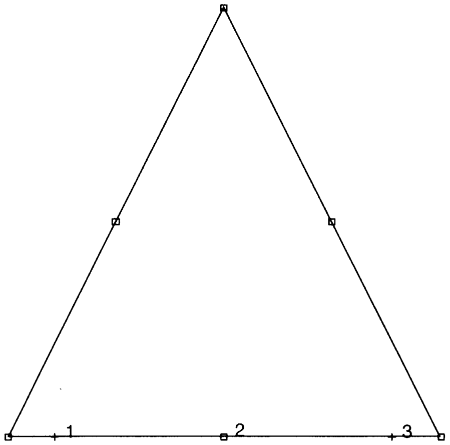
This class consists of six-noded triangular elements with quadratic interpolation functions as shown below.



The node numbering must be counterclockwise in the plane of the element. The functions are expressed with respect to area coordinate systems. The stiffness matrix is integrated using three integration points as shown below.



The distributed loads are integrated using three point integration along the edge as shown below.

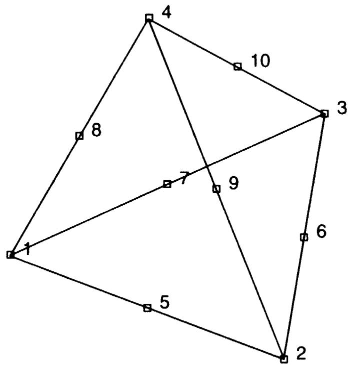


When using the Herrmann formulation, elements 128 or 129, the corner points contain an additional degree of freedom.

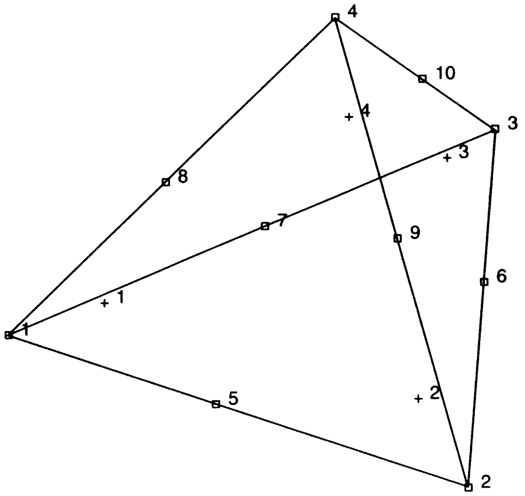
Class 15

Elements 127, 130, and 133

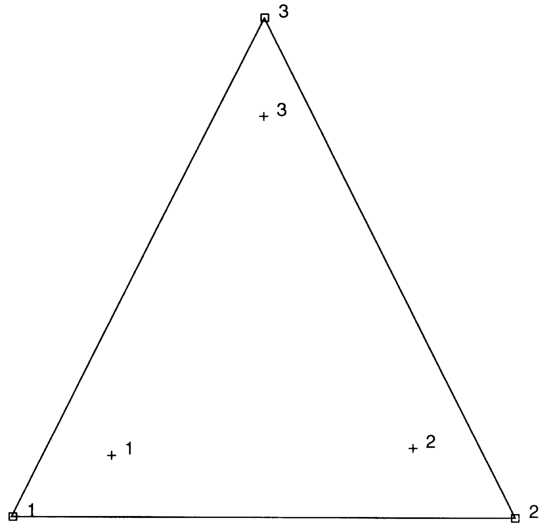
This class consists of ten-noded tetrahedral elements with quadratic interpolation functions as shown below.



The interpolation functions are expressed with respect to area coordinate systems. The stiffness matrix is integrated using four integration points as shown below.



The distributed loads on a face are integrated using three integration points as shown below.

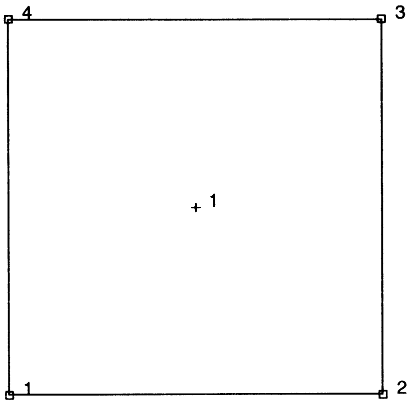


When using the Herrmann formulation, element 130, the corner points contain an additional degree of freedom.

Class 16

Elements 114, 115, 116, 118, 119, 121, and 122

This class consists of four-noded isoparametric planar elements written with respect to the natural coordinate system of the element. They are reduced integration elements, but an additional contribution has been made to the stiffness matrix to eliminate the problems associated with hourglass modes. The node numbering must be counterclockwise in the plane, as shown below, with a single integration point for the stiffness matrix.



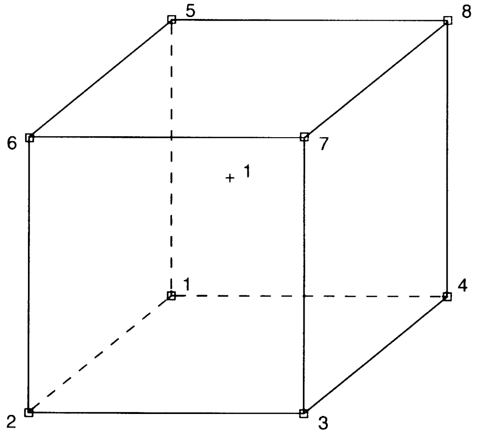
The element uses conventional 2 x 2 integration for the mass matrix and two-point integration along the edge for distributed loads.

For elements 118 and 119, there is one extra node with a single degree of freedom (pressure). These elements use a mixed formulation for incompressible analysis.

Class 17

Elements 117, 120, and 123

This class consists of eight-noded isoparametric elements written with respect to the natural coordinate system of the element. They are reduced integration elements, but an additional contribution has been made to the stiffness matrix to eliminate the problems with hourglass modes. Nodes 1, 2, 3, and 4 are corner nodes of one face given in counterclockwise order when viewed from inside the element. As shown below, node 5 is on the same edge as node 1; node 6 as node 2; node 7 as node 3; and node 8 as node 4.



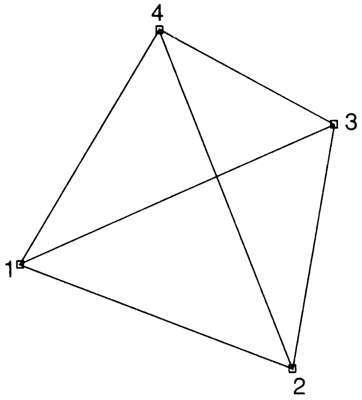
A single integration point is used for the stiffness matrix. The element uses conventional 2 x 2 x 2 integration for the mass matrix and 2 x 2 integration on a face for distributed loads.

For element 120, there is an additional node with a single degree of freedom (pressure). This element uses a mixed formulation for incompressible analysis.

Class 18

Elements 134 and 135

This class consists of four-noded tetrahedral elements with linear interpolation functions as shown below:



The interpolation functions are expressed with respect to the area coordinate system. The stiffness matrix is integrated using a single integration point at the centroid. The distributed load on a face is integrated using a single integration point at the centroid of the face.

Note that this element gives very poor behavior when used for incompressible or nearly incompressible behavior, such as plasticity.

3

Element Library

The remainder of this volume describes each element type. The geometric information that the user is required to input is described here. The output associated with each element type is also discussed.

Element 1

Straight Axisymmetric Shell

This is a two-node, axisymmetric, thick-shell element with a linear displacement assumption based on the global displacements and rotation. The strain-displacement relationships used are suitable for large displacements and large membrane strains. One-point Gaussian integration is used for the mass and pressure determination. All constitutive models can be used with this element.

Element types 15 and 89 are more accurate elements.

Quick Reference

Type 1

Axisymmetric, straight, thick-shell element.

Connectivity

Two nodes per element (see Figure 3-1).

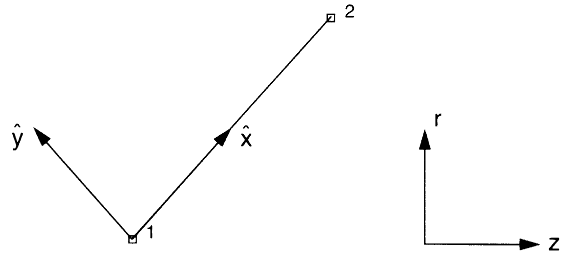


Figure 3-1 Beam-Column Element

Geometry

Constant thickness along length of the element. Thickness of the element is stored in the first data field (EGEOM1).

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)
3 = ϕ = right hand rotation

Tractions

Distributed loads. Selected with IBODY as follows:

Load Type	Description
0	Uniform pressure.
1	Nonuniform pressure.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the z, r-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure assumed positive in the direction opposite to the normal.

Output of Strains

Output of strains at the center line of the element is as follows:

1 = meridional membrane
2 = circumferential membrane
3 = transverse shear
4 = meridional curvature
5 = circumferential curvature

Output of Stresses

The stresses are given at the center line of the shell element and at the integration points through the thickness as follows:

1 = meridional stress
2 = circumferential stress
3 = transverse shear

The integration point numbering sequence progresses in the positive local $\hat{y}$ directions.

Transformation

The displacement degrees of freedom may be transformed to local directions.

Tying

Standard type 23 with elements 2 and 10.

Output Points

Centroid of the element.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in meridional and circumferential direction. Thickness will be updated.

Note: Shell theory only applies if strain variation through the thickness is small.

Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotations remain small within a load step.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 88. See Element 88 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 2

Axisymmetric Triangular Ring

Element type 2 is a three-node, isoparametric, triangular element. It is written for axisymmetric applications and uses bilinear interpolation functions. The strains are constant throughout the element and this results in a poor representation of shear behavior.

In general, one needs more of these lower order elements than the higher order elements such as type 124. Hence, use a fine mesh.

The stiffness of this element is formed using one-point integration at the centroid.

This element should not be used in cases where incompressible or nearly incompressible behavior occurs because it will lock. This includes plasticity and creep. The cross-triangle approach can be used to reduce the shortcomings of this element.

Quick Reference

Type 2

Axisymmetric triangular ring element.

Connectivity

Three nodes per element (see Figure 3-2). Node numbering must counterclockwise.

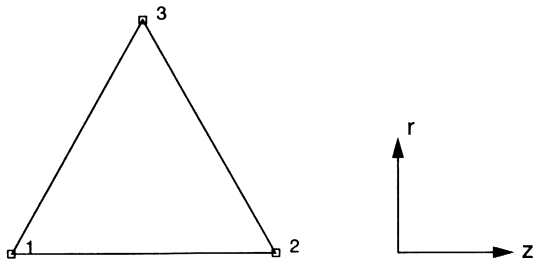


Figure 3-2 Linear Displacement Triangular Ring Element

Geometry

Not required for this element.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through subroutine FORCEM.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Concentrated loads applied at the nodes must be the value of the load integrated around the circumference.

Output of Strains

Output of strains at the centroid of the element is as follows:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{zr}$

Output of Stresses

Output of stresses is also at the centroid of the element and follows the same scheme as **Strains**.

Transformation

Two global degrees of freedom may be transformed to local coordinates. In this case, the corresponding applied nodal loads should also be in the local direction.

Tying

May be tied to axisymmetric shell type 1 by typing type 23.

Output Points

Output is only available at the centroid. Element mesh must generally be fine for accuracy.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain in global coordinate directions. “Crossed triangle” approach recommended.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 38. See Element 38 for a description of the conventions used for entering the flux and film data for this element.

Element 3

Plane Stress Quadrilateral

Element 3 is a four-node, isoparametric, arbitrary quadrilateral written for plane stress applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics may be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, one needs more of these lower-order elements than the higher-order elements such as 26 or 53. Hence, use a fine mesh.

This element is preferred over higher order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive models can be used with this element.

Note: To improve the bending characteristics of the element, the interpolation functions are modified for the assumed strain formulation.

Quick Reference

Type 3

Plane stress quadrilateral.

Connectivity

Node numbering must be counterclockwise (see Figure 3-3).

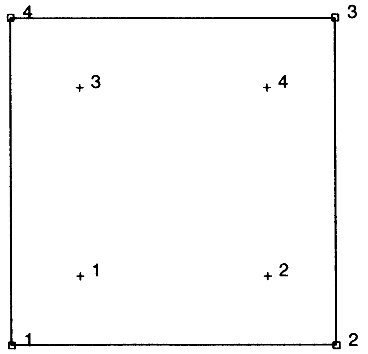


Figure 3-3 Plane Stress Quadrilateral

Geometry

The thickness is stored in the first data field (EGEOM1). Default thickness is one.

The second field is not used.

In the third field, a one activates the assumed strain formulation.

Coordinates

Two global coordinates x and y.

Degrees of Freedom

1 = u (displacement in the global x direction)
2 = v (displacement in the global y direction)

Distributed Loads

Load types for distributed loads are defined as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.

Load Type	Description
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
* 20	Uniform shear force on side 1-2 (positive from 1 to 2).
* 21	Nonuniform shear force on side 1-2; magnitude supplied through user subroutine FORCEM.
* 22	Uniform shear force on side 2-3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2-3; magnitude supplied through user subroutine FORCEM.
* 24	Uniform shear force on side 3-4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3-4; magnitude supplied through user subroutine FORCEM.
* 26	Uniform shear force on side 4-1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4-1; magnitude supplied through user subroutine FORCEM.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Load Type	Description
102	Gravity loading in global direction. Enter two magnitudes: the first value is gravity acceleration in x-direction; the second is gravity acceleration in the y-direction.
103	Coriolis and centrifugal loading; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element is as follows:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \gamma_{xy}$

Output of Stresses

Output of stresses is the same as for the **Strains**.

Transformation

Two global degrees of freedom may be transformed to local coordinates.

Tying

Use subroutine UFORMS.

Output Points

Output is available at the centroid or at the four numerical integration points shown in Figure 3-3.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of true stress and logarithmic strain in global coordinate directions. Thickness will be updated if FINITE is specified.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Assumed Strain

The assumed strain formulation is available to improve the in-plane bending behavior. This increases the stiffness assembly costs per element and improves the accuracy.

Design Variables

The thickness can be considered a design variable for this element.

Element 4

Curved Quadrilateral, Thin-Shell Element

This is an isoparametric, doubly-curved thin-shell element using bicubic interpolation functions. The element is based in Koiter-Sanders shell theory, fulfilling continuity requirements, and represents rigid body modes exactly when used as a rectangle in the mapped surface coordinate plane. The element contains no patching functions, so that it is restricted to quadrilateral meshes with a maximum of four elements sharing one common node. However, the element is rapidly convergent in most problems which allow such a mesh. Note that any suitable surface coordinate systems may be chosen, so that the mesh need not be rectangular on the actual surface. This element cannot be used with CONTACT.

Geometry

The element is isoparametric, so that the actual surface in interpolated from nodal coordinates. The mesh is defined in the $\theta^1 - \theta^2$ plane of surface coordinates. Then, the actual surface is approximated with a surface defined by cubic interpolation on the interior of each element based on the following set of 14 nodal coordinates:

$$\theta^1, \theta^2, x, \frac{\partial x}{\partial \theta^1}, \frac{\partial x}{\partial \theta^2}, y, \frac{\partial y}{\partial \theta^1}, \frac{\partial y}{\partial \theta^2}, z, \frac{\partial z}{\partial \theta^1}, \frac{\partial z}{\partial \theta^2}, \frac{\partial^2 x}{\partial \theta^1 \partial \theta^2}, \frac{\partial^2 y}{\partial \theta^1 \partial \theta^2}, \frac{\partial^2 z}{\partial \theta^1 \partial \theta^2}$$

In most practical cases, the surface is definable as follows:

$$x = x(\theta^1, \theta^2)$$

$$y = y(\theta^1, \theta^2)$$

$$z = z(\theta^1, \theta^2)$$

Then, the usual procedure is to define the mesh in the $\theta^1 - \theta^2$ plane (as a rectangular mesh) by supplying the first two coordinates (θ^1, θ^2) at each node through the COORDINATE input option. Then, the remaining 12 coordinates are defined at each node through the use of the FXORD option (see Volume A) or the UFXORD user subroutine (see Volume D).

The element may have variable thickness since the program allows linear variation of the thickness with respect to θ^1 and θ^2. Note, however, the user should ensure that the thickness is continuous from one element to the next; otherwise, the tying option must be used to uncouple the membrane strain. In a continuous mesh, continuity of all membrane strain components is assumed.

Displacement

The following 12 degrees of freedom at each node:

$$u, \frac{\partial u}{\partial \theta^1}, \frac{\partial u}{\partial \theta^2}, v, \frac{\partial v}{\partial \theta^1}, \frac{\partial v}{\partial \theta^2}, w, \frac{\partial w}{\partial \theta^1}, \frac{\partial w}{\partial \theta^2}, \frac{\partial^2 u}{\partial \theta^1 \partial \theta^2}, \frac{\partial^2 v}{\partial \theta^1 \partial \theta^2}, \frac{\partial^2 w}{\partial \theta^1 \partial \theta^2}$$

where u, v, w are the Cartesian components of displacement.

Displacement is interpolated by complete bicubic functions on the interior of an element, so that equality of the above nodal degrees of freedom at the coincident nodes of abutting elements ensures the necessary continuity required for thin shell theory.

Note that fixed displacement boundary conditions should never be associated with all 12 degrees of freedom at each node, since three degrees of freedom must always determine middle surface (membrane) strains at the node. Care must be exercised in the specification of kinematic boundary conditions. They must be fully specified, but not over specified. The application of moments and torsional springs must consider the dimensions of the generalized coordinates so that the forces and conjugate displacements multiply together to yield mechanical work.

Connectivity Specification and Numerical Integration

The nodal point numbers of the element must be given in counterclockwise order on the θ^1 - θ^2 plane, starting with the point i (min θ^1 and θ^2). Thus, in Figure 3-4, the connectivity must be given as i, j, k, l.

The element is integrated numerically using nine points (Gaussian quadrature). The first integration point is always closest to the first node of the element; then, the integration points are numbered as shown in Figure 3-4.

Point 5 (centroid of the element in the θ^1 and θ^2 plane) is used for stress output if the CENTROID option is not flagged. The CENTROID option should be used for any nonlinear analysis with this element.

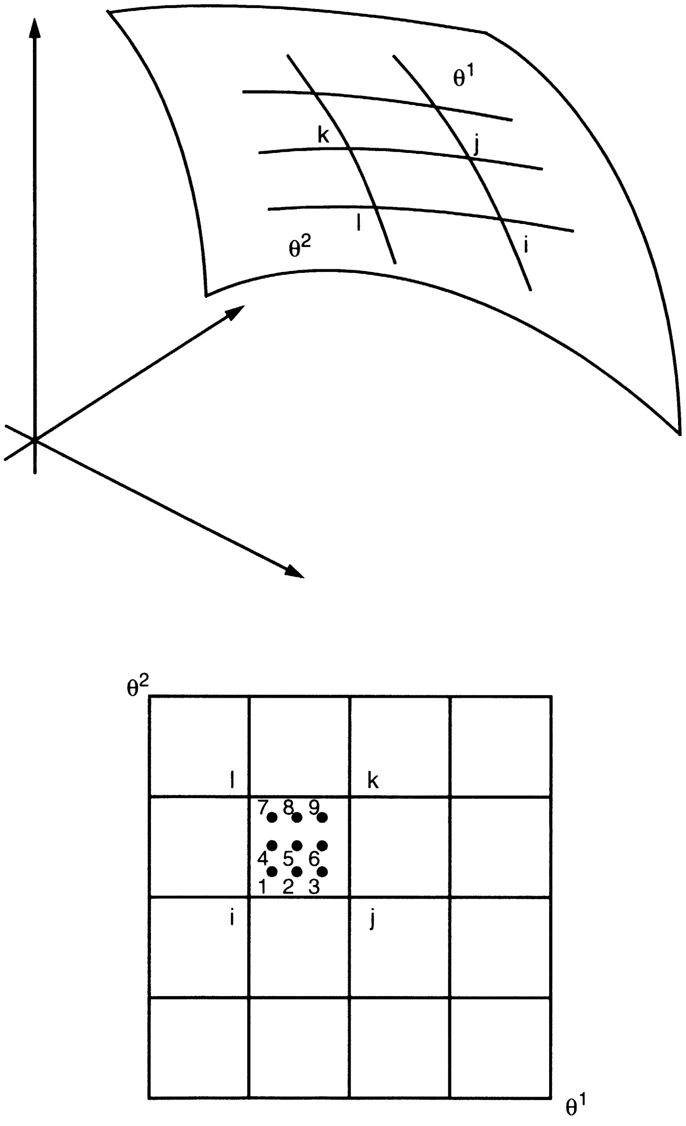


Figure 3-4 Form of Element 4

Quick Reference

Type 4

Doubly curved quadrilateral thin-shell element.

Connectivity

Four nodes per element. The first node given in the connectivity list must have minimum θ^1 and θ^2.

Geometry

Thickness at the element centroid is input in the first data field (EGEOM1).

Rate of change of thickness with respect to $\theta^1\left(\frac{\partial t}{\partial\theta^1}\right)$ is in the second data field (EGEOM2)

Rate of change of thickness with respect to $\theta^2\left(\frac{\partial t}{\partial\theta^2}\right)$ is in the third data field (EGEOM3).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

The geometry is defined by 14 nodal coordinates. The surface must be rectangular in the θ_1 and θ_2 plane. The required 14 nodal coordinates are as follows:

$$\theta_1, \theta_2, x, \frac{\partial x}{\partial\theta_1}, \frac{\partial x}{\partial\theta_2}, y, \frac{\partial y}{\partial\theta_1}, \frac{\partial y}{\partial\theta_2}, z, \frac{\partial z}{\partial\theta_1}, \frac{\partial z}{\partial\theta_2}, \frac{\partial^2 x}{\partial\theta_1\partial\theta_2}, \frac{\partial^2 y}{\partial\theta_1\partial\theta_2}, \frac{\partial^2 z}{\partial\theta_1\partial\theta_2}$$

Usually, the FXORD option or user subroutine UFXORD can be used to minimize the coordinate input.

Degrees of Freedom

There are 12 degrees of freedom per node as follows:

$$u, \frac{\partial u}{\partial\theta^1}, \frac{\partial u}{\partial\theta^2}, v, \frac{\partial v}{\partial\theta^1}, \frac{\partial v}{\partial\theta^2}, w, \frac{\partial w}{\partial\theta^1}, \frac{\partial w}{\partial\theta^2}, \frac{\partial^2 u}{\partial\theta^1\partial\theta^2}, \frac{\partial^2 v}{\partial\theta^1\partial\theta^2}, \frac{\partial^2 w}{\partial\theta^1\partial\theta^2}$$

Tractions

Distributed loadings are as follows:

Load Type	Description
1	Uniform self-weight per unit surface area in -z-direction.
2	Uniform pressure.
3	Nonuniform pressure; magnitude supplied by user subroutine FORCEM.
4	Nonuniform load per unit volume in arbitrary direction; magnitude and direction supplied in user subroutine FORCEM.
11	Uniform load per unit length on the 1-2 edge in the x-direction.
12	Uniform load per unit length on the 1-2 edge in the y-direction.
13	Uniform load per unit length on the 1-2 edge in the z-direction.
21	Uniform load per unit length on the 2-3 edge in the x-direction.
22	Uniform load per unit length on the 2-3 edge in the y-direction.
23	Uniform load per unit length on the 2-3 edge in the z-direction.
31	Uniform load per unit length on the 3-4 edge in the x-direction.
32	Uniform load per unit length on the 3-4 edge in the y-direction.
33	Uniform load per unit length on the 3-4 edge in the z-direction.
41	Uniform load per unit length on the 4-1 edge in the x-direction.
42	Uniform load per unit length on the 4-1 edge in the y-direction.
43	Uniform load per unit length on the 4-1 edge in the z-direction.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Components of stretch and curvature in surface coordinate directions.

Output of Stresses

Stress output as physical components of σ^{11}, σ^{22}, σ^{12} in surface coordinate system at points equally spaced through the thickness with the first point on the surface in the direction of the positive normal.

Transformation

Cartesian displacement components and their derivatives may be transformed to a local system. The surface coordinate system is not affected by this transformation.

Special Transformation

The shell transformation option type 4 may be used to permit easier application of point loads, moments and/or boundary conditions on a node. For a description of this transformation type, see Volume A. Note that if the FOLLOW FOR parameter is invoked, the transformations will be based on the updated configuration of the element.

Tying

Tying type 18 may be used for intersecting shells. Tying type 19 may be used for beam stiffened shells using element 13 as a stiffener on this element.

Output Points

Centroid or nine Gaussian integration points, if the CENTROID or ALL POINTS parameter is used as shown in Figure 3-4.

Note: Element is sensitive to boundary conditions. Ensure that every required boundary condition is applied properly.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain as for total Lagrangian approach. Thickness will be updated.

Note: Shell theory only applies if strain variation through the thickness is small.

Large Deformation Analysis

The large deformation analysis allows either the Lagrangian or updated Lagrangian description used in MARC. In the present version, however, only large deflection terms corresponding to the stretching strains have been introduced. This approximation is usually acceptable even for nonlinear buckling analysis.

Coupled Analysis

Not available for this element. Use either element types 22, 72, 75, or 139.

Element 5

Beam Column

This element is a straight, two-node, rectangular-section, beam-column element using linear interpolation parallel to its length, and cubic interpolation in the normal direction. (Element 16 is a full cubic beam that is more accurate for many applications.)

The degrees of freedom are the u and v displacements, and the right-handed rotation at the two end points of the element. The strain-stress transformation is formed by a Simpson's rule integration using the number of points defined in the SHELL SECT parameter through the thickness of the element. The stiffness is formed by Gaussian integration along the length of the element using three points. All constitutive relations may be used with this element type.

Quick Reference

Type 5

Two-dimensional, rectangular-section beam-column.

Connectivity

Two nodes per element (see Figure 3-5).

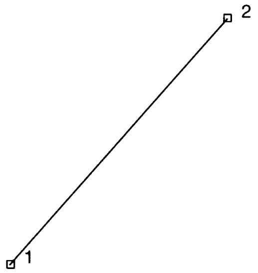


Figure 3-5 Beam-Column Element

Geometry

A rectangular section is assumed. The height is input in the first data field (EGEOM1). The cross-sectional area is input in the second data field (EGEOM2). The third data field is not used.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = right-handed rotation

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform pressure force/unit length.
1	Nonuniform pressure force/unit length.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter the magnitudes of gravity acceleration in the x- and y-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

In the nonuniform case, the load magnitude must be specified by user subroutine FORCEM.

Output of Strains

Two generalized strains, membrane stretching and bending.

Output of Stresses

Stresses at each integration point or the centroid. Each of these points will have points equally spaced through its thickness. The stress will be output at each representative point.

The first point is on the positive normal (positive local y) face. The last point is on the negative face.

Transformation

Standard transformation will transform first two global degrees of freedom to local degrees of freedom.

Tying

No standard tying available. Use subroutine UFORMS.

Output Points

Centroid or three Gaussian integration points.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available. Use element type 16 instead.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The height and/or the cross sectional area can be considered as design variables.

Element 6

Two-Dimensional Plane Strain Triangle

Element 6 is a three-node, isoparametric, triangular element written for plane strain applications. This element uses bilinear interpolation functions. The strains are constant throughout the element. This results in a poor representation of shear behavior.

In general, one needs more of these lower-order elements than the higher-order elements such as element type 125. Hence, use a fine mesh.

The stiffness of this element is formed using one point integration at the centroid.

This element should not be used in cases where incompressible or nearly incompressible behavior occurs because it will lock. This includes plasticity and creep. The cross triangle approach can be used to reduce the short comings of this element.

Quick Reference

Type 6

Two-dimensional, plane strain, three-node triangle.

Connectivity

Node numbering must be counterclockwise (see Figure 3-6).

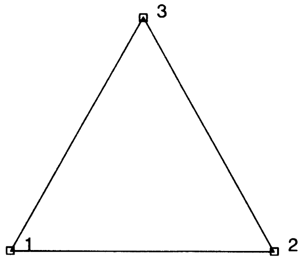


Figure 3-6 Plane-Strain Triangle

Geometry

Thickness stored in first data field (EGOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = u displacement
2 = v displacement

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through subroutine FORCEM.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Output of strains at the centroid of the element is as follows:

1 = ε_{xx}
2 = ε_{yy}
3 = $\varepsilon_{zz} = 0$
4 = γ_{xy}

Output of Stresses

Same as **Output of Strains.**

Transformation

Nodal degrees of freedom may be transformed to local degrees of freedom. Corresponding nodal loads must be applied in local direction.

Tying

Use subroutine UFORMS.

Output Points

Only available at the centroid. Element must be small for accuracy.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain in global coordinate directions. "Crossed triangle" approach recommended.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 37. See Element 37 for a description of the conventions used for entering the flux and film data for this element.

Element 7

Three-Dimensional Arbitrarily Distorted Brick

Element type 7 is an eight-node, isoparametric, arbitrary hexahedral. As this element uses trilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics may be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, one needs more of these lower-order elements than the higher-order elements such as types 21 or 57. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using eight-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method which eliminates potential element locking is flagged through the GEOMETRY option.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 84 instead. Element type 84 is also preferable for small strain incompressible elasticity.

Notes: For the assumed strain formulation, the interpolation functions are modified to improve the bending characteristics of the element.

As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 7

Three-dimensional, eight-node, first-order, isoparametric element (arbitrarily distorted brick).

Connectivity

Eight nodes per element. Node numbering must follow the scheme below (see Figure 3-7):

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1. Node 6 has the same edge as node 2. Node 7 has the same edge as node 3. Node 8 has the same edge as node 4.

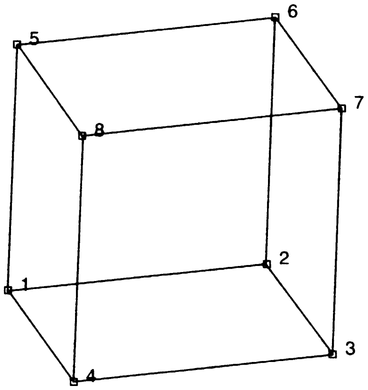


Figure 3-7 Arbitrarily Distorted Cube

Geometry

If the automatic brick to shell constraints are to be used, the first field must contain the transition thickness (see Figure 3-8). Note that in a coupled analysis, there are no constraints for the temperature degrees of freedom.

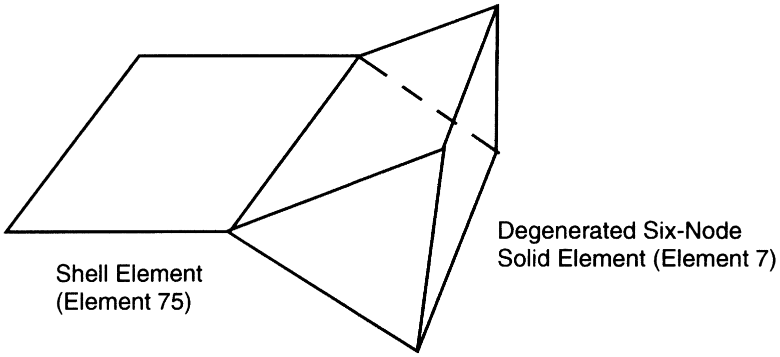


Figure 3-8 Shell-to-Solid Automatic Constraint

If a nonzero value is entered in the second data field (EGEOM2), the volumetric strain will be constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution.

If a one is placed in the third field, the assumed strain formulation will be activated.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM).
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force per unit volume in -z-direction.

Load Type	Description
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in FORCEM.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in FORCEM.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in FORCEM.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in FORCEM.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in FORCEM.
40	Uniform shear 1-2-3-4 face in the 1-2 direction.
41	Nonuniform shear 1-2-3-4 face in the 1-2 direction.
42	Uniform shear 1-2-3-4 face in the 2-3 direction.
43	Nonuniform shear 1-2-3-4 face in the 2-3 direction.
48	Uniform shear 6-5-8-7 face in the 5-6 direction.
49	Nonuniform shear 6-5-8-7 face in the 5-6 direction.
50	Uniform shear 6-5-8-7 face in the 6-7 direction.
51	Nonuniform shear 6-5-8-7 face in the 6-7 direction.
52	Uniform shear 2-1-5-6 face in the 1-2 direction.
53	Nonuniform shear 2-1-5-6 face in the 1-2 direction.
54	Uniform shear 2-1-5-6 face in the 1-5 direction.
55	Nonuniform shear 2-1-5-6 face in the 1-5 direction.
56	Uniform shear 3-2-6-7 face in the 2-3 direction.
57	Nonuniform shear 3-2-6-7 face in the 2-3 direction.
58	Uniform shear 3-2-6-7 face in the 2-6 direction.

Load Type	Description
59	Nonuniform shear 2-3-6-7 face in the 2-6 direction.
60	Uniform shear 4-3-7-8 face in the 3-4 direction.
61	Nonuniform shear 4-3-7-8 face in the 3-4 direction.
62	Uniform shear 4-3-7-8 face in the 3-7 direction.
63	Nonuniform shear 4-3-7-8 face in the 3-7 direction.
64	Uniform shear 1-4-8-5 face in the 4-1 direction.
65	Nonuniform shear 1-4-8-5 face in the 4-1 direction.
66	Uniform shear 1-4-8-5 in the 1-5 direction.
67	Nonuniform shear 1-4-8-5 face in the 1-5 direction.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure forces are positive into element face.

Output of Strains

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \gamma_{yz}$
$6 = \gamma_{zx}$

Output of Stresses

Output of stresses is the same as for **Output of Strains**.

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available. An automatic constraint is available for brick-to-shell transition meshes (see **Geometry**).

Output Points

Centroid or the eight integration points as shown in Figure 3-9.

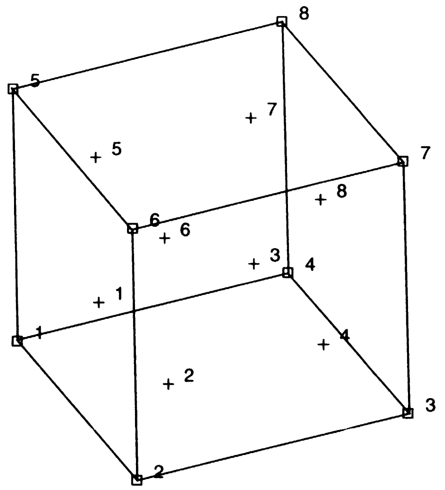


Figure 3-9 Eight-Point Gauss Integration Scheme for Element 7

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 43. See Element 43 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Assumed Strain

The assumed strain formulation is available to improve the bending behavior. This increases the stiffness assembly costs per element, but it improves the accuracy.

Notes: The element can be collapsed to a tetrahedron.

By collapsing one plane of the element to a line (see Figure 3-8), a transition element for connecting bricks with four-node shell element type 75 is generated. Thickness of the shell must be specified in the geometry field of the brick element.

Element 8

Curved Triangular Shell Element

This element is an isoparametric, curved, triangular, thin-shell element based on the Koiter-Sanders shell theory, which fulfills continuity requirements and represents rigid-body motions exactly. This element cannot be used with CONTACT.

Geometry

The middle surface of the shell is defined by the equations:

$$\begin{aligned} x &= x(\theta^1, \theta^2) \\ y &= y(\theta^1, \theta^2) \\ z &= z(\theta^1, \theta^2) \end{aligned} \qquad (1)$$

where

(x,y,z) are Cartesian coordinates.

(θ^1, θ^2) denote Gaussian coordinates on the middle surface of the shell.

The domain of definition in the plane (θ^1, θ^2) is divided into a mesh of triangles which are mapped onto curved elements on the middle surface Σ. The actual middle surface is approximated by a smooth surface $\bar{\Sigma}$ which has the same coordinates (x-y-z) and the same tangent plane at each nodal point of the mesh. Practically, the mesh is defined by the Caussian coordinates (θ_i^1, θ_i^2) of the nodal points, and the surface Σ is defined by the values of the functions (1) and their first derivatives at these points. According to the terminology of MARC, the coordinates are, therefore, the set:

$$\begin{aligned} &\theta_i^1, \theta_i^2, x(p_i), \partial x(p_i)/\partial\theta^1, \partial x(p_i)/\partial\theta^2 \\ &\qquad y(p_i), \partial y(p_i)/\partial\theta^1, \partial y(p_i)/\partial\theta^2 \\ &\qquad z(p_i), \partial z(p_i)/\partial\theta^1, \partial z(p_i)/\partial\theta^2 \end{aligned} \qquad (2)$$

where

$x(p_i)$ stand for $x(\theta_i^1, \theta_i^2)$, (θ_i^1, θ_i^2) being the coordinates of the node p_i.

In the general case, these 11 coordinates must be given. Particular shapes are available through the FXORD option (described in Volume A). Often the user subroutine UFXORD may be used to generate the coordinates from a reduced set (see Volume D). The thickness of the shell may vary linearly in an element: the values at the three nodes are given in EGEOM1, EGEOM2, EGEOM3. If EGEOM2 or EGEOM3 is given as zero, that thickness is set equal to EGEOM1.

There are nine degrees of freedom for each nodal point p_i. These degrees of freedom are defined in terms of the Cartesian components of displacement u, v, and w, and rates of change with respect to the Gaussian coordinates:

$u(p_i)$, $\partial u(p_i)/\partial\theta^1$, $\partial u(p_i)/\partial\theta^2$

$v(p_i)$, $\partial v(p_i)/\partial\theta^1$, $\partial v(p_i)/\partial\theta^2$

$w(p_i)$, $\partial w(p_i)/\partial\theta^1$, $\partial w(p_i)/\partial\theta^2$

The displacements within an element are defined by interpolation functions. These interpolation functions $\phi(\theta^1, \theta^2)$ are such that compatibility of displacements and their first derivatives is insured between adjacent elements. Hence, for an element whose vertices are the nodal points p_i, p_j, and p_k, the components u, v, w are defined as:

$$u(\theta^1, \theta^2) = (\underline{U}_i^T, \underline{U}_j^T, \underline{U}_k^T) \cdot \phi(\theta^1, \theta^2)$$

$$v(\theta^1, \theta^2) = (\underline{V}_i^T, \underline{V}_j^T, \underline{V}_k^T) , \phi(\theta^1, \theta^2)$$

$$w(\theta^1, \theta^2) = (\underline{W}_i^T, \underline{W}_j^T, \underline{W}_k^T) , \phi(\theta^1, \theta^2)$$

Numerical Integration

For this element, seven integration points are used with an integration rule which is exact for all polynomials up to the fifth order. See Figure 3-10.

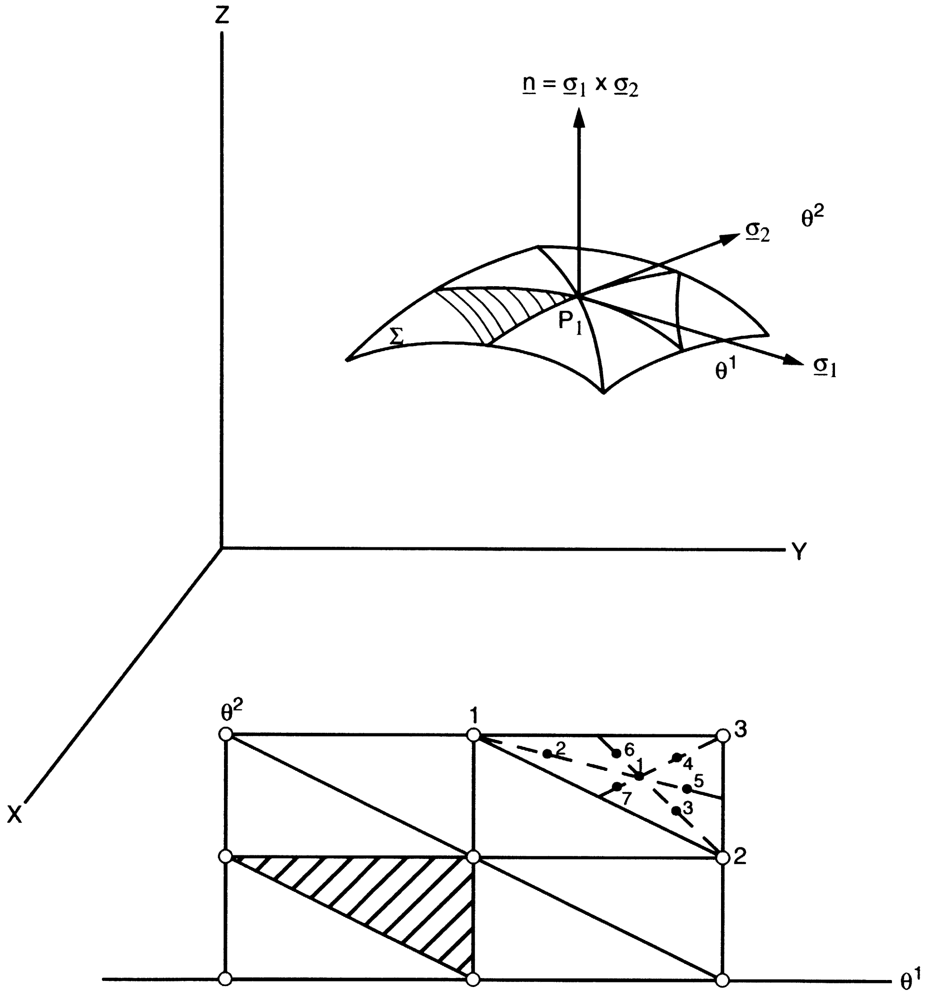


Figure 3-10 Uniform Loads On and Integration Points for Element 8

Quick Reference

Type 8

Arbitrary doubly curved triangular shell.

Connectivity

Three nodes per element. Numbering may be clockwise or counterclockwise for this element.

Geometry

The element can have linear variation of thickness in the θ^1, θ^2 plane. Thickness at the first node is input at the first data field (EGEOM1). Thickness at the second node in the second data field (EGEOM2). In the third data field (EGEOM3), thickness is at the third node. If only the first data field is used, the element defaults to a constant thickness.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

The coordinates are defined by Gaussian coordinates (θ^1, θ^2) on the middle surface of the shell with 11 coordinates per node required in the general case:

$1 = \theta^1$

$2 = \theta^2$

$3 = x$

$4 = \frac{\partial x}{\partial \theta^1}$

$5 = \frac{\partial x}{\partial \theta^2}$

$6 = y$

$7 = \frac{\partial y}{\partial \theta^1}$

$8 = \frac{\partial y}{\partial \theta^2}$

$9 = z$

$10 = \frac{\partial z}{\partial \theta^1}$

$11 = \frac{\partial z}{\partial \theta^2}$

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement

$2 = \frac{\partial u}{\partial \theta^1}$

$3 = \frac{\partial u}{\partial \theta^2}$

4 = v displacement

$5 = \frac{\partial v}{\partial \theta^1}$

$6 = \frac{\partial v}{\partial \theta^2}$

7 = w displacement

$8 = \frac{\partial w}{\partial \theta^1}$

$9 = \frac{\partial w}{\partial \theta^2}$

Tractions

Distributed loading types are as follows:

Load Type	Description
1	Uniform weight per surface area in the negative z-direction.
2	Uniform pressure; magnitude of pressure is positive when applied in negative normal vector direction.
3	Nonuniform pressure; magnitude given by user subroutine FORCEM.
4	Nonuniform load per unit volume in arbitrary direction; magnitude and direction supplied in user subroutine FORCEM.
11	Uniform load per unit length on the 1-2 edge in the x-direction.
12	Uniform load per unit length on the 1-2 edge in the y-direction.
13	Uniform load per unit length on the 1-2 edge in the z-direction.
21	Uniform load per unit length on the 2-3 edge in the x-direction.
22	Uniform load per unit length on the 2-3 edge in the y-direction.
23	Uniform load per unit length on the 2-3 edge in the z-direction.
31	Uniform load per unit length on the 3-1 edge in the x-direction.
32	Uniform load per unit length on the 3-1 edge in the y-direction.
33	Uniform load per unit length on the 3-1 edge in the z-direction.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Components of stretch and curvature in surface coordinate directions.

Output of Stresses

Physical components in surface coordinate directions at the points through the thickness: The first point on surface in the direction of positive normal; last point on surface in the direction of negative normal. Stress components are physical components in the θ^1, θ^2 directions:

$1 = \sigma^{11}$
$2 = \sigma^{22}$
$3 = \sigma^{12}$

Transformation

Cartesian displacement components and their derivatives may be transformed to a local system. The surface coordinate system is not affected by this transformation.

Special Transformation

The shell transformation option type 2 may be used to permit easier application of point loads, moments and/or boundary conditions of a node. For a description of the transformation type, see Volume A. Note that if the FOLLOW FOR parameter is invoked, the transformation will be based on the updated configuration of the element.

Tying

Typing type 18 is used for shell intersection. Tying type 19, 20, and 21 are provided for tying beam type 13 as a stiffener on this shell element.

Output Points

Centroid or seven integration points. These are located in the θ^1, θ^2 plane as shown in Figure 3-10.

Note: These results are sensitive to correct boundary condition specifications.

Section Stress Integration

Use SHELL SECT parameter to set number of points for Simpson rule integration through the thickness.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain as for total Lagrangian approach. Thickness will be updated.

Note: Shell theory only applies if strain variation through the thickness is small.

Large Deformation Analysis

The large deformation analysis allows either the Lagrangian or updated Lagrangian description used in MARC. In the present version, however, only large deflection terms corresponding to the stretching strains have been introduced. This approximation is usually acceptable even for nonlinear buckling analysis.

Element 9

Three-Dimensional Truss

Element type 9 is a simple linear straight truss with constant cross-section. The strain-displacement relations are written for large strain, large displacement analysis. All constitutive relations can be used with this element. This element can be used as an actuator in mechanism analyses.

Note: This element has no bending stiffness.

Quick Reference

Type 9

Two- or three-dimensional, two-node, straight truss. Used by itself or in conjunction with any 3-D element, this element will have three coordinates and three degrees of freedom. Otherwise, it has two coordinates and two degrees of freedom.

Connectivity

Two nodes per element (see Figure 3-11).

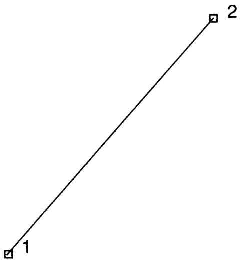


Figure 3-11 Two-Node Truss

Geometry

The cross-sectional area is input in the first data field (EGEOM1). Default area is equal to 1.0. The second and the third data fields are not used. The fourth field is used to define the length of the element when used as an actuator. The ACTUATOR history definition option, or the user subroutine UACTUAT can be used to modify the distance of the link.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

Global displacement degrees of freedom:
1 = u displacement
2 = v displacement
3 = w displacement (optional)

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform load (force per unit length) in the direction of the global x-axis.
1	Uniform load (force per unit length) in the direction of the global y-axis.
2	Uniform load (force per unit length) in the direction of the global z-axis.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in the x- and y-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Uniaxial in the truss member.

Output of Stresses

Uniaxial in the truss member.

Transformation

The three global degrees of freedom for any node may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Only one integration point available.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output as for total Lagrangian approach. Cross section will be updated.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The cross-sectional area can be considered a design variable for this element.

Element 10

Arbitrary Quadrilateral Axisymmetric Ring

Element type 10 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior.

In general, one needs more of these lower-order elements than the higher-order elements such as types 28 or 55. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method which eliminates potential element locking is flagged through the GEOMETRY option.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 82 instead. Element type 82 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 10

Axisymmetric, arbitrary ring with a quadrilateral cross section.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-12).

Geometry

If a nonzero value is entered in the second data field (EGEOM2), the volume strain will be constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady-state solution. For details, see Volume F, Reference XXV.

Coordinates

Two coordinates in the global z- and r-direction.

Degrees of Freedom

1 = u (displacement in the global z-direction)
2 = v (displacement in the global r-direction).

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
20	Uniform shear force on side 1-2 (positive from 1 to 2).
21	Nonuniform shear force on side 1-2; magnitude supplied through user subroutine FORCEM.
22	Uniform shear force on side 2-3 (positive from 2 to 3).

Load Type	Description
23	Nonuniform shear force on side 2-3; magnitude supplied through user subroutine FORCEM.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through user subroutine FORCEM.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through user subroutine FORCEM.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes. The magnitude of point loads must correspond to the load integrated around the circumference.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{qq}$
$4 = \gamma_{rz}$

Output of Stresses

Same as for **Output of Strains.**

Transformation

Two global degrees of freedom may be transformed into local coordinates.

Tying

May be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 3-12.

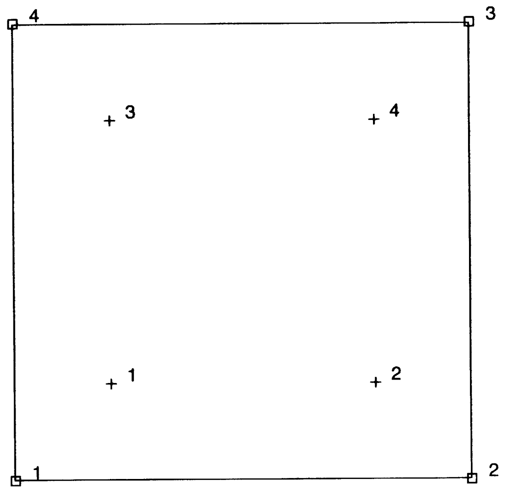


Figure 3-12 Integration Points for Element 10

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration recommended. (See **Geometry**.)

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 11

Arbitrary Quadrilateral Plane-Strain

Element type 11 is a four-node, isoparametric, arbitrary quadrilateral written for plane strain applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, one needs more of these lower-order elements than the higher-order elements such as types 27 or 54. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 82 instead. Element type 80 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 11

Plane-strain quadrilateral.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-13).

Geometry

The thickness is entered in the first data field (EGEOM1). Default thickness is one.

If a nonzero value is entered in the second data field (EGEOM2), the volume strain will be constant throughout the element. That is particularly useful for analysis of approximately incompressible materials and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady-state solution.

If a one is entered in the third field, the assumed strain formulation will be used.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
20	Uniform shear force on side 1-2 (positive from 1 to 2).

Load Type	Description
21	Nonuniform shear force on side 1-2; magnitude supplied through user subroutine FORCEM.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through user subroutine FORCEM.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through user subroutine FORCEM.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through user subroutine FORCEM.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter the magnitude of gravity acceleration in the z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{xy}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Two global degrees of freedom may be transformed into local coordinates.

Tying

Use subroutine UFORMS.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 3-13.

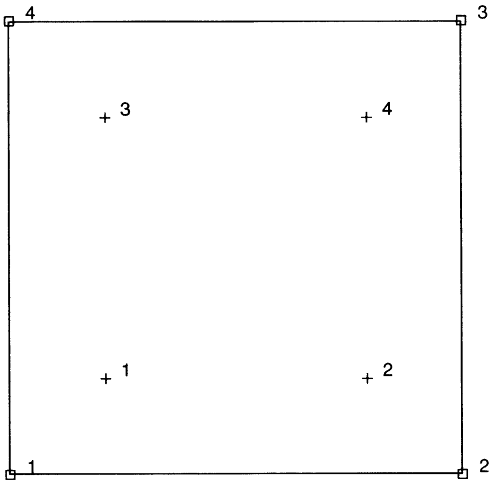


Figure 3-13 Gaussian Integration Points for Element Type 11

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration recommended. (See **Geometry**.)

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Assumed Strain

The assumed strain formulation is available to improve the bending characteristics of this element. Although this increases the stiffness assembly costs per element, it does improve the accuracy.

Element 12

Friction And Gap Link Element

This element provides frictional and gapping connection between any two nodes of a structure. Essentially, the element is based on imposition of a gap closure constraint and frictional stick or slip via Lagrange multipliers.

The element may be used with any other elements in the program by invoking suitable tying (user subroutine UFORMS), if necessary.

Three different options for the definition of the gap have been included. The default formulation is a gap in a fixed direction. This option is useful for geometrically linear analysis or geometrically nonlinear analysis if a body is not to penetrate a given flat surface.

The second formulation constrains the true distance between the two end-points of the gap to be greater or less than a specified value. This option is useful for geometrically nonlinear analysis or for analysis in which a body is not to penetrate a given circular (2D) or spherical (3D) surface. This option is activated by specification of a "1" in the seventh data field of GAP DATA model definition block.

The third formulation allows specification of closure distance and gap direction in the user subroutine GAPU. The gap direction and distance can then be updated during analysis to model sliding along a curved surface. The fixed direction gap option in GAP DATA block must be activated if GAPU is used.

Fixed Direction Gap

Description of the Fixed Direction Gap for Two-Dimensional Problems

The element is implemented in the program as a four-node element (link). The first and fourth nodes have (u, v) Cartesian displacements to couple to the rest of the structure.

Node 2 is the gap node.It has one degree of freedom, F_n, the force being carried across the link. The coordinate data for this node is used to input (n_x, n_y), the direction of $\underset{\sim}{n}$, the gap closure direction. If these data are not given (or are all zero), the program defines:

$$\underset{\sim}{n} = (\underset{\sim}{X}_4 - \underset{\sim}{X}_1)/|\underset{\sim}{X}_4 - \underset{\sim}{X}_1|;$$

i.e., the gap closure direction is along the element in its original configuration. The user should note that, in many cases, the gap is very small (or, indeed, may be of zero length if the two surfaces are initially touching), so that inaccuracies may be introduced by taking a small difference between two large values. This is the reason for allowing separate input of (n_x, n_y).

Node 3 is the friction node. It has degrees of freedom F; the frictional force being carried across the link, and s, the net frictional slip. The coordinate data for this node may be given as (t_x, t_y), the frictional direction. If the user does not input this data, $\underset{\sim}{t}$ is defined by the program as:

$$\underset{\sim}{t} = \underset{\sim}{k} \times \underset{\sim}{n}$$

where $\underset{\sim}{n}$ is the gap direction (see above) and $\underset{\sim}{k}$ is the unit vector normal to the plane of analysis.

Description of the Fixed Direction Gap for Three-Dimensional Problems

The first and fourth nodes have (u, v, w) Cartesian displacements to couple to the rest of the structure.

Node 2 is the gap node. It has one degree of freedom, F_n; the normal force being carried across the link. The coordinate data for this node is used to input (n_x, n_y, n_z), the direction of $\underset{\sim}{n}$, the gap closure direction. If these data are not given (or are all zero), the program defines:

$$\underset{\sim}{n} = (\underset{\sim}{X}_4 - \underset{\sim}{X}_1)/|\underset{\sim}{X}_4 - \underset{\sim}{X}_1| \;;$$

i.e., the gap closure direction is along the element in its original configuration. You should note that in many cases, the gap is very small (or, indeed, may be of zero length if the two surfaces are initially touching), so that inaccuracies may be introduced by taking a small difference between two large values. This is the reason for allowing separate input of (n_x, n_y, n_z).

Node 3 is the friction node. It has degrees of freedom (F_1, F_2); the frictional forces being carried cross the link and s, the net frictional slip.

The coordinate data for this note may be given as (t_x^1, t_y^1, t_z^1), the first frictional direction. If you do not input this data, $\underset{\sim}{t}^1$ is defined by the program as:

$$\underset{\sim}{t}^1 = \underset{\sim}{i} \times \underset{\sim}{n}$$

where $\underset{\sim}{n}$ is the gap direction (see above) and $\underset{\sim}{i}$ is the unit vector in the global $\underset{\sim}{x}$-direction. If $\underset{\sim}{n}$ is parallel to $\underset{\sim}{i}$, the first friction direction is defined as:

$$\underset{\sim}{t}^1 = -\underset{\sim}{j}$$

where j is the global y-direction. The second friction direction $\underset{\sim}{t}^2$ is calculated by the program as:

$$\underset{\sim}{t}^2 = \underset{\sim}{n} \times \underset{\sim}{t}^1$$

Note on the Fixed Directional Gap

The gap is closed when $(\underset{\sim}{u}_1 - \underset{\sim}{u}_4) \bullet \underset{\sim}{n} = u_{c1}$, and this relative displacement in direction $\underset{\sim}{n}$ cannot be exceeded. The closure distance u_{c1} is given in the first data field of the GAP DATA option. A negative number insures that a closed gap condition is prescribed during increment zero. Note that no nonlinearity is accounted for in increment zero; i.e., the gap should be either open or closed as defined by the user. The closure distance can be updated by the user through user subroutine GAPU.

The second data field of the GAP DATA option is used to input the coefficient of friction, μ. If the coefficient of friction is set to zero, the friction calculations are skipped and the element acts as a gap only.

True Distance Gap

Description of the True Distance Gap

In this formulation, the nodes have the same meaning as in the fixed direction formulation. Nodes 1 and 4 have (u, v) Cartesian displacements to couple to the rest of the structure. For three-dimensional problems, nodes 1 and 4 have (u, v, w) Cartesian displacements. Node 2 has one degree of freedom, the gap force F_n. In contrast to the fixed direction gap, the constraint enforced by this true distance gap is as follows:

$$|\underset{\sim}{x}_4 - \underset{\sim}{x}_1| \geq d, \qquad \text{if } d > 0$$

or

$$|\underset{\sim}{x}_4 - \underset{\sim}{x}_1| \leq -d \qquad \text{if } d < 0$$

where |d| is the minimum or maximum distance between the end-points defined in the first data field of the GAP DATA option. Note that this distance must always be positive. From the above equation follows that the gap closure direction is defined as follows:

$$\underset{\sim}{n} = \pm(\underset{\sim}{x}_4 - \underset{\sim}{x}_1)/|\underset{\sim}{x}_4 - \underset{\sim}{x}_1| ;$$

i.e., the gap closure direction is along the element in its current configuration. The user cannot specify any different direction.

General Comments

Since it is very important that the degrees of freedom of the gap element are eliminated in an appropriate order, automatic internal renumbering of the nodes connected to the gap is carried out prior to the analysis (but after eventual optimization). The user cannot influence this procedure. In problems with large numbers of gaps and/or high friction coefficients, the convergence of the gap and friction algorithm is sometimes rather slow. Very often, this is due to iterations of elements in areas on the borderline of opening-closing and/or slipping-sticking. This is a local effect that shows itself in the often good convergence of other measures, such as the displacements. Hence, even if gap convergence is not reached completely in the specified number of cycles, the solution may still be sufficiently accurate for all practical purposes. In that case, the program will continue with the analysis after issuing a warning message. If no convergence problems occur in subsequent increments, such a nonconvergence usually has no significant effect in the subsequent results. To obtain information regarding gap convergence, use the PRINT,5 option.

Quick Reference

Type 12

Four-node friction/gap element. May be used with any other element types, if necessary through appropriate tying.

Connectivity

Four nodes per element. Nodes 1 and 4 are the ends of the link to connect to the rest of the structure, node 2 is the gap node and node 3 is the friction node.

The program automatically renumbers the internal node numbers to avoid equation solver problems. This occurs after optimization and may lead to a non-optimal bandwidth.

Coordinates

Nodes 1 and 4 - Cartesian coordinates (x, y) for two-dimensional problems, otherwise (x, y, z).

Node 2 and 3 - (fixed direction gap only):

Node 2 - gap direction cosines (n_x, n_y) or (n_x, n_y, n_z) for 3D problems
Node 3 - friction direction cosines (t_x, t_y) or (t_x, t_y, t_z) for 3D problems

Gap Data

First Data Field
For the fixed direction gap, this field is used to define u_{c1}, the closure distance.

For the true distance gap, this field is used to define d, the minimum distance between the end-points. If $d > 0$, the two end-points will be at least a distance d apart. If $d < 0$, the two end-points will not move further apart than a distance -d.

Second Data Field
This data field is used to define the coefficient of friction.

Third Data Field
The data field is used to define the elastic stiffness (spring stiffness) of the closed gap in the gap direction. If the field is left blank, the gap is assumed to be rigid when closed.

Fourth Data Field
This data field is used to define the elastic stiffness (spring stiffness) of the closed gap in friction direction. If the field is left blank, the nonslipping gap is rigid in the slip direction.

Fifth Data Field
User supplied momentum ratio for the first gap node.

Sixth Data Field
User supplied momentum ratio for the fourth gap node.

Seventh Data Field
User enters a "1" for true distance gap; a "0" for fixed direction gap.

Eighth Data Field
User enters a "1" for the condition that the gap is closed during increment 0; a "0" for the condition that the gap is open during increment 0.

Degrees Of Freedom

Nodes 1 and 4 (u, v) Cartesian components of displacement in two-dimensional problems, (u, v, w) Cartesian components of displacement in three-dimensional problems.

Node 2 - F_n, force in the gap direction.

Node 3 - F_s, frictional force and, s, net frictional slip in two-dimensional problems, or F_1, F_2, frictional forces and s, net frictional slip in three-dimensional problems.

s is the accumulated total slip under nonzero frictional forces.

Special Considerations

The TRANSFORMATION option should not be invoked at nodes 2 and 3 of this element.

Updated Lagrange And Finite Strain Plasticity

Use true distance gap or subroutines GAPU if necessary – large strain option not relevant.

Element 13

Open Section Thin-Walled Beam

This element is an open-section, curved, thin-walled beam of arbitrary section. The geometry is interpolated cubically from coordinate and direction information at two nodes. The element is illustrated in Figure 3-14. The following pages describe how the user may set up the cross section and the orientation of the beam and its section in space, and define the degrees of freedom, strains and distributed loads associated with the element. The element is based on classical theory of thin-walled beams with nondeforming sections.

Primary warping effects are included, but twisting is always assumed to be elastic as follows:

$$M_T = GJ\frac{d\phi}{ds}$$

where G = shear modulus and $\frac{d\phi}{ds}$ = rate of twist per unit length along the beam axis.

Thus, the shear stress is assumed small compared to the axial stress and is neglected in the formation of elastic-plastic and creep relations. The twisting stiffness is formed directly by numerical integration:

$$J = \int_0^{\Sigma}\frac{1}{3}(t\ (\sigma))^3\ d\sigma$$

where σ is the distance along the section, $0<\sigma<\Sigma$, and $t(\sigma)$ is the wall thickness.

The axial force, bending moments and bimoment are formed by numerical integration of the direct stress, which is obtained via one-dimensional elastic-plastic creep constitutive theory.

The cross section is assumed to remain undeformed during loading of the beam. Large displacement effects are included in the direct strain only.

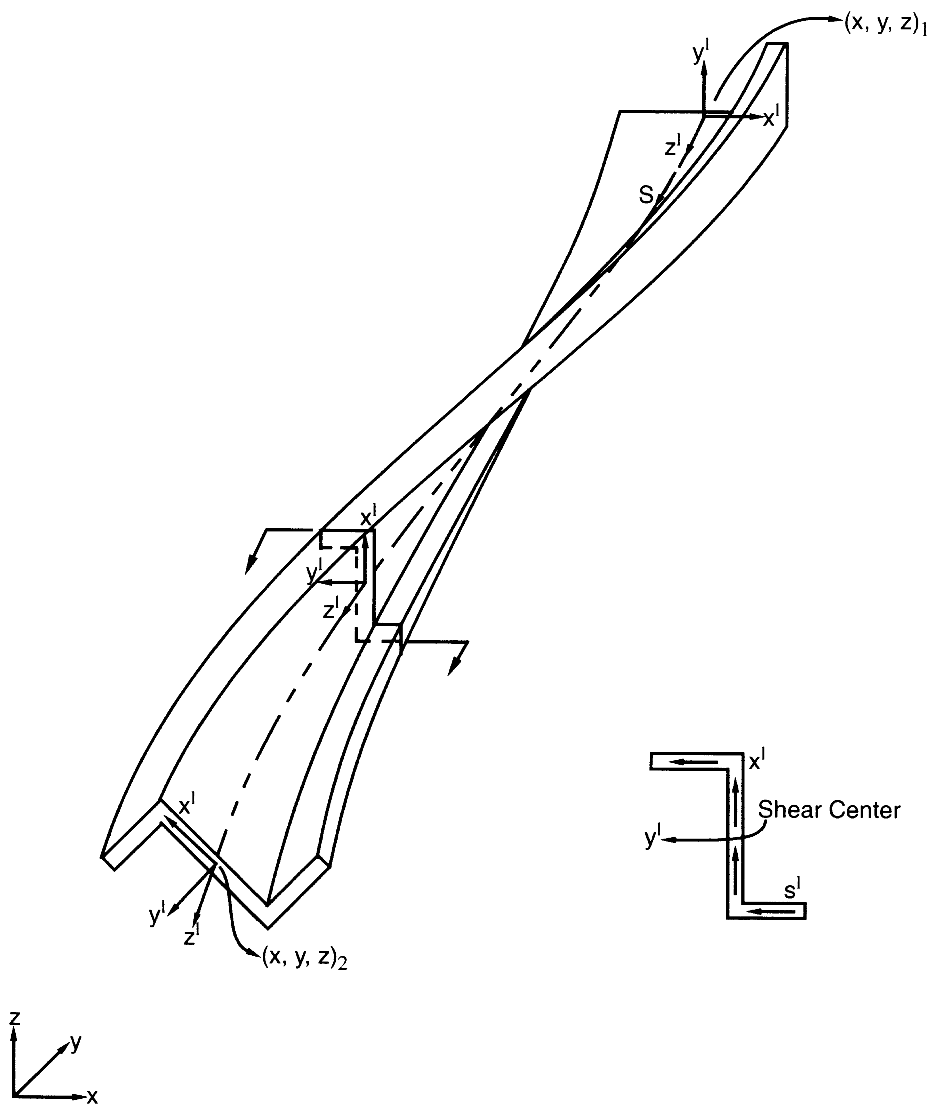


Figure 3-14 Typical Beam Element, (x^l, y^l, z^l)

Definition of Open Section Beam Geometry

Axis System

The convention adopted in Element 13 for the director set at a point of the beam is as follows:

> The first and second directions (local x and y) at a point are normal to the beam axis.
>
> The third director (local z) is tangent to the beam axis and is in the direction of increasing distance s along the beam.

The director set must form a right-handed system.

Orientation of the Section in Space

The beam axis in an element is interpolated by a cubic from the first six coordinates at the two nodes of an element. The coordinates are as follows:

$$x, y, z, \frac{dx}{ds}, \frac{dy}{ds}, \frac{dz}{ds}$$

where s is the continuous distance along the beam and is given as the thirteenth coordinate at each node. The orientation of the beam section in an element is defined by the direction of the first director at a point (local x), and this direction is interpolated by a cubic from the seventh through the twelfth coordinates at the two nodes of an element. The coordinates are as follows:

$$a_1, a_2, a_3, \frac{da_2}{ds}, \frac{da_3}{ds}, \frac{da_3}{ds}$$

where a_1, a_2, a_3, are the components in the global directions of the first director at the node. Since the interpolated director $\underline{a}$, will not, in general, be orthogonal to the interpolated beam axis tangent $\underline{a}_3$, an internal correction is applied at each numerical integration point of the element according to the formula:

$$\underline{b}_1^c = \frac{1}{\sqrt{1-(\underline{b}_1 \cdot \underline{b}_3)^2}}(\underline{b}_1 \bar{\underline{b}}_1^c \cdot \underline{b}_3)\underline{b}_3)$$

Where $\underline{b}_1$, $\underline{b}_3$, are the unit vectors along $\underline{a}_1$, a_3, and $\underline{b}_1^c$ is the corrected unit vector along the first director. The second director is then obtained from the cross product $\underline{b}_3 \times \underline{b}_1$.

Displacements

There are eight degrees of freedom at each node. These are as follows:

$$u, \frac{du}{ds}, v, \frac{dv}{ds}, w, \frac{dw}{ds}, \phi, \frac{d\phi}{ds}$$

where u, v, w, are the components of displacement in the global directions and ϕ is the rotation about the beam axis. Here, $\frac{d}{ds}$ represents differentiation with respect to distance along the beam.

Strains

Five generalized strains are associated with each integration point along the beam axis. These are oriented with respect to the interpolated director set (1, 2, 3) at the integration point are defined as follows:

1 = Direct strain on the beam axis (along third director).
2 = Curvature about the first director.
3 = Curvature about the second director.
$4 = \frac{d^2\phi}{ds^2}$ where ϕ is rotation about the third direction; i.e., warping of the section.
$5 = \frac{d\phi}{ds}$ = twist of the section.

Distributed Loads

Distributed loads are available as uniform load per unit length in the global (x, y, z) directions. The type of load is given as 1(x), 2(y), or 3(z) and the magnitude is given as the load per unit length along the beam.

Quick Reference

Type 13

Open-section, thin-walled beam of arbitrary section, including twist and warping.

Connectivity

Two nodes per element.

Geometry

Section is defined by you in an additional set of data blocks. The section number is given for an element by input in the second data field (EGEOM2). The other data fields are not used.

Coordinates

Beam axis and cross-section orientation interpolated cubically from 13 coordinates per node:

$$x, y, z, \frac{dx}{ds}, \frac{dy}{ds}, \frac{dz}{ds}, a_1, a_2, a_3, \frac{da_1}{ds}, \frac{da_2}{ds}, \frac{da_3}{ds}, s$$

where (a_1, a_2, a_3) is a vector defining the direction of the first local axis of the cross-section.

Degrees of Freedom

Eight degrees of freedom per node: $u, \frac{du}{dv}, v, \frac{dv}{ds}, w, \frac{dw}{ds}, \phi, \frac{d\phi}{ds}$ where s is the distance along the beam.

Tractions

Distributed loads are per unit length of beam in the three global directions.

Load Type	Description
1	Uniform load per unit length in x-direction.
2	Uniform load per unit length in y-direction.
3	Uniform load per unit length in z-direction.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Five generalized strains are: stretch along the beam axis, two curvatures about local x- and y-axis, warp and twist per unit length.

Output of Stresses

The program provides the user with the location of each stress point on the section. The stress output is axial stress at each stress point of the sections. (Maximum of 31 points.)

Transformation

The displacement vectors may be transformed into local degrees of freedom.

Tying

Use tying type 13 to join two elements under an arbitrary angle. May be used as a stiffener on the arbitrary curved shell element (element type 8) by tying types 19, 20, 21.

Output Points

Centroid or three Gaussian integration points along the beam. First point is closest to the first node of the beam.

For all beam elements, the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain, or nonzero temperature. This default printout may be changed via the PRINT CHOICE option.

Beam Sect

BEAM SECT parameter is required.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is not available. Use element types 77 or 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Element 14

Thin-Walled Beam in Three Dimensions without Warping

This is a simple, straight beam element with no warping of the section, but including twist. The default cross section is a thin-walled circular closed-section beam. The user may specify alternative cross sections through the BEAM SECT parameter.

The degrees of freedom associated with each node are three global displacements and three global rotations; all defined in a right-handed convention. The generalized strains are stretch, two curvatures, and twist per unit length. Stresses are direct (axial) and shear given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined in Geometry fields 4, 5, and 6. Using the GEOMETRY option, a vector in the plane of the local x-axis and the beam axis must be specified. If no vector is defined here, the local coordinate system may alternatively be defined by the fourth, fifth, and sixth coordinates at each node, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam toward this point. The local z-axis is along the beam from the first to the second node and the local y-axis forms a right-handed set with the local x and local z.

For other than the default (circular) section, the stress points are defined by the user in the local x-y set through the BEAM SECT parameter set. For the circular hollow section, EGEOM1 is the wall thickness, EGEOM2 is the radius. Otherwise, EGEOM2 gives the section choice from the BEAM SECT input. Section properties are obtained by numerical integration over the stress points of the section.

All constitutive models can be used with this element.

Standard (Default) Circular Section

The positions of the 16 numerical integration points are shown in Figure 3-15. The contributions of the 16 points to the section quantities are obtained by numerical integration using Simpson's rule.

Note: For noncircular sections, the BEAM SECT parameter must be used to describe the section.

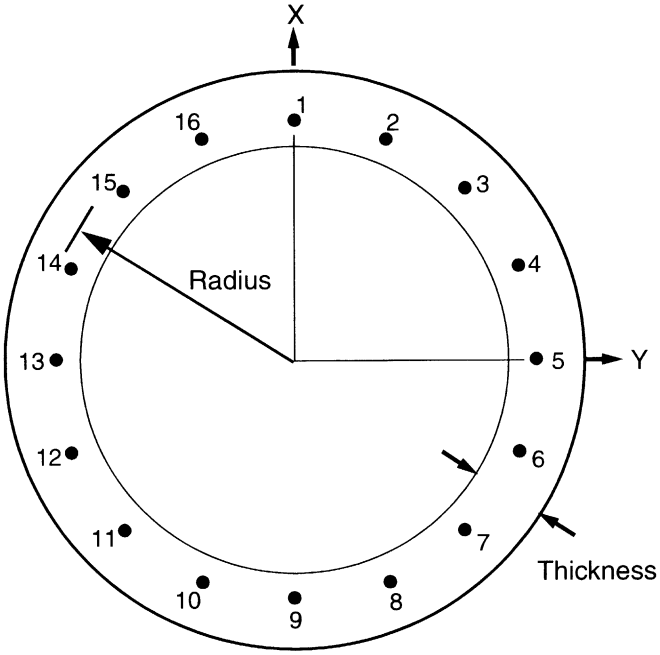


Figure 3-15 Default Cross Section

Special Considerations

Note that element 25 is the same as this element, but with axial strain as an additional degree of freedom at each node. This yields superior results for large displacement problem or problems involving axial temperature gradients. Elements of types 14, 25, 52, 76, 77, 78, 79, and 98 may be used together directly.

For all beam elements, the default printout gives section forces and moments plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.

Quick Reference

Type 14

Closed section beam, Euler-Bernoulli theory.

Connectivity

Two nodes per element (see Figure 3-16).

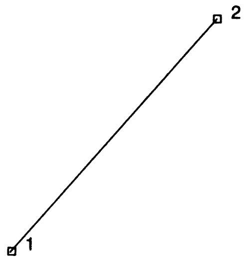


Figure 3-16 Closed-section Beam

Geometry

In the default section of a hollow, circular cylinder, the first data field is for the thickness (EGEOM1). For noncircular section, set EGEOM1 to 0. For circular section, set EGEOM2 to radius. For noncircular section, set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.) EGEOM4-EGEOM6: Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis will lie on the same side as the specified vector.

Coordinates

Six coordinates per node. The first three are global (x,y,z). The fourth, fifth, and sixth are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x axis is a vector normal to the beam axis through the point described by the fourth, fifth and sixth coordinates. The local x axis is positive progressing from the beam to the point. The fourth, fifth, and sixth coordinates will only be used if the local x-axis direction is not specified in the GEOMETRY block.

Degrees of Freedom

$1 = u$ $4 = \theta_x$
$2 = v$ $5 = \theta_y$
$3 = w$ $6 = \theta_z$

Tractions

Distributed load types are as follows:

Load Type	Description
1	Uniform load per unit length in the global x-direction.
2	Uniform load per unit length in the global y-direction.

Load Type	Description
3	Uniform load per unit length in the global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via user subroutine FORCEM.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads and moments may be applied at the nodes.

Output of Strains

1 = axial stretch.

2 = curvature about local x-axis of cross-section.

3 = curvature about local y-axis of cross-section.

4 = twist per unit length.

Output of Stresses

Stresses, 1 = axial stress; 2 = twisting shear

Transformation

Displacement and rotations at the nodes may be transformed to a local coordinate reference.

Tying

Special tying types exist for use of this element with element 17 to form complete pipelines for nonlinear piping system analysis. See element 17 description. Use tying type 100 for fully moment-carrying joints and tying type 52 for pinned joints.

Output Points

Centroid or three Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is at the midspan location of the beam. The third point is near the second node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available for this element. This element does not have a finite strain capability.

Coupled Analysis

In a coupled thermal-mechanical, analysis the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and/or the radius can be considered as design variables.

Element 15

Axisymmetric Shell, Isoparametric Formulation

Element type 15 is a two-node, axisymmetric, thin-shell element, with a cubic displacement assumption based on the global displacements and their derivatives with respect to distance along the shell. The strain-displacement relationships used are suitable for large displacements with small strains. The stress-strain relationship is integrated through the thickness using Simpson's rule, the first and last points being on the surfaces. Three-point Gaussian integration is used along the element. All constitutive relations may be used with this element.

Quick Reference

Type 15

Axisymmetric, curved, thin-shell element.

Connectivity

Two nodes per element (see Figure 3-17).

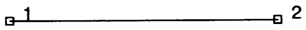


Figure 3-17 Axisymmetric, Curved Thin-Shell Element

Geometry

Linear thickness variation along length of the element. Thickness at first node of the element store in the first data field (EGEOM1).

Thickness at second node store in the third data field (EGEOM3).

If EGEOM3=0, constant thickness is assumed. Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used since, in the other case, section properties formed at the centroid of the element are used for all integration points.

The second data field is not used (EGEOM2).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

$1 = z$

$2 = r$

$3 = \frac{dz}{ds}$

$4 = \frac{dr}{ds}$

$5 = s$

Note: The redundancy in the coordinate specification is retained for simplicity of use with generators.

Degrees of Freedom

$1 = u =$ axial (parallel to symmetry axis)

$2 = v =$ radial (normal to symmetry axis)

$3 = \frac{du}{ds}$

$4 = \frac{dv}{ds}$

Tractions

Distributed loads selected with IBODY are as follows:

Load Type (IBODY)	Description
0	Uniform pressure.
1	Uniform load in 1 direction (force per unit area).
2	Uniform load in 2 direction (force per unit area).
3	Nonuniform load in 1 direction (force per unit area).
4	Nonuniform load in 2 direction (force per unit area).
5	Nonuniform pressure.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter the magnitudes of gravity acceleration in the z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure assumed positive in direction opposite of the normal obtained by rotation of 90° from direction of increasing s (see Figure 3-17).

In the nonuniform cases (IBODY = 3, 4, or 5), the load magnitude must be supplied by user subroutine FORCEM.

Concentrated loads applied at the nodes must be integrated around the circumference.

Output of Strains

Generalized strains are as follows:

1 = meridional membrane (stretch)
2 = circumferential membrane (stretch)
3 = meridional curvature
4 = circumferential curvature

Output of Stresses

Stresses are output at the integration points through the thickness of the shell. The first point is on the surface of the positive normal.

1 = meridional stress
2 = circumferential stress

Transformation

The degrees of freedom may be transformed to local directions.

Special Transformation

The shell transformation option type 1 may be used to permit easier application of moments and/or boundary conditions on a node. For a description of this transformation type, see Volume A. Note that if the FOLLOW FOR option is invoked, the transformations will be based on the updated configuration of the element.

Output Points

Centroid or three Gaussian integration points. The first Gaussian integration point is closest to the first node as defined in the connectivity data. The second integration point is at the midspace location. The third integration point is at the second node as defined in the connectivity data.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in meridional and circumferential direction. Thickness will be updated.

Note: Shell theory only applies if strain variation through the thickness is small.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is element type 88. See Element 88 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 16

Curved Beam in Two-Dimensions, Isoparametric Formulation

Element type 16 is a two-node curved beam, with displacements interpolated cubically from the global displacements and their derivatives with respect to distance along the beam at the two end nodes. The strain-displacement relations used are suitable for large displacements with small strains. The stress-strain relationship is integrated through the thickness by a Simpson rule; the first and last points being on the two surfaces. Three-point Gaussian integration is used along the element. The cross section is a solid rectangle. All constitutive relations can be used with this element.

Quick Reference

Type 16

Two-node curved beam element.

Connectivity

Two nodes per element (see Figure 3-18).

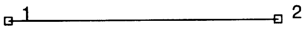


Figure 3-18 Two-Node Curved Beam Element

Geometry

Linear thickness variation along the element. Thickness at the first node of the element is stored in the first data field (EGEOM1). Thickness at second node is stored in the third data field (EGEOM3). If EGEOM3=0, constant thickness assumed.

Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used; since in the other case, section properties formed at the centroid of the element are used for all integration points.

The beam width is in the second data field (EGEOM2). The default width is unity.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Nodal coordinates in right-hand set (x,y)

1 = x
2 = y
$3 = \frac{dx}{ds}$
$4 = \frac{dy}{ds}$
5 = s

where s is the distance along the beam measured continuously starting from one end of the beam.

Degrees of Freedom

Degrees of freedom in right-hand set (u,v):

1 = u
2 = v
$3 = \frac{du}{ds}$
$4 = \frac{dv}{ds}$

where s = distance along the beam.

Tractions

Distributed loads selected with IBODY are as follows:

Load Type (IBODY)	Description
0	Uniform pressure.
1	Uniform load (force per unit length) in 1 direction.
2	Uniform load (force per unit length in 2 direction.
3	Nonuniform load (force per unit length) in 1 direction.
4	Nonuniform load (force per unit length) in 2 direction.
5	Nonuniform pressure.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.

Load Type (IBODY)	Description
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter the magnitudes of gravity acceleration in the x- and y-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure assumed positive in direction opposite of the normal obtained by rotation of 90° from direction of increasing s (see Figure 3-18).

In the nonuniform cases (IBODY = 3, 4, or 5), the load magnitude must be supplied by user subroutine FORCEM.

Output of Strains

Generalized strains:

1 = membrane (stretch)
2 = curvature

Output of Stresses

Output of axial stress at points through thickness (first and last points are on surfaces). First point is on surface up positive normal. Points proceed down normal at equal intervals.

Transformation

All degrees of freedom can be transformed to local directions.

Special Transformation

The shell transformation option type 1 may be used to permit easier application of moments and/or boundary conditions on a node. For a description of this transformation type, see Volume A, section 2. Note that if the FOLLOW FORCE option is invoked, the transformations will be based on the updated configuration of the element.

Tying

Requires user subroutine UFORMS.

Output Points

The first Gaussian integration point is closest to the first node defining the element in the CONNECTIVITY data. The second point is at the midspan beam location. The third point is closest to the second node describing the element in the CONNECTIVITY data.

Section Stress Integration

Integration through-the-thickness is performed numerically using Simpson's rule. The number of integration points is specified with the SHELL SECT parameter. This number must be odd.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in beam direction. Thickness will be updated, but beam width is assumed to be constant.

Note: Beam theory only applies if strain variation over the thickness is small.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is element type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness (beam height) and/or the beam width can be considered as design variables.

Element 17

Constant Bending, Three-Node Elbow Element

This element modifies Element type 15 into a pipe-bend approximation. The main purpose of the element is to provide nonlinear analysis of complete piping loops at realistic cost: The straight pipe sections are modeled by Element type 14, and the bends are built up by using several sections of these modified axisymmetric elements. Each such section has a beam mode, with constant stretch and curvatures, superposed on the axisymmetric shell modes so that ovalization of the cross section is admitted. The element has no flexibility in torsion so that the twisting of a pipe-bend section is ignored, and the rotations along the section secant are made equal by tying. Pipe-bend sections are coupled together and into straight beam elements by extensive use of special default tying types and are described later in this section. Thus, a complete pipe-bend might consist of several sections of elements tied together with each section being build up from several elements and all sharing a common "elbow" node and having the usual two nodes on the shell surface. All constitutive relations may be used with this element. This element cannot be used with CONTACT.

A pipe-bend section is shown in Figure 3-19. In the plane of the section (the z-r plane), there are several elements. At the first two nodes of each element of the section are the shell degrees of freedom associated with Element type 15:

where u and v are displacements in the z and r directions in the plane of the section. The third node is the same at all elements of the section and with this node are associated the beam modes:

Δu - normal motion of one end plane with the other end plane fixed.

$\Delta\phi$ - in-plane rotation of one end plane with the other end plane fixed, that is, rotation about the z-axis in Figure 3-19 (ϕ positive closes elbow).

$\Delta\psi$ - out-of-plane rotation of one end plane with the other end plane fixed (ψ positive gives tension in $z>0$ in Figure 3-19).

It is assumed that these motions create a stress state in the section independent of position around the bend.

The geometry of the elbow is defined in two ways: in the section and in space. In the section, the geometry is input by placing the pipe surface nodes around the pipe in a z-r section, so that the axis of the pipe-bend torus is on $r = 0$, and the pipe center line is on $z = 0$.

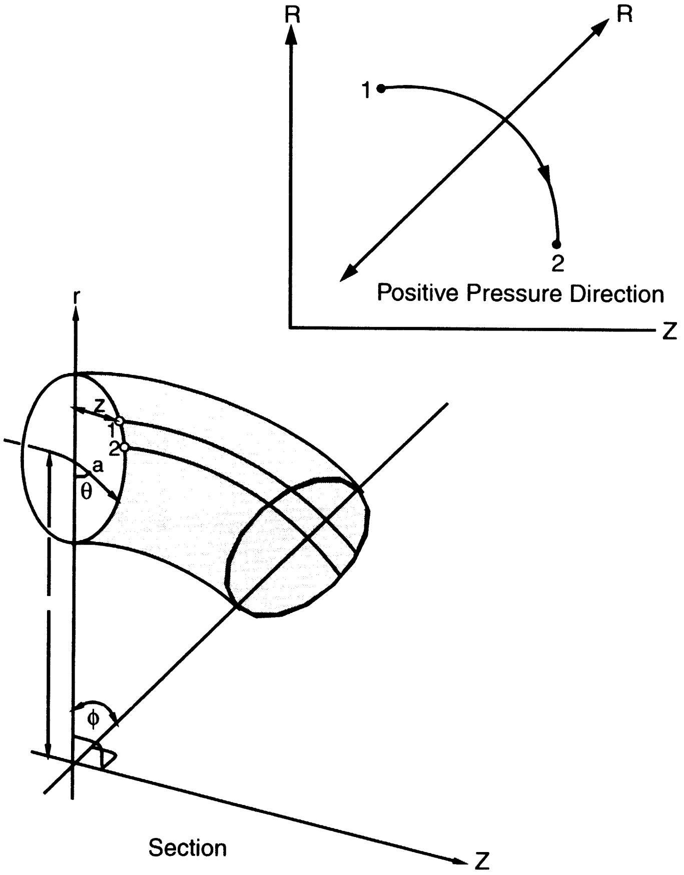


Figure 3-19 Typical Pipe-Bend Section

At each pipe surface node, the coordinates are those for Element type 15:

$$z, r, \frac{dz}{ds}, \frac{dr}{ds}, s$$

where s is the distance around the pipe surface. Notice that since the section is close, there is discontinuity in s (s = 0 is the same point as s = 2πa). Two nodes must be placed at this point and constrained to the same displacement by tying. Notice also that it is permissible to have unequal sized elements in this plane.

The geometry of the section in space is defined by the GEOMETRY option as follows. For each element on the section:

EGEOM1 = pipe thickness.

EGEOM2 = angle ϕ (Figure 3-19) in degrees; i.e., angular extent of pipe-bend section around the pipe-bend torus.

EGEOM3 = torus radius; i.e., radius to center of pipe in r-z plane.

There are no coordinates associated with the "elbow" (shared) node of the section.

The section is oriented in space and linked to other such sections or straight pipes by the introduction of additional nodes placed at the actual location in space of the center point of the ends of the section. These nodes are not associated with any element, but serve to connect the complete pipeline through the special default tying types developed for this element and described below. Each of these nodes has six coordinates. The first three are its (x,y,z) position in space and the second three are the (x,y,z) position of the center of the pipe-bend torus with which the node is associated.

Tying of the Pipe-Bend Section

Two tying types are required to complete a pipe-bend section. A further tying is required to link the section to the two nodes introduced at the ends of the section. This latter is not necessary if the pipe-bend section is used alone; e.g., to study nonlinear behavior of an individual component under in-plane loadings (see Volume A). The data required for each of the above tying types are described below:

A. The pipe must be closed around the discontinuity in s (in the z-r section) by tying all degrees of freedom at the last shell node of the section to the corresponding degrees of freedom at the first shell node of the section. This can be achieved by specifying tying type 100, with one retained node. The last node on the section ($s = 2\pi a$) is given as tied node, and the first node on the section ($s = 0$) is given as the retained node.

B. The rigid body modes in the r-z section must be eliminated. This is achieved by the special default tying type 16, which ensures that the integrated u- and v-displacements around the section are zero, based on the cubic displacement assumption in this plane within each element. This tying drops u and v at the first node of the section in terms of the four shell degrees of freedom at all other nodes of the section and $\frac{du}{ds}$, $\frac{dv}{ds}$ at the first node, according to:

$$\bar{u} = o = \frac{1}{(n= 1)}\{u_1 + u_2 + \ldots + u_{n-1} + \frac{1}{12}(s_2 - s_1 - s_n + s_{n-1})\frac{du_1}{ds} +$$

$$(s_1 - 2s_2 + s_3)\frac{du_2}{ds} + \ldots + (s_{n-2} - 2s_{n-1} + s_n)\frac{du_{n-1}}{ds}\}$$

C. with n = number of nodes on the section. Notice that this tying specifies v = 0. Net radial motion of the torus section is accounted for by the first degree of freedom at the elbow (shared) node of the section. This means that load coupling between these nodes must be created externally by the user: for example, with pressure in the pipe, the net out-of-balance force on the section must be appropriately applied at the first degree of freedom of the "elbow" node of the section.

The data defining this tying type are: tying type 16, with the number of retained nodes equal to the number of shell nodes in the z-r plane of the section (counting the two coincident end nodes separately). The tied node is the first shell node of the section (at s = 0) and the list of retained nodes gives all other shell nodes of the section in order of increasing s (including the end node at s = 2πa); then, lastly, the first shell node of the section again.

D. For out-of-plane bending, the low stiffness mode associated with the section rotating about the beam axis should be removed. This is done by dropping the u_z degree of freedom at the second node on the section according to:

$$\sum_{n_i = 1}^{n} u_i^t = 0$$

E. where u^t is the displacement tangent to the pipe wall in the section,

$$u^t = u_z \cos\phi + u_R \sin\phi$$

F. The data for this tying type are: tying type 15, number of retained nodes equal to one less than that used for tying type 16. The tied node is the second shell node of the section and the retained nodes are the first, third, fourth, etc., up to the next to the last node of the section; then, lastly, repeat the tied node.

G. If the section is being tied to the two external nodes introduced at the center points of its end planes, tying type 17 must be used with two retained nodes and one of the external nodes as tied node, and the other external node, then, the "elbow" node of the section as retained nodes. This tying removes all degrees of freedom at the external node given as tied node. Care must be taken with multiple-section bends to ensure these nodes are removed in an appropriate order.

Basis of the Element

The element is based on a superposition of purely axisymmetric strains generated by the shell modes, and the beam bending strains introduced through the degrees of freedom at the "elbow" node. Defining the latter as Δu, $\Delta\psi$ and $\Omega\chi$ for stretch, relative in-plane and out-of-plane rotations, respectively. An additional strain is superposed on the circumferential strain of the axisymmetric shell:

$$\varepsilon^{22} = \left[\frac{\Delta u}{r} + \Delta\phi\left(1 - \frac{\bar{r}}{r}\right) + \frac{z}{r}\Delta\Psi\right]\frac{1}{\phi}$$

where $\bar{r}$, z, and ϕ are defined in Figure 3-19.

Since $\frac{\Delta u}{\phi}$ causes the same strain as a mean radial motion of the torus in the (z-r) plane, the

constraint $\bar{v} = \frac{1}{2\pi}\int_{o}^{2\pi} v d\theta = 0$ is applied. Similarly, $\bar{u} = \frac{1}{2\pi}\int_{o}^{2\pi} u d\theta = 0$ is applied to remove this rigid body mode.

The pipe-bend section is linked externally by coupling relative motions of the external nodes introduced at the centers of the end planes to the relative displacements used as degrees of freedom at the "elbow" node of the section. Referring to Figure 3-20, in terms of the local set:

$$(u_x, u_y, u_z, \phi_x, \phi_y, \phi_z)$$

introduced in the local coordinates (x^1, y^1, z^1) at the two external nodes A and B and $\Delta\phi$, $\Delta\phi$, $\Delta\psi$ at the "elbow" node, these couplings are:

$$u_x^B = u_x^A \cos\phi = u_y^A \sin\phi - \theta_z^A \bar{r}(1 - \cos\phi) + \Delta u + \frac{\bar{r}}{\phi}(1 - \sin\phi)\Delta\phi$$

$$u_y^B = u_x^A \sin\phi + u_y^A \cos\phi + \theta_z^A \sin\phi - \frac{\bar{r}}{\phi}(1 - \cos\phi)\Delta\phi$$

$$u_z^B = u_z^A - \theta_x^A \bar{r}(1-\cos\phi) - \theta_y^A \bar{r} \sin\phi - \bar{r}\ \phi\ \Delta\psi$$

$$\theta_y^B = \theta_y^A + \Delta\psi$$

$$\theta_z^B = \theta_z^A - \Delta\phi$$

Additionally, in order to provide some coupling of the θ_x rotations, the rotation along the secant is made the same at each end:

$$\theta_x^B \cos\frac{\phi}{2} + \theta_y^B \sin\frac{\phi}{2} = \theta_x^A \cos\frac{\phi}{2}\ \theta_y^A \sin\frac{\phi}{2}$$

These constraints are transformed internally to corresponding constraints in terms of the six degrees of freedom at A and B in the global coordinate system.

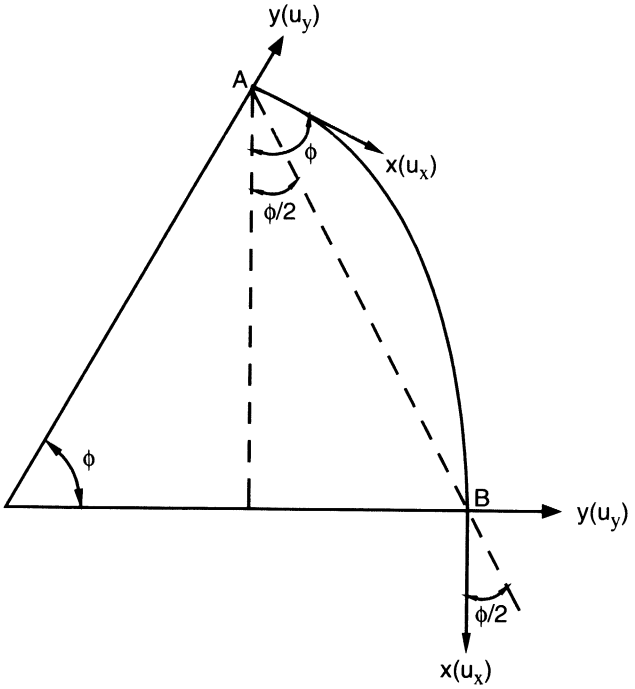


Figure 3-20 Pipe-Bend Section, Relative Motion Couplings

Quick Reference

Type 17

Constant bending, three-node elbow element.

Connectivity

Three nodes per element. Last node is a common point for the bend.

Geometry

For each element section, the geometry is defined as follows:

> The pipe thickness is input in the first data field (EGEOM1).
>
> The angular extent of the pipe-bend section around the pipe-bend truss is input in the second data field (EGEOM2), in degrees.
>
> The radius to the center of the pipe in the r-z plane is input in the third data field (EGEOM3).
>
> No coordinates associated with the common node.

Coordinates

At each pipe surface node, the coordinates are those for Element 15:

$$z, r \frac{dz}{ds}, \frac{dr}{ds}, s$$

where s is the distance around the pipe surface.

Degrees of Freedom

The degrees of freedom for the first two nodes are:

$$u, v, \frac{du}{ds}, \frac{dv}{ds}$$

For the common third node, they are in-plane stretch and out-of-plane curvature of the section.

Tractions

Distributed load selected with IBODY are as follows:

Load Type (IBODY)	Description
o	Uniform pressure.
1	Uniform load in 1 direction.
2	Nonuniform load in 1 direction.
3	Uniform load in 2 direction.
4	Nonuniform load in 2 direction.
5	Nonuniform pressure.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure assumed positive in the direction of normal obtained by rotation of -90° from direction of increasing s (see Figure 3-19).

In the nonuniform cases (IBODY-3, 4, or 5), the load magnitude must be supplied by user subroutine FORCEM.

Output of Strains

Output of strain on the center line of the element is:

1 = meridional membrane
2 = circumferential membrane
3 = meridional curvature
4 = circumferential curvature

Output of Stresses

Output of stress is at points through the thickness. The stresses are given in pairs as:

1 = meridional stress
2 = circumferential stress

proceeding from the top surface (up the positive local normal, see Figure 3-19) to the bottom surface in equal divisions.

Transformation

All degrees of freedom will be transformed to local directions.

Shell Transformation

The shell transformation option type 1 may be used for the pipe surface nodes. It permits an easier application of symmetry conditions in case the user wants to use a half section only. For a description of this transformation type, see Volume A. Note that if the FOLLOW FORCE option is invoked, the transformations will be based on the updated configuration of the element.

Tying

Special tying to form complete pipeline available with Element type 14.

Output Points

Centroid or three Gaussian integration points. The first Gaussian point is closest to first node of the element. The second Gaussian point is at the centroidal section. The third Gaussian point is closest to the second node.

Plotting

This element may be plotted, undeformed or deformed circumference. The tying data must be included with the connectivity and coordinate data.

Notes: For dynamics, the element mass matrix is only associated with the ovalization degree of freedom in the (r-z) plane. An equivalent straight beam (element type 14) should be used across the elbow segment, with zero stiffness (set Young's modulus to zero) and the same density to obtain the mass terms associated with the beam modes of elbow movement. With this approach, the element provides satisfactory modeling for dynamic as well as static response.

Adjacent elbow segments should have the same curvature and lies in the same plane. A short straight segment (element 14) may be used as a link in situations where this is not the case.

As to torus radius increases, the element exhibits less satisfactory behavior; the extreme case of modeling a straight pipe with this element should, therefore, be avoided.

Section Stress Integration

Use SHELL SECT parameter to set number of points for Simpson rule integration through the thickness. Three points are enough for linear material response. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.g., dynamic plasticity). The default is 11 points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 18

Four-Node, Isoparametric Membrane

Element 18 is a four-node, isoparametric, arbitrary quadrilateral written for membrane applications. As a membrane has no bending stiffness, the element is very unstable.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as 30. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive models may be used with this element.

This element is usually used with the LARGE DISP parameter, in which case the (tensile) initial stress stiffness increase the rigidity of the element.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface will form a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$, which for the centroid are defined in the following way: (see Figure 3-21).

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \eta}\right| , \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$\underset{\sim}{s} = \underset{\sim}{t}_1 + \underset{\sim}{t}_2 , \underset{\sim}{d} = \underset{\sim}{t}_1 - \underset{\sim}{t}_2$$

After normalizing these vectors by:

$$\bar{\underset{\sim}{s}} = \underset{\sim}{s}/\sqrt{2}|\underset{\sim}{s}| \quad \bar{\underset{\sim}{d}} = \underset{\sim}{d}/\sqrt{2}|\underset{\sim}{d}|$$

The local orthogonal directions are then obtained as:

$$\underset{\sim}{V}_1 = \bar{\underset{\sim}{s}} + \bar{\underset{\sim}{d}}$$

$$\underset{\sim}{V}_2 = \bar{\underset{\sim}{s}} - \bar{\underset{\sim}{d}}$$

and

$$\underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

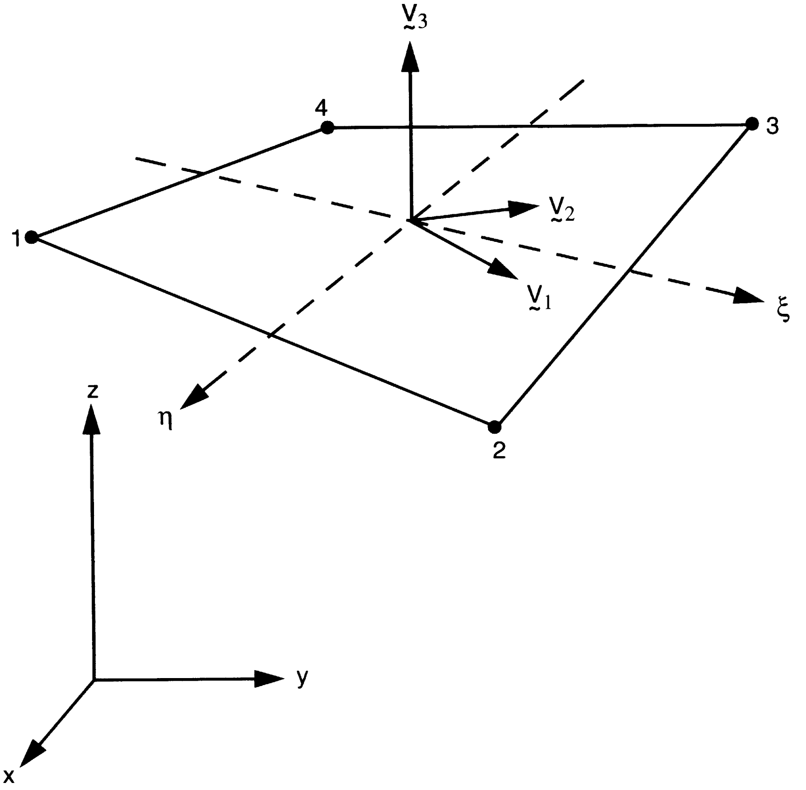


Figure 3-21 Form of Element 18

In this way, the vectors $\frac{\partial \underset{\sim}{x}}{\partial \xi}, \frac{\partial \underset{\sim}{x}}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Quick Reference

Type 18

Four-node membrane element (straight edge) in three-dimensional space.

Connectivity

Four nodes per element (see Figure 3-21).

Geometry

The thickness is input in the first data field (EGEOM1). The other two data fields are not used.

Coordinates

Three global Cartesian coordinates x, y, z.

Degrees of Freedom

Three global degrees of freedom u, v, and w.

Tractions

Four distributed load types are available, depending on the load type definition:

Load Type	Description
1	Gravity load, proportional to surface area, in negative global z direction.
2	Pressure (load per unit area) positive when in direction of normal $\underset{\sim}{V}_3$.
3	Nonuniform gravity load, proportional to surface area, in negative global z direction (use FORCEM).
4	Nonuniform pressure, proportional to surface area, positive when in direction of normal $\underset{\sim}{V}_3$ (use FORCEM).
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains and Stresses

Output of stress and strain is in the local ($\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$) directions defined above.

Transformation

Nodal degrees of freedom may be transformed to local degrees of freedom.

Output Points

Centroid or four Gaussian integration points (see Figure 3-22).

Notes: Sensitive to excessive distortions, use essentially a rectangular mesh.

An eight-node membrane distorted quadrilateral (Element 30) is available and is the preferred element.

Membrane analysis is extremely difficult due to rigid body modes. For example, a circular cylinder shape is particularly numerically sensitive.

This element has no bending stiffness.

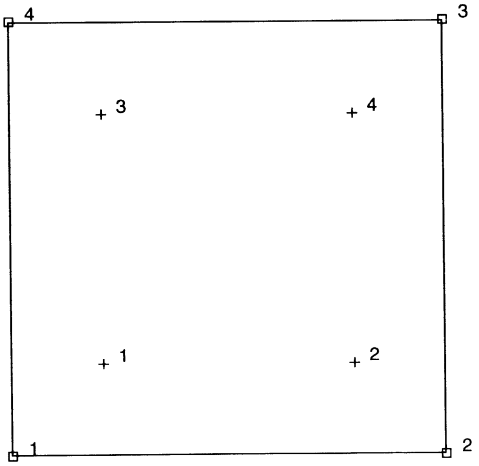


Figure 3-22 Gaussian Integration Points for Element 18

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element.

Element 19

Generalized Plane Strain Quadrilateral

This element is an extension of the plane strain isoparametric quadrilateral (Element type 11) to the generalized plane strain case. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the to p surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

In general, one needs more of these lower-order elements than the higher-order elements such as types 29 or 56. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 82 instead. Element type 81 is also preferable for small strain incompressible elasticity.

These elements cannot be used with the element-by-element iterative solver.

Quick Reference

Type 19

Generalized plane strain quadrilateral.

Connectivity

Node Numbering:

First four nodes are the corners of the element in the x-y plane, and must proceed counter-clockwise around the element when the x-y plane is viewed from the positive z side.

Fifth and sixth nodes are shared by all generalized plane strain elements in this part of the structure. These two nodes should have the highest node numbers in the generalized plane strain part of the structure, to reduce matrix solution time (see Figure 3-23).

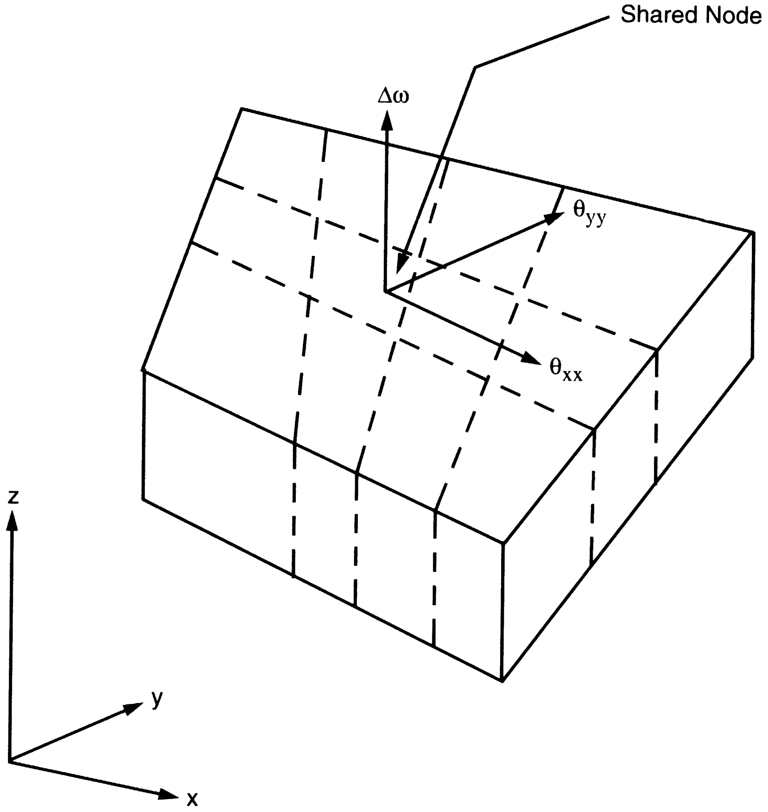


Figure 3-23 Generalized Plane Strain Element

Geometry

The thickness is entered in the first data field (EGEOM1). Default is unit thickness.

If a nonzero value is entered in the second data field (EGEOM2), the volumetric strain will be constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution. For details see Volume F.

Coordinates

Coordinates are X and Y at all nodes. Note the position of the first shared node (node 5 of each element) determines the point where the thickness change will be measured. The user chooses the location of nodes 5 and 6. These nodes should be at the same location in space. The thickness is defined by the geometry block.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement (parallel to x-axis)
2 = v displacement (parallel to y-axis)

at all nodes except the two shared nodes (nodes 5 and 6 of each element).

For the first shared node (node 5 of each element):

1 = Δw = thickness change at that point
2 – is not used

For the second shared node (node 6 of each element):

1 = $\Delta\theta_{xx}$ = relative rotation of top surface of generalized plane strain section of structure, with respect to its bottom surface, about the x-axis.

2 = $\Delta\theta_{yy}$ = relative rotation of the top surface with respect to the bottom surface about the y-axis.

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.

Load Type	Description
* 3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
* 20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
* 21	Nonuniform shear force on side 1 - 2; magnitude supplied through user subroutine FORCEM.
* 22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2 - 3; magnitude supplied through user subroutine FORCEM.
* 24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3 - 4; magnitude supplied through user subroutine FORCEM.
* 26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4 - 1; magnitude supplied through user subroutine FORCEM.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads may be applied at the nodes.

Output of Strains and Stresses

Output of strain and stress:

$1 = \varepsilon_{xx}$ (σ_{xx})
$2 = \varepsilon_{yy}$ (σ_{yy})
$3 = \varepsilon_{zz}$ (σ_{zz})
$4 = \gamma_{xy}$ (τ_{xy})

Transformation

Only in the x-y plane.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points.

Updated Lagrange Procedure And Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Thickness will be updated. Reduced volume strain integration is recommended – see **Geometry**.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 20

Axisymmetric Torsional Quadrilateral

Element type 20 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications including torsional strains. It is assumed that there are no variations in behavior in the circumferential direction.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior.

In general, one needs more of these lower order elements than the higher order elements such as types 67 or 73. Hence, use a fine mesh.

This element is preferred over higher order elements when used in a contact analysis. Note, in a contact analysis, there is no friction associated with the torsion.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 82 instead. Element type 83 is also preferable for small strain incompressible elasticity.

Notice that there is no friction contribution in the torsional direction when the CONTACT option is used.

Quick Reference

Type 20

Axisymmetric, arbitrary, ring with a quadrilateral cross section.

Connectivity

Four nodes per element. Numbering in a right-handed manner (counterclockwise).

Geometry

If a nonzero value is entered in the second data field (EGEOM2), the volume strain will be constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution. For details, see Volume F, Reference XXV.

Coordinates

Two coordinates in the global z- and r-directions respectively.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement (along symmetric axis)
2 = radial displacement
3 = angular displacement about symmetric axis (measured in radians)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through subroutine FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through subroutine FORCEM.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform pressure on 3-4 face of the element.

Load Type	Description
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear force on side 1 - 2; magnitude supplied through user subroutine FORCEM.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3; magnitude supplied through user subroutine FORCEM.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4; magnitude supplied through user subroutine FORCEM.
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1; magnitude supplied through user subroutine FORCEM.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.
102	Gravity loading in global direction. Enter the magnitude of gravity acceleration in the z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. In addition, point loads and torques may be applied at the nodes. The magnitude of concentrated loads must correspond to the load integrated around the circumference.

Output of Strains

Output of strains at the centroid of the element in global coordinates:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{zr}$
$5 = \gamma_{r\theta}$
$6 = \gamma_{\theta z}$

Output of Stresses

Output for stress is the same direction as for **Output of Strains**.

Transformation

First two degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points (see Figure 3-24).

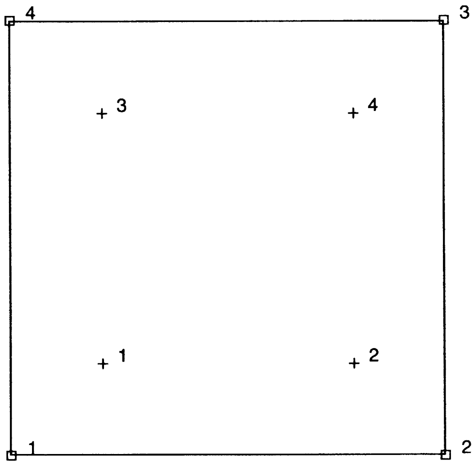


Figure 3-24 Integration Points

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration is recommended – see **Geometry**.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 21

Three-Dimensional 20-Node Brick

Element type 21 is a 20-node, isoparametric, arbitrary hexahedral. This element uses triquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower -order elements, such as type 7, are preferred in contact analyses.

The stiffness of this element is formed using 27-point Gaussian integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 35 instead. Element type 35 is also preferable for small strain incompressible elasticity.

Note: Reduction to Wedge or Tetrahedron – By simply repeating node numbers on the same spatial position, the element may be reduced as far as a tetrahedron. Element type 127 is preferred for tetrahedrals.

QUICK REFERENCE

Type 21

Twenty-nodes, isoparametric, arbitrary, distorted cube.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-25.

Geometry

In general, not required. The first field contains the transition thickness when the automatic brick to shell transition constraints are used (see Figure 3-26). In a coupled analysis, there are no constraints for the temperature degrees of freedom.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v and w.

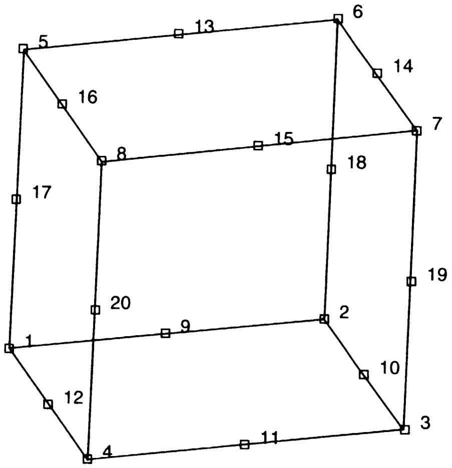


Figure 3-25 Form of Element 21

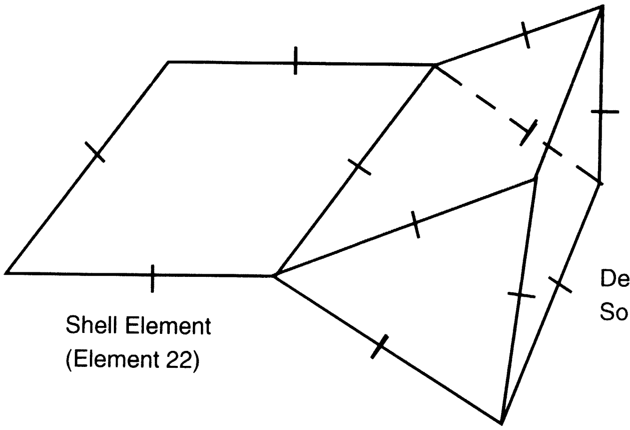


Figure 3-26 Shell-to-Solid Automatic Constraint

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through user subroutine FORCEM)
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in FORCEM.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in FORCEM.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in FORCEM.

Load Type	Description
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in FORCEM.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in FORCEM.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.

Load Type	Description
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified on ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

The subroutine FORCEM is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the twenty-seven integration points.

For nonuniform surface pressure, force values need only be supplied for the nine integration points on the face of application. Nodal (concentrated) loads may also be supplied.

Output of Strains

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \gamma_{yz}$
$6 = \gamma_{zx}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS. An automatic constraint is available for brick-to-shell transition meshes. (See **Geometry**.)

Note: There is an automatic constraint option for transitions between bricks and shells in element type 22. By collapsing a one-sided plane to a line as shown in Figure 3-26, this transition is created. Thickness of the shell must be specified in the GEOMETRY field of the brick element.

Output Points

Centroid or 27 Gaussian integration points (see Figure 3-27).

Notes: A large bandwidth results in a lengthy central processing time.

The user should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

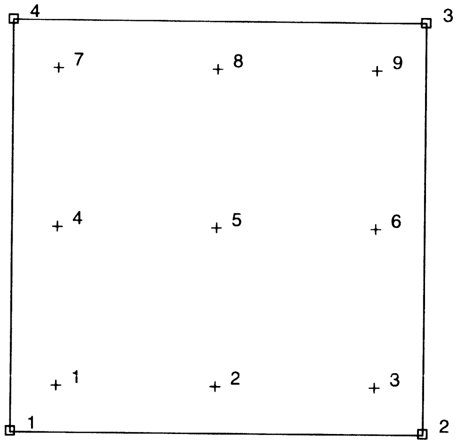


Figure 3-27 Element 21 Integration Plane

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note: Distortion of element during analysis may cause bad solution. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 44. See Element 44 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 22

Quadratic Thick Shell Element

Element type 22 is an eight-node thick shell element with global displacements and rotations as degrees of freedom. Second-order interpolation is used for coordinates, displacements and rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at ten special points and interpolated to the integration points. In this way, this element behaves correctly in the limiting case of thin shells. The element can be degenerated to a triangle by collapsing one of the sides. Tying type 22, which connects shell and solid, is available for this element.

Lower-order elements, such as type 75, are preferred in contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive relations may be used with this element.

Geometric Basis

The element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. The stress output is given with respect to local orthogonal surface directions, $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$ for which each integration point is defined in the following way (see Figure 3-28).

At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized:

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \eta}\right|, \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now, a new basis is being defined as follows:

$$\underset{\sim}{s} = \underset{\sim}{t}_1 + \underset{\sim}{t}_2, \underset{\sim}{d} = \underset{\sim}{t}_1 - \underset{\sim}{t}_2$$

After normalizing these vectors by $\bar{\underset{\sim}{s}} = \underset{\sim}{s}/\sqrt{2}|\underset{\sim}{s}|$ $\quad \bar{\underset{\sim}{d}} = \underset{\sim}{d}/\sqrt{2}|\underset{\sim}{d}|$, the local orthogonal directions are then obtained as follows:

$$\underset{\sim}{V}_1 = \bar{\underset{\sim}{s}} + \bar{\underset{\sim}{d}}$$

$$\underset{\sim}{V}_2 = \bar{\underset{\sim}{s}} - \bar{\underset{\sim}{d}}$$

and

$$\underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

In this way, the vectors $\frac{\partial \underset{\sim}{x}}{\partial \xi}, \frac{\partial \underset{\sim}{x}}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

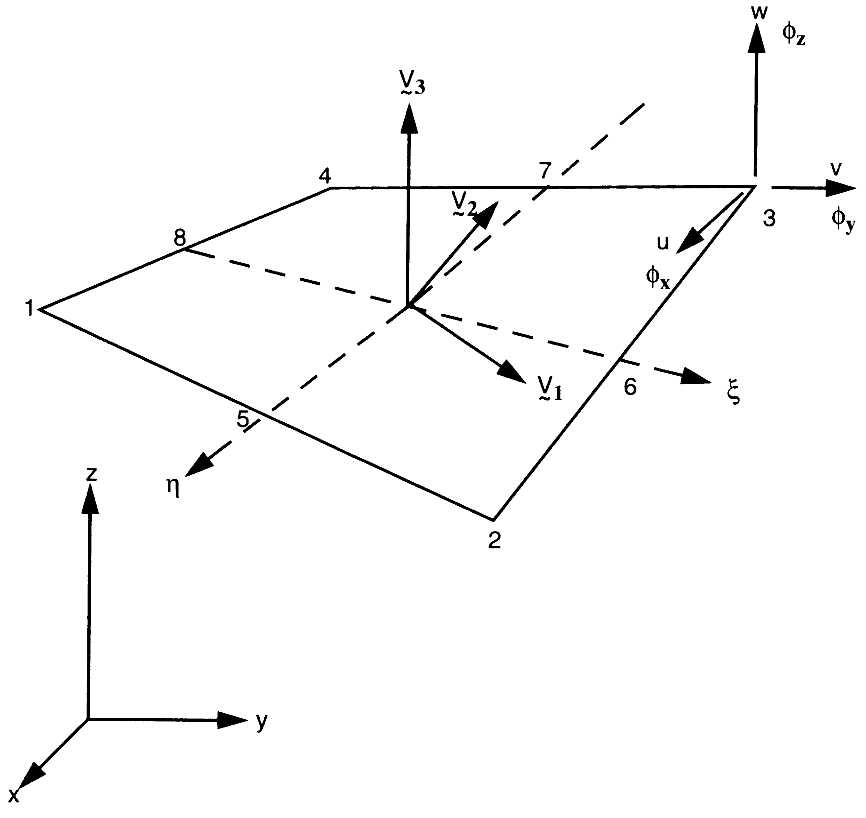


Figure 3-28 Form of Element 22

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x-, y-, and z-axis, respectively.

Quick Reference

Type 22

Bilinear, eight-node shell element including transverse shear effects.

Connectivity

Eight nodes per element. The element may be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2 = EGEOM3 = EGEOM4 = 0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

Distributed load types follow below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction; magnitude given in user subroutine FORCEM.
4	Nonuniform pressure with positive magnitude in -V_3-direction; magnitude given in user subroutine FORCEM.
5	Nonuniform load per surface area in arbitrary direction; magnitude given in user subroutine FORCEM.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load; magnitude given in FORCEM in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load; magnitude and direction given in FORCEM on 1-2 edge.
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load; magnitude given in FORCEM in the plane of the surface on 2-3 edge.
23	Nonuniform edge load; magnitude and direction given in FORCEM on 2-3 edge.
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load; magnitude given in FORCEM in the plane of the surface on 3-4 edge.
33	Nonuniform edge load; magnitude and direction given in FORCEM on 3-4 edge.
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load; magnitude given in FORCEM in the plane of the surface on 4-1 edge.
43	Nonuniform edge load; magnitude and direction given in FORCEM on 4-1 edge.

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified on ROTATION A option.
102	Gravity loading in global direction. Enter 3 magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads and moments may also be applied at the nodes.

Output of Strains

Generalized strain components are as follows:

Middle surface stretches: ε_{11} ε_{22} ε_{12}
Middle surface curvatures: κ_{11} κ_{22} κ_{12}
Transverse shear strains: γ_{23} γ_{31}

in local (V_1, V_2, V_3) system.

Output of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local (V_1, V_2, V_3) system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at each node may be transformed to local direction.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, the user has to select his loadsteps such that the rotation remains small within a load step. Thickness will only be updated if the FINITE parameter is specified.

Section Stress – Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 86. See Element 86 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Design Variables

The thickness can be considered as a design variable.

Element 23

Three-Dimensional 20-Node Rebar Element

This element is an isoparametric, three-dimensional, 20-node empty brick in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 20-node brick continuum element (e.g., elements 21, 35, 57, or 61) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element faces (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element faces to its opposite one. For instance (see Figure 3-29), if the layer is similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-29). At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

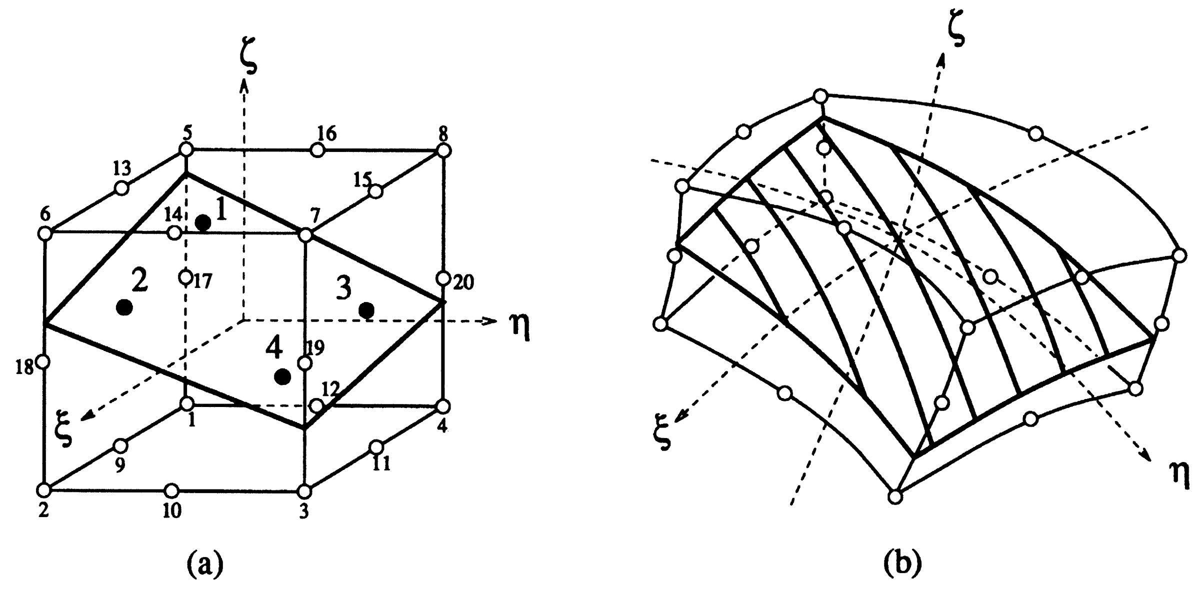


Figure 3-29 Twenty-Node Rebar Element

Quick Reference

Type 23

Twenty-node, isoparametric rebar element to be used with 20-node brick continuum element.

Connectivity

Twenty nodes per element. Node numbering of the element is same as that for elements 21, 35, 57, or 61.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, set to either 1, 2, or 3:

1. Rebar layers are similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element (see Figure 3-29).

Midside Nodes: $\frac{\partial u}{\partial n}, \frac{\partial v}{\partial n}, \frac{\partial w}{\partial n}$ (3 degrees of freedom)

where (u, v, w) are the Cartesian components of displacement and n is normal distance in the $\theta^1 - \theta^2$ plane; n has positive projection on the θ^1 axis ($n^1>0$) or, if $n^1 = 0$, $n^2>0$.

The De Veubeke interpolation function ensures continuity of all displacement components and their first derivatives $\frac{\partial u_i}{\partial \theta^1}, \frac{\partial u_i}{\partial \theta^2}$ between elements when the above sets of nodal degrees of freedom take identical values at shared nodes on element edges.

Care should be taken with the application of kinematic boundary conditions since they must be fully, but not over-fully, specified and in the application of edge moments, so that they are conjugate to the appropriate degrees of freedom and so generate a mechanical work rate.

Connectivity

The node numbering order is given in Figure 3-30. Corners should be numbered first, in counterclockwise order in the $\theta^1 - \theta^2$ plane, then midside nodes, the fifth node being between the first and second, etc.

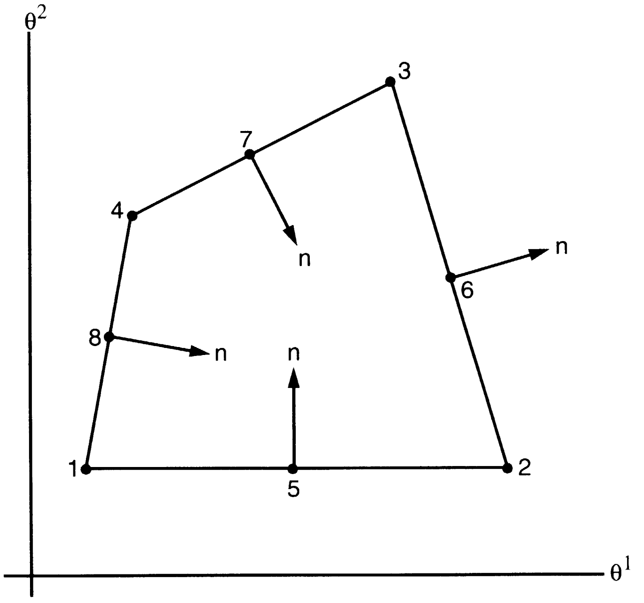


Figure 3-30 Definitions of the Positive Midside Normal Direction

Numerical Integration

The element uses a 28-point integration scheme, based on a 7-point, fifth order scheme in each of the contributing triangles. The locations of the points are shown in Figure 3-31. The distance ratios are $(1+\sqrt{15})/7$ and $(1-\sqrt{15})/7$ of the distance from the centroid to a vertex in each triangle.

Note that the density of integration points takes full advantage of the high order of the element.

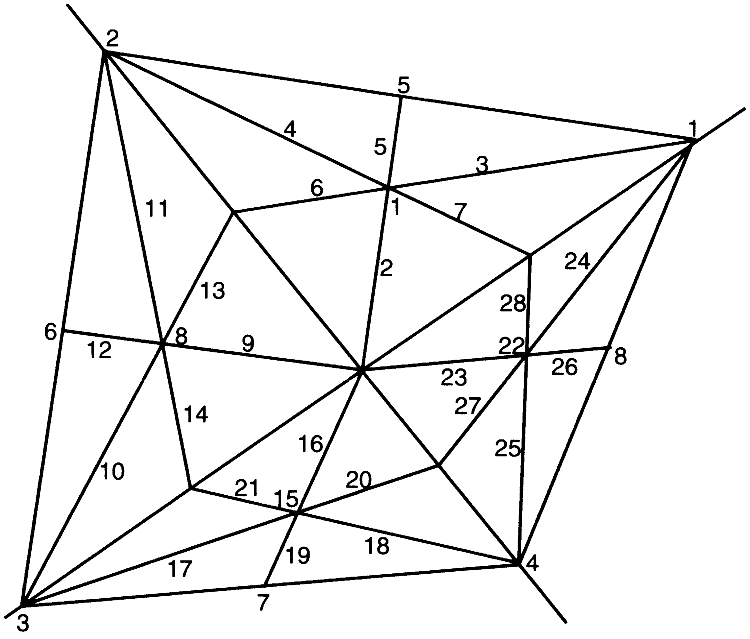


Figure 3-31 Integration Points for Element 24 (not to scale)

Quick Reference

Type 24

Arbitrary, doubly-curved, quadrilateral shell element.

Connectivity

Eight nodes per element. Four corner nodes and four midside nodes numbered as shown in Figure 3-30.

Geometry

The element thickness is input in the first data field (EGEOM1). The other two data fields are not used.

Coordinates

Eleven coordinates for nodes:

$1 = \theta^1;\qquad 2 = \theta^2$

$3 = x;\qquad 4 = \dfrac{\partial x}{\partial \theta^1};\qquad 5 = \dfrac{\partial x}{\partial \theta^2}$

$6 = y;\qquad 7 = \dfrac{\partial y}{\partial \theta^1};\qquad 8 = \dfrac{\partial y}{\partial \theta^2}$

$9 = z;\qquad 10 = \dfrac{\partial z}{\partial \theta^1};\qquad 11 = \dfrac{\partial z}{\partial \theta^2}$

This coordinate set is input at all nodes.

Degrees of Freedom

Nine degrees of freedom for the corner nodes:

$1 = u;\qquad 2 = \dfrac{\partial u}{\partial \theta^1};\qquad 3 = \dfrac{\partial u}{\partial \theta^2}$

$4 = v;\qquad 5 = \dfrac{\partial v}{\partial \theta^1};\qquad 6 = \dfrac{\partial v}{\partial \theta^2}$

$7 = w;\qquad 8 = \dfrac{\partial w}{\partial \theta^2};\qquad 9 = \dfrac{\partial w}{\partial \theta^2}$

Three degrees of freedom for the midside nodes:

$1 = \dfrac{\partial u}{\partial n};\qquad 2 = \dfrac{\partial v}{\partial n};\qquad 3 = \dfrac{\partial w}{\partial n}$

n = normal to side in $\theta^1 - \theta^2$ plane, directed such that the projection of n on the θ^1 axis is positive, or, if the side is parallel to the θ^1-axis, n is in the direction of increasing θ^1.

Tractions

Point loads on nodes (care should be used with moments, since derivative degrees of freedom may not measure rotation in radians).

Distributed Loads

Type 1

Uniform self-weight (magnitude = load per unit surface area) in negative z direction.

Type 2

Uniform pressure in negative surface normal direction (surface normal $\underset{\sim}{a}_3 = \underset{\sim}{a}_1 \times a_2$, where $\underset{\sim}{a}_1$ and $\underset{\sim}{a}_2$ are base vectors along θ^1 and θ^2 surface coordinate lines).

Type 3

Nonuniform pressure in negative surface normal direction; magnitude specified by user subroutine FORCEM.

Type 4

Nonuniform load unit volume in arbitrary direction; magnitude and direction defined by user subroutine FORCEM.

This subroutine will be called once per integration point for all elements of type 24 listed with load type 4. For these elements the magnitude supplied in the TRACTIONS option (associated with load type 4) will be overwritten by the value defined in the subroutine.

Types 11-43

These loads represent edge loads per unit length, applied in any of the global (x, y, z) directions on any of the four edges of the element. The first digit of the load type chooses the edge, the second chooses the global force directions as follows:

Load Type	Description
11	Uniform load per unit length on the 1-2 edge in the x-direction.
12	Uniform load per unit length on the 1-2 edge in the y-direction.
13	Uniform load per unit length on the 1-2 edge in the z-direction.
21	Uniform load per unit length on the 2-3 edge in the x-direction.
22	Uniform load per unit length on the 2-3 edge in the y-direction.
23	Uniform load per unit length on the 2-3 edge in the z-direction.
31	Uniform load per unit length on the 3-4 edge in the x-direction.
32	Uniform load per unit length on the 3-4 edge in the y-direction.

Load Type	Description
33	Uniform load per unit length on the 3-4 edge in the z-direction.
41	Uniform load per unit length on the 4-1 edge in the x-direction.
42	Uniform load per unit length on the 4-1 edge in the y-direction.
43	Uniform load per unit length on the 4-1 edge in the z-direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Six generalized strain components, three surface stretches and three surface curvature changes, are given as components with respect to the surface coordinate system.

Output of Stresses

σ^{11}, σ^{22}, σ^{12}, physical components of stress in the $(\theta^1 - \theta^2)$ directions at points through the thickness of the shell. First point is on surface in the direction of positive surface normal. The last point is on opposite surface.

Transformation

Cartesian components of displacement and displacement derivatives may be transformed to local directions. Surface coordinate directions remain unchanged.

Special Transformation

The shell transformation options type 2 (for the corner nodes) and type 3 (for the midside nodes) may be used to permit easier application of point loads, moments and/or boundary conditions on a node. For a description of these transformation types, see Volume A. Note that if the FOLLOW FOR parameter is invoked, the transformation will be based on the updated configuration of the element.

Tying

No special type of tying. Use subroutine UFORMS.

Output Points

Centroid or 28 integration points as shown in Figure 3-31.

Notes: Quadrilaterals must not be re-entrant.

This is a very expensive element.

For buckling analysis, elements 4 and 8 are preferable due to the expense of element 24.

Section Stress Integration

Use SHELL SECT parameter to set number of points for Simpson rule integration through the thickness.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain as for total Lagrangian approach. Thickness will be updated.

Note: Shell theory only applies if strain variation over the thickness is small.

Coupled Analysis

Coupled analysis is not available for this element. Use element types 22, 72, 75, or 139.

Element 25

Thin-Walled Beam in Three-Dimensions

This is a straight beam element with no warping of the section, but including twist. It is obtained by modifying element type 14 to give a linear variation of strain along its axis. This improves the element for large displacement analysis and for cases where linear axial strain is necessary (e.g., thermal gradient along the axis).

The degrees of freedom associated with each node are three global displacements and three global rotations, all defined in a right-handed convention. In addition, a seventh degree of freedom measures the rates of change of displacement along the beam axis. The generalized strains are stretch, two curvatures and twist per unit length. Stress is direct axial and shear given at each point of the cross section. The local coordinate system which establishes the orientation of the cross section is defined in GEOMETRY fields 4, 5, and 6. Using the GEOMETRY option, a vector in the plane of the local x-axis must be specified. If no vector is defined through this option, the local coordinate system may alternatively be defined by giving the coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam towards this point. The default cross section is circular. The user may specify alternative cross sections through the BEAM SECT parameter.

All constitutive relations may be used with this element.

Quick Reference

Type 25

Closed-section beam; Euler-Bernoulli theory.

Connectivity

Two nodes per element (see Figure 3-32).

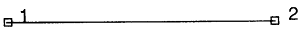


Figure 3-32 Two-Node Closed-Section Beam

Geometry

For the default section of a hollow circular cylinder, the first data field is for the thickness (EGEOM1).

For non-circular section, set EGEOM1 to 0.

For circular section, set EGEOM2 to the radius.

For noncircular section, set EGEOM2 to the section number needed.

EGEOM4-EGEOM6: Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis will lie on the same side as the specified vector.

Coordinates

The first three coordinates at each node are the global (x,y,z) coordinates.

The fourth, fifth, and sixth coordinates are the (x,y,z) coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point. The local x-axis is normal to the beam vector and is positive moving from the beam vector to the point. The fourth, fifth, and sixth coordinates will only be used if the local x-axis direction is not specified in the GEOMETRY block.

Degrees of Freedom

1 = u
2 = v
3 = w
4 = θ_x
5 = θ_y
6 = θ_z
7 = $d\bar{u}/ds$ (local)

Tractions

Distributed load types are as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via user subroutine FORCEM.

Load Type	Description
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads and moments may be applied at the nodes.

Output of Strains

Generalized strains:

1 = axial stretch
2 = K_{xx} = curvature about local cross-sectional x-axis
3 = K_{yy} = curvature about local cross-sectional y-axis
4 = ϕ = twist per unit length

Output of Stresses

Stresses: 1 = axial stress
2 = twisting shear stress

Transformation

Displacements and rotations at the nodes may be transformed to a local coordinate reference.

Tying

Use tying type 53 for fully moment-carrying joint. Use tying type 52 for pinned joint.

Output Points

Centroid or three Gaussian integration points.

Special Considerations

The seventh degree of freedom will only be shared between two adjacent elements when the beam section and properties are the same for both elements. Tying should be used in other cases. Elements of types 13, 14, 52, 76, 77, 78, 79, or 98 may be used together directly.

For all beam elements, the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout may be changed via the PRINT ELEMENT option.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available for this element. This element does not have a finite strain capability.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and/or the radius can be considered as design variables.

Element 26

Plane Stress, Eight-Node Distorted Quadrilateral

Element type 26 is an eight-node, isoparametric, arbitrary quadrilateral written for plane stress applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for a more accurate representation of the strain fields in elastic analyses than lower order elements.

Lower-order elements, such as type 3, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration.

All constitutive models may be used with this element.

Quick Reference

Type 26

Second-order, isoparametric, distorted quadrilateral. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-34.

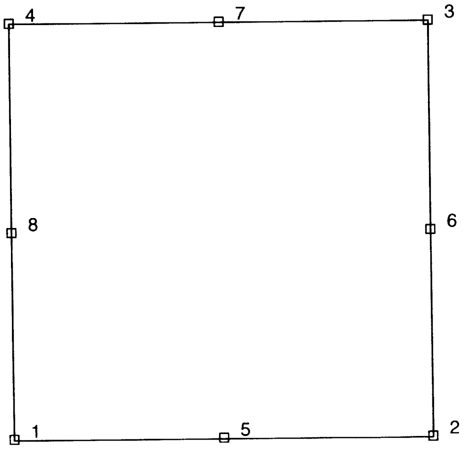


Figure 3-34 Nodes of Eight-Node, 2D Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.

Load Type (IBODY)	**Description**
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-35 and Output Points) in the following order:

$1 = \varepsilon_{xx}$, direct
$2 = \varepsilon_{yy}$, direct
$3 = \gamma_{xy}$, shear

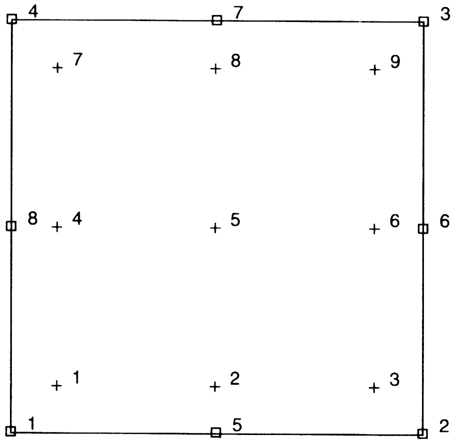


Figure 3-35 Integration Points of Eight-Node, 2D Element

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-35.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-35. This is the usual option for a second order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness will be updated.

Note: Distortion of element during analysis may cause bad solution. Element type 3 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element.

Element 27

Plane Strain, Eight-Node Distorted Quadrilateral

Element type 27 is an eight-node, isoparametric, arbitrary quadrilateral written for plane strain applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 11, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 32 instead. Element type 32 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 27

Second-order, isoparametric, distorted quadrilateral. Plane strain.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-36.

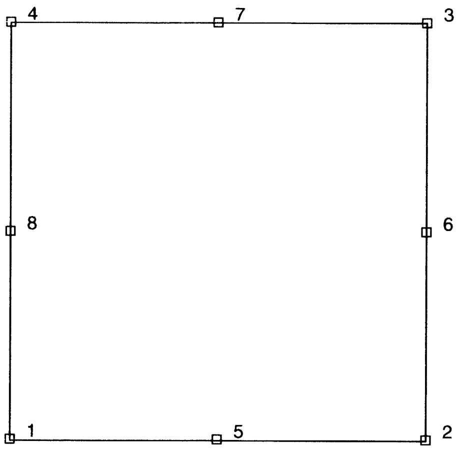


Figure 3-36 Nodes of Eight-Node, 2D Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.
21	Nonuniform shear force on side 1-5-2.

Load Type (IBODY)	Description
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Four strain components are printed in the order listed below:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy}, shear

Output of Stresses

Output for stresses is the same as for **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROIDS parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-37.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-37. This is the usual option for a second-order element.

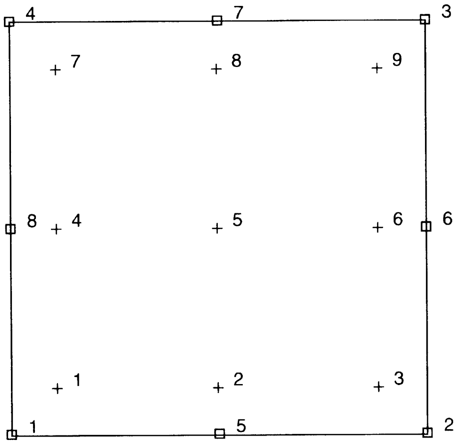


Figure 3-37 Integration Points of Eight-Node, 2D Element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates.

Note: Distortion of element during analysis may cause bad solution. Element type 6 or 11 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Element 28

Axisymmetric, Eight-Node Distorted Quadrilateral

Element type 28 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 10, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 33 instead. Element type 33 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 28

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-38.

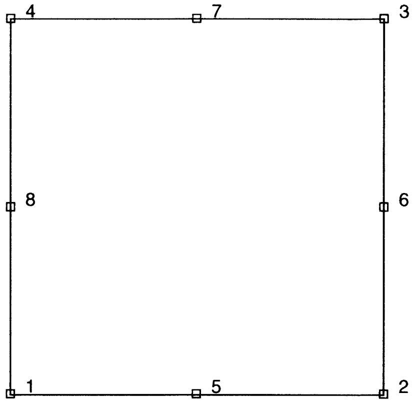


Figure 3-38 Nodes of Eight-Node Axisymmetric Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.
21	Nonuniform shear force on side 1-5-2.

Load Type	Description
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Concentrated nodal loads must be the value of the load integrated around the circumference.

Output of Strains

Four strain components are printed in the order listed below:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direct
4 = γ_{zr}, shear in the section

Output of Stresses

Output for stresses is the same as for Output of Strains.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-39.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 3-39. This is the usual option for a second-order element.

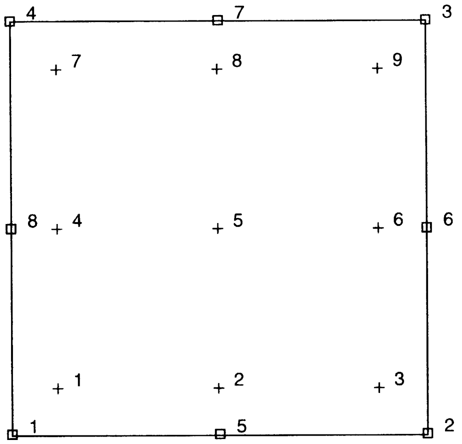


Figure 3-39 Integration Points of Eight-Node, Axisymmetric Element

Updating Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates.

Note: Distortion of element during analysis may cause bad solution. Element type 2 or 10 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 42. See Element 42 for a description of the conventions used for entering the flux and film data for this element.

■ Element 29

Generalized Plane Strain, Distorted Quadrilateral

This element is an extension of the plane strain isoparametric quadrilateral (element type 27) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

This element uses biquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 19, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 34 instead. Element type 34 is also preferable for small strain incompressible elasticity.

This element cannot be used with the element-by-element iterative solver.

Quick Reference

Type 29

Second-order, isoparametric, distorted quadrilateral. Generalized plane strain formulation.

Connectivity

Corners numbered first in a counterclockwise direction (right-handed convention in x-y plane). Then the fifth node between first and second; the sixth node between second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Global x and global y coordinate at each of the ten nodes. The ninth and tenth nodes may be anywhere in the (x, y) plane.

Degrees of Freedom

Two at each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

One at the ninth node:

1 = Δz = relative z-direction displacement of front and back surfaces.See Figure 3-40.

Two at the tenth node:

1 = $\Delta\theta_x$ = relative rotation of front and back surfaces about global x- axis.
2 = $\Delta\theta_y$ = relative rotation of front and back surfaces about global y-axis. (See Figure 3-40).

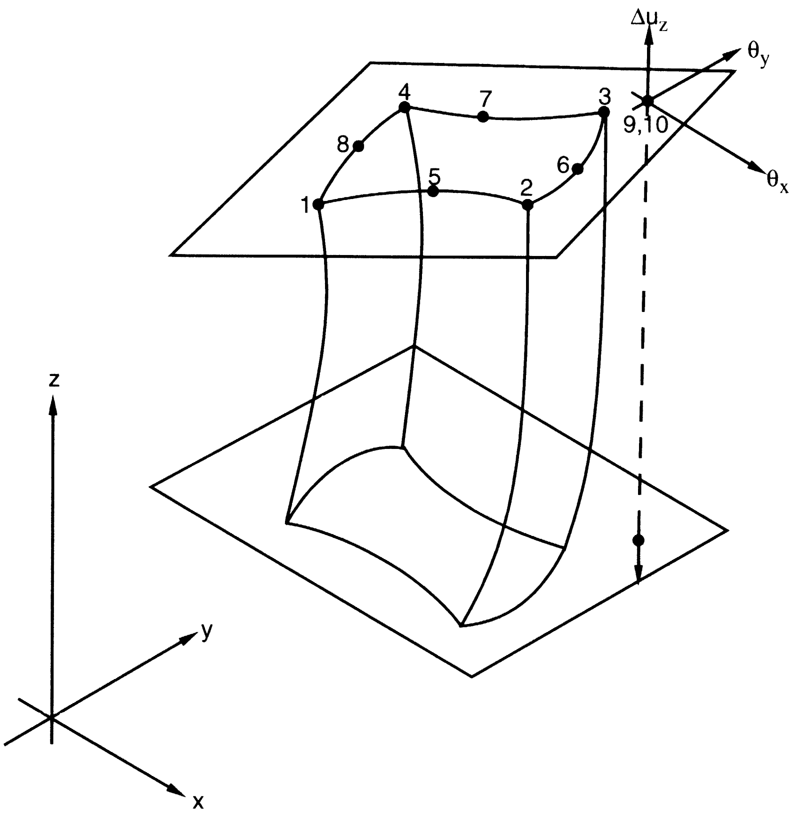


Figure 3-40 Generalized Plane Strain Distorted Quadrilateral

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.

Load Type	Description
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Strains are printed in the order listed below:

1 = ε_{xx}, direct
2 = ε_{xy}, direct
3 = ε_{zz}, thickness direction direct
4 = γ_{xy}, shear in the (x-y) plane

No γ_{xz}, γ_{yz}, or shear – relative rotations of front and back surfaces.

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-41.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 3-41. This is the usual option for a second order element.

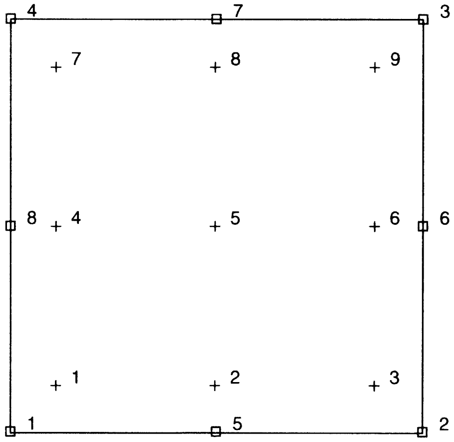


Figure 3-41 Integration Points of Eight-Node, 2D Element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates. Thickness will be updated.

Note: Distortion of element during analysis may cause bad solution. Element type 19 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 30

Membrane, Eight-Node Distorted Quadrilateral

Element type 30 is an eight-node, isoparametric, arbitrary quadrilateral written for membrane applications. As a membrane has no bending stiffness, the element is very unstable.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for a more accurate representation of the strain fields in elastic analyses than lower order elements.

Lower-order elements, such as type 18, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

All constitutive models may be used with this element.

This element is usually used with the LARGE DISP parameter, in which case the (tensile) initial stress stiffness increases the rigidity of the element.

Quick Reference

Type 30

Eight-node, second-order, isoparametric membrane element. Plane stress.

Connectivity

The corners are numbered first in a counterclockwise direction. The fifth node is located between nodes 1 and 2; the sixth node is located between nodes 2 and 3, etc.

Geometry

The thickness is input in the first data field (EGEOM1). Other fields are not used.

Coordinates

Three global coordinates (x, y, z) at each node.

Degrees of Freedom

Three at each node – (u, v, w) in the global rectangular Cartesian system.

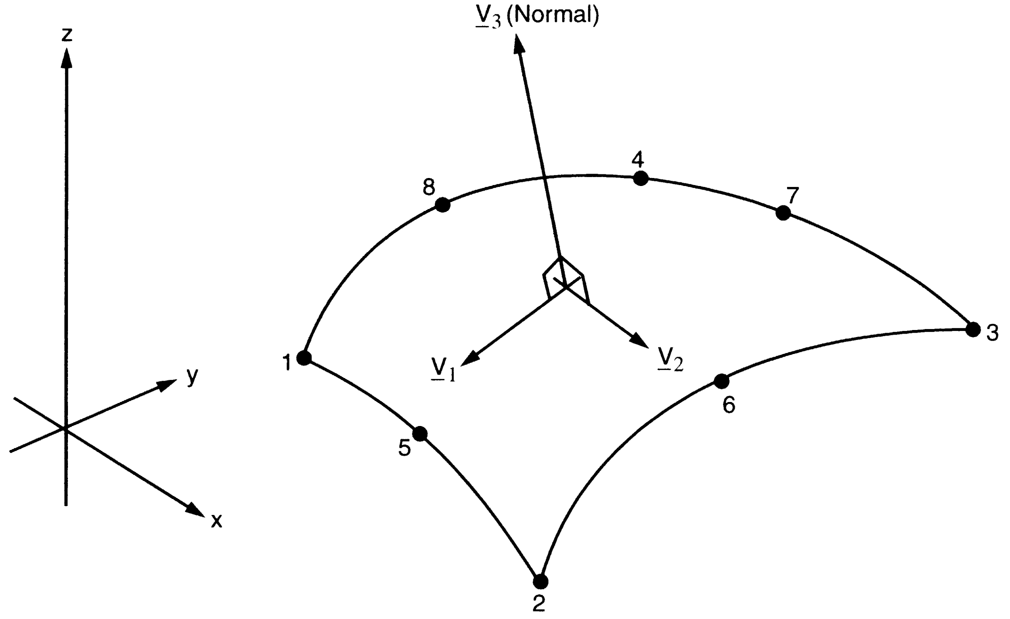


Figure 3-42 Eight-Node, Second Order Membrane Element

Tractions

Pressure

Pressure is specified as load type 2 and is positive in the direction of $\underline{V}_3$, the surface normal.

Self-weight

Self-weight is a force in the negative global z-direction proportional to surface areas. It is chosen as load type 1.

Nonuniform pressure

Specified as load type 4, positive in the direction of $\underline{V}_3$, the surface normal (use FORCEM).

Nonuniform self-weight

Specified as load type 3, force in the negative global z-direction, proportional to surface area (use FORCEM).

Output of Stresses

Output of stress and strain is in the local ($\underline{V}_1$, $\underline{V}_2$) directions defined above.

Transformation

The global degrees of freedom may be transformed at any node.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at point 5 in Figure 3-43. If the ALL POINTS parameter is included, output is given at all nine points shown in Figure 3-43. The latter is the usual option.

Notes: Sensitive to excessive distortions. Use a rectangular mesh.

Membrane analysis is extremely difficult due to rigid body modes. For example, a circular cylinder shape is particularly numerically sensitive.

This element has no bending stiffness.

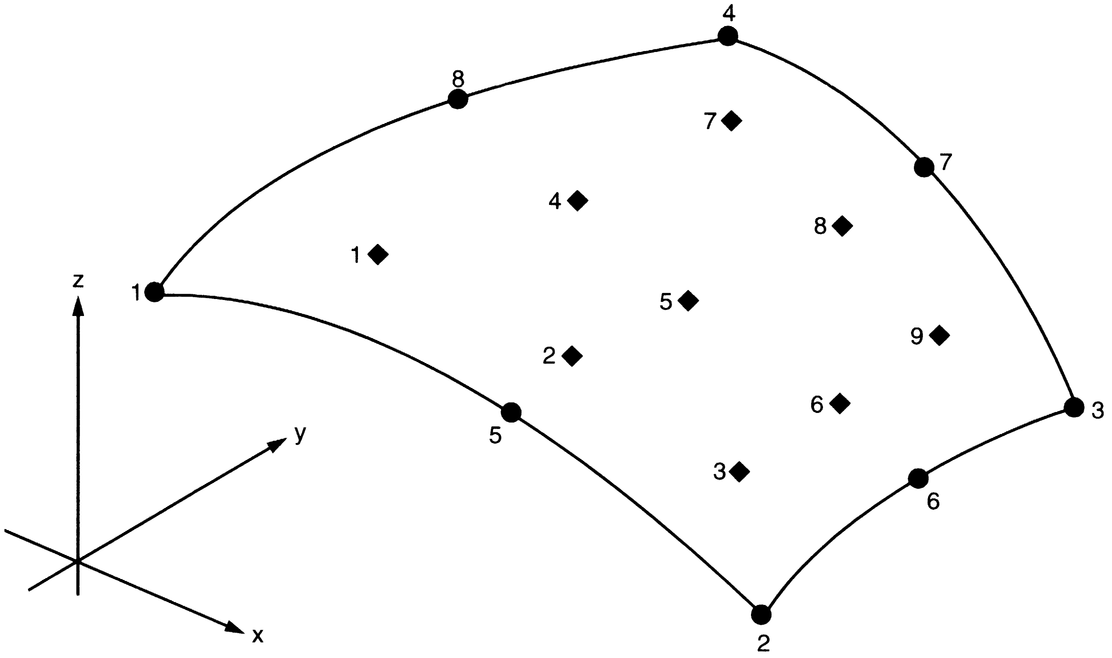


Figure 3-43 Integration Point Numbers for Eight-Node Membrane Element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered the design variable for this element.

Element 31

Elastic Curved Pipe (Elbow)/Straight Beam

This is an elastic, curved-pipe beam element with flexibility of the elbow based on an analytical elastic solution of the elbow segment. The flexibility and resulting stress distribution is not only a function of the cross-sectional properties, but also of the internal pressure in the elbow.

In addition, arbitrary cross sections can be specified for this element thus allowing for the possibility of analyzing arbitrary curved beams. The element can be degenerated into a straight elastic beam with elastic properties. The effects of axial stiffness, bending stiffness and shear stiffness are included in this straight beam element.

A curved beam is specified by the coordinates of the end points of the beam segment (COORDINATES option), the bending radius and center of rotation (GEOMETRY option) (Figure 3-44). If a zero is specified for the bending radius, the element is assumed to be straight. Cross-sectional properties are specified via the GEOMETRY option or the BEAM SECT parameter for arbitrary cross-sectional properties.

Element 31 has three displacement and three rotation components of degrees of freedom with respect to the global system. On the output, however, the local quantities such as forces and moments in the plane and normal to the plane for each element are also given.

Quick Reference

Type 31

Curved-pipe elbow or straight beam; elastic behavior.

Connectivity

Two nodes per element.

Geometry

The default cross section is a hollow, circular cylinder. Other cross sections are specified on the BEAM SECT parameter and are cross-referenced here.

EGEOM1
Thickness of the hollow cylinder if default cross section is used.
Enter a zero if a noncircular pipe is defined through the BEAM SECT parameter.

EGEOM2
Radius of the circular cross section if default cross section is used. Otherwise, the BEAM SECT identifier is specified on the parameter.

EGEOM3

Bending radius of the elbow if a curved beam has to be used.
Enter a zero if the element is straight.

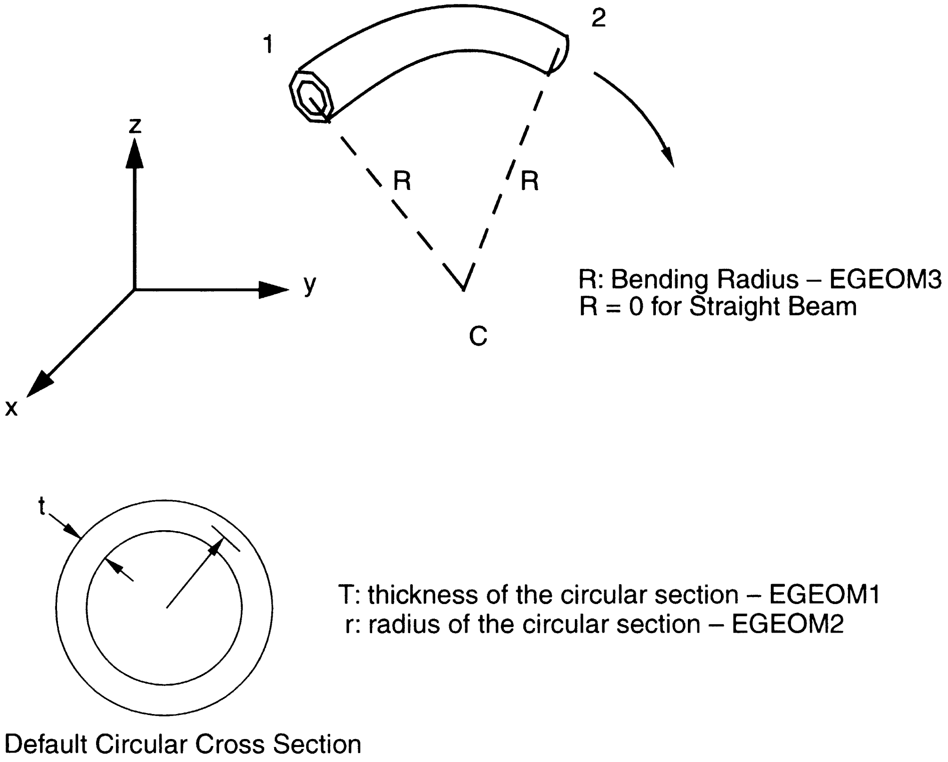


Figure 3-44 Elastic Curved (Elbow) / Straight Beam Element

The values EGEOM4, EGEOM5, and EGEOM6 need to be specified only if the element is straight. They represent the components of a vector in the plane of the locate x-axis and the beam axis (constructed by connecting node 1 and 2). The local x-axis will lie on the same side as the specified vector.

Coordinates

It is sufficient to specify only three coordinates per node. They represent the spatial global x, y, z position of a node.

Degrees of Freedom

$1 = u$ global displacement in x direction
$2 = v$ global displacement in y direction
$3 = w$ global displacement in z direction
$4 = \theta_x$ global rotation about the x-axis
$5 = \theta_y$ global rotation about the y-axis
$6 = \theta_z$ global rotation about the z-axis

Distributed Loads

Distributed loads can be entered as follows:

Load Type	Description
3	Internal pressure with closed-end caps.
-3	Internal pressure with open-end caps.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may be applied at the nodes.

Output

For each element, the following output will be obtained in a local reference system.

1	In-plane shear force
2	Axial force
3	Out-of-plane shear force
4	Out-of-plane moment
5	Torque around beam axis
6	In-plane moment

In addition, the following (maximum) stress quantities will be printed.

1	Shear stress due to in-plane shear force
2	Axial stress
3	Shear stress due to out-of-plane force
4	Maximum out-of-plane bending stress
5	Shear stress due to torsion
6	Maximum in-plane bending stress
7	Hoop stress due to internal pressure

Output Points

Output of stress quantities is at the nodal points. All quantities are based on analytical solutions. No numerical integration is required.

Transformation

Displacement and rotations at the node may be transformed to a local coordinate system.

Updated Lagrange Procedure and Finite Strain

This element does not have a finite strain capability.

Coupled Analysis

This element cannot be used for a coupled analysis; it has no heat transfer equivalent.

Dynamic Analysis

This element can be used in a dynamic analysis. The mass matrix is based on a subdivision of the total mass onto the displacement degrees of freedom of the two end nodes.

Element 32

Plane Strain Eight-Node Distorted Quadrilateral, Herrmann Formulation

Element type 32 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible plane strain applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 80, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 27 when other material behavior, such as plasticity, must be represented.

This element may also be used in coupled soil-pore pressure analyses.

Quick Reference

Type 32

Second-order, isoparametric, distorted quadrilateral. Plane strain. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first in a counterclockwise direction (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-45.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other data fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

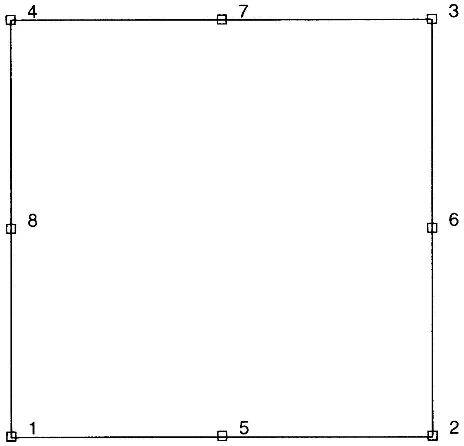


Figure 3-45 Eight-Node, Plane Strain Herrmann Element

Degrees of Freedom

At each corner node.

1 = u = global x-direction displacement
2 = v = global y-direction displacement at corner nodes only
3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney, Ogden, or Soil)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.

Load Type (IBODY)	Description
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

In coupled soil analyses, add 70 to the IBODY to apply distributed mass flux.

Output of Strains

Strains are printed in the order listed below:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy}, shear
5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative pressure (for Mooney, Ogden, or Soil)

Output of Stresses

Four stresses corresponding to the first four strains.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-46.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-35. This is the usual option for a second-order element.

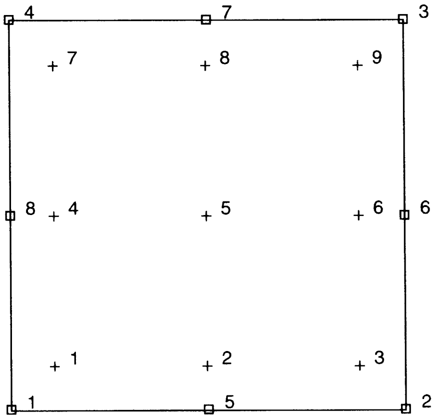


Figure 3-46 Integration Points for Eight-Node Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Element 33

Axisymmetric, Eight-Node Distorted Quadrilateral, Herrmann Formulation

Element type 33 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 81, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 28 when other material behavior, such as plasticity, must be represented.

This element may also be used in coupled soil-pore pressure analyses.

Quick Reference

Type 33

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation. Hybrid formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first in a counterclockwise direction (right-handed convention in z-r plane). Then the fifth node between the first and second; the sixth node between the second and third, etc. See Figure 3-47.

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

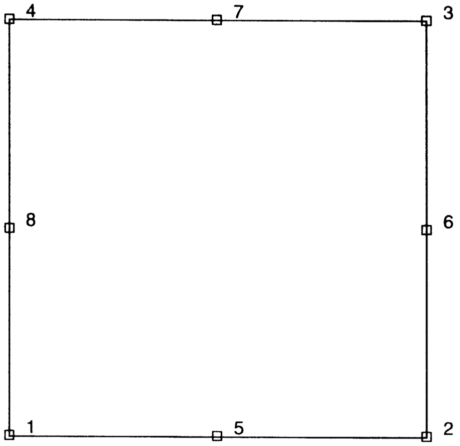


Figure 3-47 Eight-Node Axisymmetric Herrmann Element

Degrees of Freedom

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Additional degree of freedom at each corner node (Herrmann)

3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney, Ogden, or Soil).

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.

Load Type (IBODY)	Description
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒→3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

In coupled soil analyses, add 70 to the IBODY to apply distributed mass flux.

Output of Strains

The strains are printed in the order defined below:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direct
4 = γ_{zr}, shear in the section
5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative pressure (for Mooney, Ogden, or Soil)

Output of Stresses

Output for stresses is the same as the first four **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-48.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 3-48. This is the usual option for a second-order element.

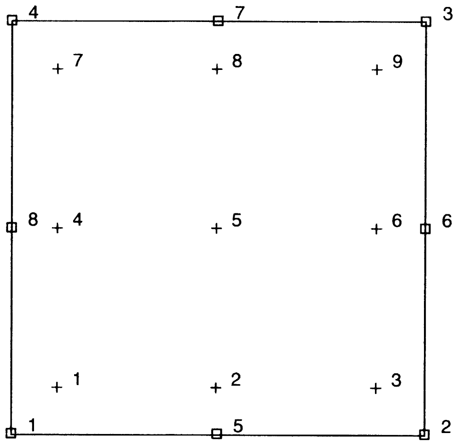


Figure 3-48 Integration Points for Eight-Node Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 38. See Element 38 for a description of the conventions used for entering the flux and film data for this element.

Element 34

Generalized Plane Strain Distorted Quadrilateral, Herrmann Formulation

This element is an extension of the plane strain isoparametric quadrilateral (element type 32) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.These elements cannot be used with the element-by-element iterative solver.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 81, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 29 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 34

Second order, isoparametric, distorted quadrilateral, generalized plane strain, Hybrid formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in x-y plane). Then the fifth node between the first and second; the sixth node between the second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each of the ten nodes. Note that the ninth and tenth nodes may be anywhere in the (x,y) plane.

Degrees of Freedom

At each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Additional degree of freedom at the first four (corner) nodes:

3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

One at the ninth node:

1 = Δ_z = relative z-direction displacement of front and back surfaces. See Figure 3-40.

Two at the tenth node:

1 = $\Delta\theta_x$ = relative rotation of front and back surfaces about global x-axis.
2 = $\Delta\theta_y$ = relative rotation of front and back surfaces about global y-axis.

See Figure 3-49.

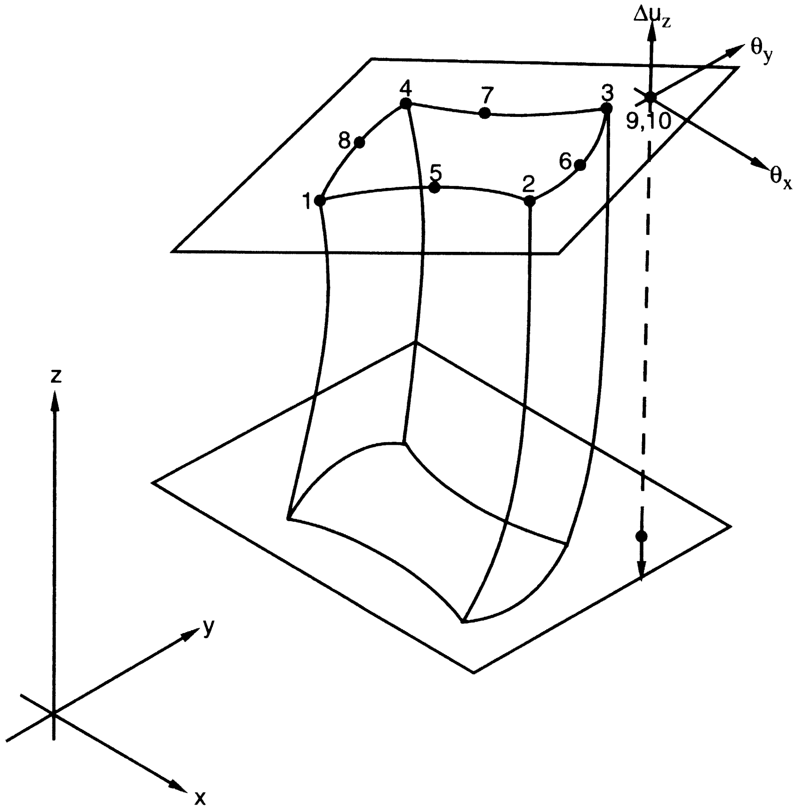


Figure 3-49 Generalized Plane Strain Distorted Quadrilateral

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.

Load Type (IBODY)	Description
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Strains are printed in the order defined below:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy}, shear in the (x-y) plane

No γ_{xz} or γ_{yz} shear

5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Output for stresses is the same as the first four strains.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-50.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 3-50. This is the usual parameter for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

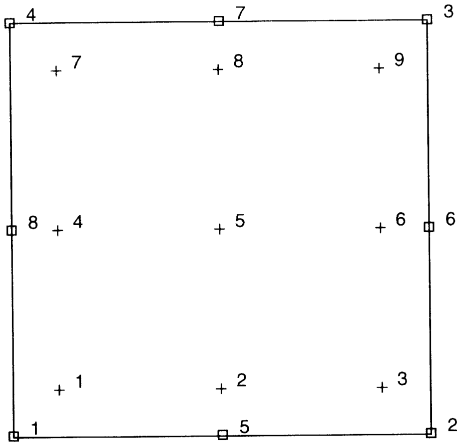


Figure 3-50 Integration Points for Eight-Node Element

Element 35

Three-Dimensional 20-Node Brick, Herrmann Formulation

Element type 35 is a 20-node, isoparametric, arbitrary hexahedral written for incompressible applications. This element uses triquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using trilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 84, are preferred in contact analyses.

The stiffness of this element is formed using 27-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 21 when other material behavior, such as plasticity, must be represented.

This element may also be used in coupled soil-porc pressure analyses.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of 20 nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in a counterclockwise direction when viewed from inside the element. Nodes 5, 6, 7, 8 are the corners of the opposite face; node 5 shares an edge with 1, node 6 with 2, etc. Nodes 9, 10, 11, 12 are middle of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc.

Note that in most normal cases, the elements will be generated automatically, so that the user need not concern himself with the node numbering scheme.

Reduction to Wedge or Tetrahedron

The element may be reduced as far as a tetrahedron, simply by repeating node numbers. Element type 130 would be preferred for tetrahedrals.

Integration

The element is integrated numerically using 27 points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 3-51). Two similar planes follow, moving toward the 5, 6, 7, 8 face. Thus, point 14 represents the "centroid" of the element, and is used for stress output, if only one point is flagged.

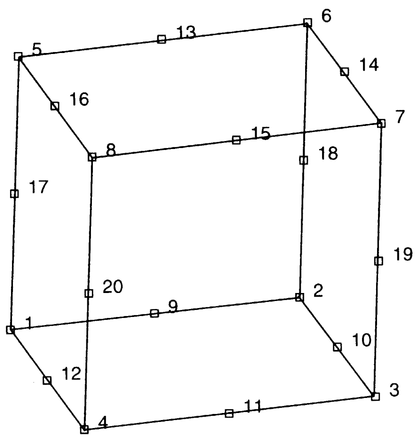


Figure 3-51 Twenty-Node – Brick Element

The subroutine FORCEM is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the 27 integration points.

For nonuniform surface pressures, values need only be supplied for the nine integration points on the face of application.

Quick Reference

Type 35

Twenty-nodes, isoparametric arbitrary distorted cube. Herrmann formulation. See Volume A for details on this theory.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and is shown in Figure 3-51.

Geometry

Not required.

Coordinates

Three global coordinates in the x, y, and z directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w, at all nodes. Additional degree of freedom at the corner nodes (first 8 nodes) is:

σ_{kk}/E = mean pressure variable (for Herrmann)
-p = negative hydrostatic pressure (for Mooney, Ogden)

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force in z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.

Load Type	Description
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force in z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude supplied through user subroutine FORCEM.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in FORCEM.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2156 face; magnitude and direction supplied in FORCEM.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3267 face; magnitude and direction supplied in FORCEM.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4378 face; magnitude and direction supplied in FORCEM.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in FORCEM.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.

Load Type	Description
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

In coupled soil analyses, add 70 to the IBODY to apply distributed mass flux.

Output of Strains

Strain output in global components:

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = ε_{xy}
5 = ε_{yz}
6 = ε_{zx}
7 = σ_{kk}/E = mean pressure variable (for Herrmann)
= $-p$ = negative hydrostatic pressure (for Mooney, Ogden, or Soil)

Output of Stresses

Output for stresses is the same as the first six strains.

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or twenty-seven Gaussian integration points (see Figure 3-52).

Note: Large bandwidth results in long run times; optimize.

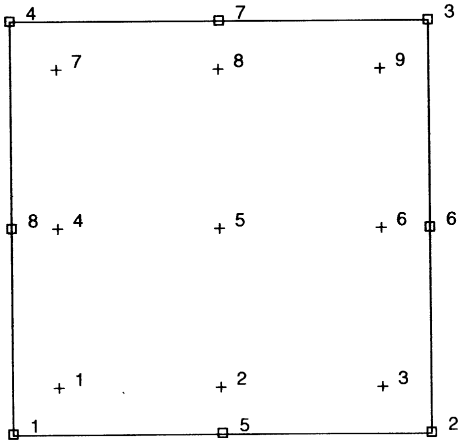


Figure 3-52 Integration Points on First Plane of Brick Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 44. See Element 44 for a description of the conventions used for entering the flux and film data for this element.

Element 36

Three-Dimensional Link (Heat Transfer Element)

This element is a simple, linear, straight link with constant cross-sectional area. It is the heat-transfer equivalent of element type 9.

This element can be used as a convection-radiation link for the simulation of convective and/or radiative boundary conditions (known ambient temperatures) or, for the situation of cross convection and/or cross radiation. In order to use this element as a convection-radiation link, additional input data must be entered using the GEOMETRY option.

Quick Reference

Type 36

Three-dimensional, two-node, heat transfer link.

Connectivity

Two nodes per element (see Figure 3-53).

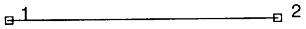


Figure 3-53 Three-Dimensional Heat Link

Geometry

The cross-sectional area is input in the first data field (EGEOM1); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If the element is used as a convection-radiation link, the following data must be entered:

EGEOM2 = emissivity

EGEOM3 = Stefan-Boltzmann constant

EGEOM4 = absolute temperature conversion factor (degrees Rankine = 459.7 + degrees Fahrenheit or, degrees Kelvin = 273.15 + degrees centigrade)

EGEOM5 = constant film coefficient

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

One degree of freedom per node:

1 = temperature

Fluxes

Distributed fluxes according to value of IBODY are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross-sectional area).
1	Volumetric flux on entire element (per volume).
2	Nonuniform flux given in user subroutine FLUX on first node (per cross-sectional area).
3	Nonuniform volumetric flux given in user subroutine FLUX on entire element (per volume).

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Current

Same specification as **Fluxes**.

Output Points

A single value at the centroid is given.

Element 37

Arbitrary Planar Triangle (Heat Transfer Element)

Element type 37 is a three-node, isoparametric, triangular element written for planar heat transfer applications. This element may also be used for electrostatic and magnetostatic analysis. As this element uses bilinear interpolation functions, the thermal gradients are constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as type 131. Hence, use a fine mesh.

The conductivity of this element is formed using one-point integration at the centroid.

Quick Reference

Type 37

Two-dimensional, arbitrary, three-node, heat transfer triangle.

Connectivity

Node numbering must follow right-handed convention (see Figure 3-54).

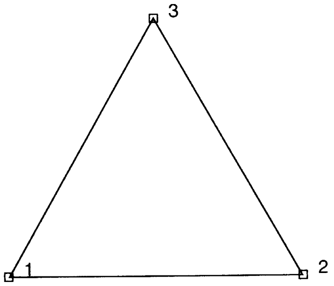


Figure 3-54 Planar Heat Transfer Triangle

Geometry

The thickness of the element may be specified in the first data field (EGEOM1); the other fields are not used. If not specified, unit thickness is assumed.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Fluxes are distributed according to value of IBODY as follows:

Flux Type	Description
0	Flux on 1-2 face of element (per unit area).
1	Volumetric flux on entire element (per unit volume).
3	Nonuniform flux given in user subroutine FLUX on 1-2 face (per unit area).
4	Nonuniform volumetric flux given in user subroutine FLUX on entire element (per unit volume).
6	Uniform flux on 2-3.
7	Nonuniform flux on 2-3.
8	Uniform flux on 3-1.
9	Nonuniform flux on 3-1.

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Magnetostatic

Capability is available.

Electrostatic

Capability is available.

Charge

Same specification as **Fluxes**.

Current

Same specification as **Fluxes**.

Output Points

A single value at the centroid is given.

Element 38

Arbitrary Axisymmetric Triangle (Heat Transfer Element)

Element type 38 is a three-node, isoparametric, triangular element written for axisymmetric heat transfer applications. This element may also be used for electrostatic and magnetostatic analysis.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as type 132. Hence, use a fine mesh.

The conductivity of this element is formed using one-point integration at the centroid.

Quick Reference

Type 38

Arbitrary, three-node, axisymmetric, heat transfer triangle.

Connectivity

Node numbering must follow right-handed convention (see Figure 3-55).

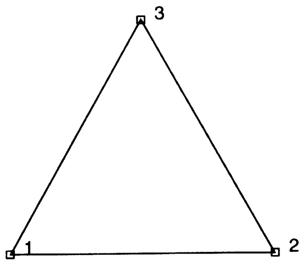


Figure 3-55 Axisymmetric Heat Transfer Triangle

Geometry

No geometry input required.

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Fluxes are distributed according to value of IBODY as follows:

Flux Type	Description
0	Flux on 1-2 face of element (per unit area).
1	Volumetric flux on entire element (per unit volume).
3	Nonuniform flux given in user subroutine FLUX on 1-2 face (per unit area).
4	Nonuniform volumetric flux given in user subroutine FLUX on entire element (per unit volume).

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Charge

Same specification as **Fluxes**.

Current

Same specification as **Fluxes**.

Output Points

A single value at the centroid is given.

■ Element 39

Planar Bilinear Quadrilateral (Heat Transfer Element)

Element type 39 is a four-node, isoparametric, arbitrary quadrilateral written for planar heat transfer applications. This element may also be used for electrostatic or magnetostatic applications.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as types 41 or 69. Hence, use a fine mesh.

The conductivity of this element is formed using four-point Gaussian integration.

Quick Reference

Type 39

Arbitrary, planar, heat transfer quadrilateral.

Connectivity

Node numbering must follow the right-hand convention (see Figure 3-56).

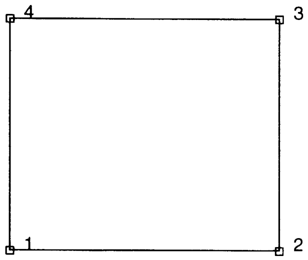


Figure 3-56 Planar Heat Transfer Quadrilateral

Geometry

Thickness is input in the first data field (EGEOM1). The other two data fields are not used. If no thickness is input, unit thickness will be assumed.

Coordinates

Two global coordinates, x and y.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in subroutine FLUX.
4	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
5	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element; magnitude given in subroutine FLUX.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes.**

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes.**

Charge

Same specifications as **Fluxes.**

Output Points

Centroid or four Gaussian integration points.

Tying

Use subroutine UFORMS.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

Element 40

Axisymmetric Bilinear Quadrilateral Element (Heat Transfer Element)

Element type 40 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric heat transfer applications. This element may also be used for electrostatic or magnetostatic applications.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

In general, one needs more of these lower-order elements than the higher-order elements such as types 42 or 70. Hence, use a fine mesh.

The conductivity of this element is formed using four-point Gaussian integration.

Quick Reference

Type 40

Arbitrarily distorted axisymmetric heat transfer quadrilateral.

Connectivity

Node numbering must follow right-hand convention (see Figure 3-57).

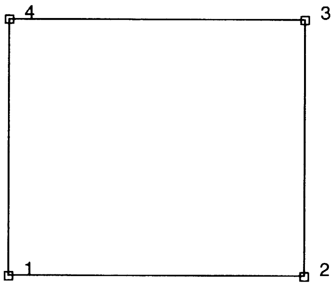


Figure 3-57 Axisymmetric Heat Transfer Quadrilateral

Geometry

Not applicable.

Coordinates

Two global coordinates, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in subroutine FLUX.
4	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
5	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.

Flux Type	Description
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element; magnitude given in subroutine FLUX.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid or four Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

View Factors Calculation for Radiation

Capability is available.

Element 41

Eight-Node Planar Biquadratic Quadrilateral (Heat Transfer Element)

Element type 41 is a eight-node, isoparametric, arbitrary quadrilateral written for planar heat transfer applications. This element may also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements, hence the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

The conductivity of this element is formed using nine-point Gaussian integration.

Quick Reference

Type 41

Second-order, distorted heat quadrilateral.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes starting with node 5 in between 1 and 2, and so on (see Figure 3-58).

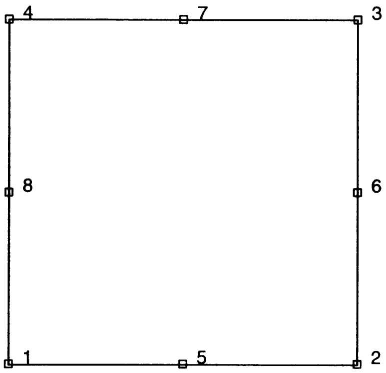


Figure 3-58 Eight-Node Planar Heat Transfer Quadrilateral

Geometry

Thickness stored in EGEOM1 field. If not specified, unit thickness is assumed.

Coordinates

Two global coordinates per node – x and y.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Surface Fluxes:
Surface fluxes are specified as below. All are per unit surface area. All nonuniform fluxes are specified through user subroutine FLUX.

Traction Type	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.

Traction Type	Description
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through user subroutine FLUX.

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid or nine Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
2	(4 - 8 - 1) to (2 - 6 - 3)

Element 42

Eight-Node Axisymmetric Biquadratic Quadrilateral (Heat Transfer Element)

Element type 42 is a eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric heat transfer applications. This element may also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

The conductivity of this element is formed using nine-point Gaussian integration.

Quick Reference

Type 42

Second-order, distorted axisymmetric heat quadrilateral.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes with node 5 located between nodes 1 and 2, etc. (see Figure 3-59).

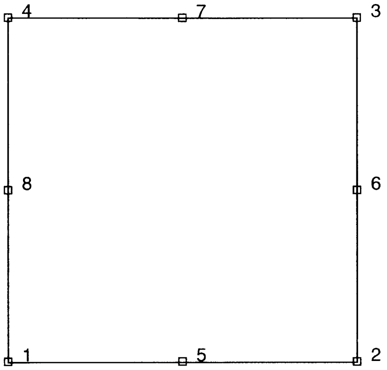


Figure 3-59 Eight-Node Axisymmetric Heat Transfer Quadrilateral

Geometry

Not applicable.

Coordinates

Two global coordinates per node, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Surface Fluxes

Surfaces fluxes are specified as below. All are per unit surface area. All nonuniform fluxes are specified through user subroutine FLUX.

Flux Type (IBODY)	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.

Flux Type (IBODY)	Description
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through user subroutine FLUX.

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid or nine Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
	(4 - 8 - 1) to (2 - 6 - 3)

View Factors Calculation for Radiation

Capability is available.

Element 43

Three-Dimensional Eight-Node Brick (Heat Transfer Element)

Element type 43 is a eight-node, isoparametric, arbitrary quadrilateral written for three-dimensional heat transfer applications. This element may also be used for electrostatic applications.

As this element uses trilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

In general, one needs more of these lower order elements than the higher order elements such as 44 or 71. Hence, use a fine mesh.

The conductivity of this element is formed using eight-point Gaussian integration.

Quick Reference

Type 43

Eight-node, three-dimensional, first-order isoparametric heat transfer element.

Connectivity

Eight nodes per element.

Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on (see Figure 3-60).

Geometry

Not applicable for this element.

Coordinates

Three coordinates in the global x, y, and z directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

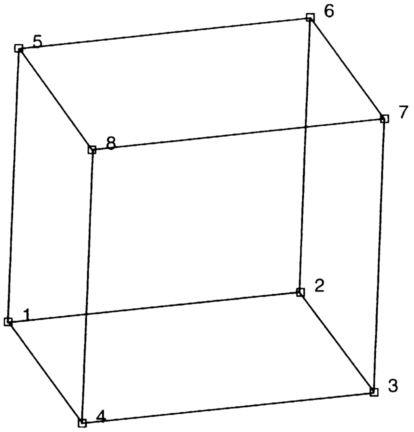


Figure 3-60 Arbitrarily Distorted Heat Transfer Cube

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are assumed positive when directed into the element.

Load Type (IBODY)	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform surface flux (supplied via user subroutine FLUX) on 1-2-3-4 face.
2	Uniform volumetric flux.
3	Nonuniform volumetric flux (with FLUX).
4	Uniform flux on 5-6-7-8 face.
5	Nonuniform surface flux on 5-6-7-8 face (FLUX).
6	Uniform flux on 1-2-6-5 face.

Load Type (IBODY)	Description
7	Nonuniform flux on 1-2-6-5 face (FLUX).
8	Uniform flux on 2-3-7-6 face.
9	Nonuniform flux on 2-3-7-6 face (FLUX).
10	Uniform flux on 3-4-8-7 face.
11	Nonuniform flux on 3-4-8-7 face (FLUX).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (FLUX).

For IBODY= 3, P is the magnitude of volumetric flux at volumetric integration point NN of element N. For IBODY odd but not equal to 3, P is the magnitude of surface flux for surface integration point NN of element N.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid or eight Gaussian integration points (see Figure 3-61).

Note: As in all three-dimensional analysis, a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

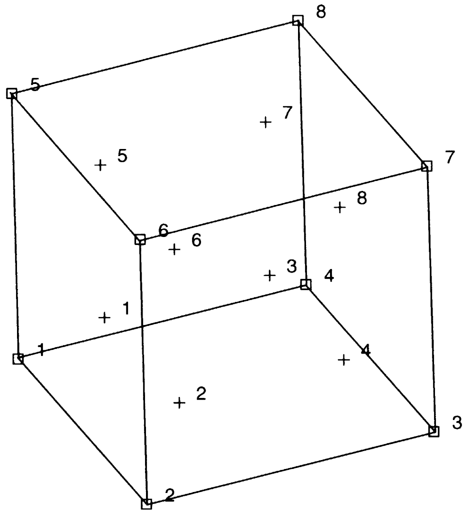


Figure 3-61 Integration Points for Element 43

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 44

Three-Dimensional 20-Node Brick (Heat Transfer Element)

Element type 44 is a 20-node isoparametric arbitrary quadrilateral written for three-dimensional heat transfer applications. This element may also be used for electrostatic applications.

This element uses triquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

The conductivity of this element is formed using 27-point Gaussian integration.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, and 4 are the corners of one face, given in a counterclockwise direction when viewed from inside the element. Nodes 5, 6, 7, 8 are the corners of the opposite face; node 5 shares an edge with 1; node 6 shares an edge with 2, etc. Nodes 9, 10, 11, 12 are the middles of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc. (see Figure 3-62).

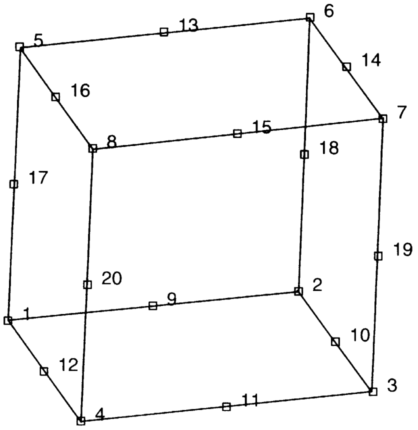


Figure 3-62 Twenty-Node Heat Transfer Brick

Note that in most normal cases, the elements will be generated automatically so you need not concern yourself with the node numbering scheme.

Integration

The element is integrated numerically using 27 points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 3-63). Two similar planes follow, moving toward the 5, 6, 7, 8 face. Thus, point 14 represents the "centroid" of the element, and is used for temperature output if the CENTROID parameter is used.

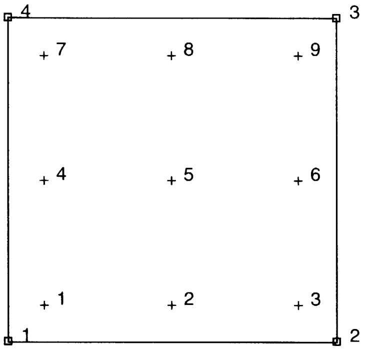


Figure 3-63 Points of Integration in a Sample Integration Plane

Fluxes

Distributed fluxes chosen by value of IBODY.

Reduction to Wedge or Tetrahedron

The element may be reduced as far as a tetrahedron, simply by repeating node numbers at the same spatial position. Element type 133 is preferred for tetrahedrals.

Notes: A large bandwidth results in long run times. Optimize as much as possible.

The lumped specific heat option gives poor results with this element at early times in transient solutions. If accurate transient analysis is required, the user should not use the lumping option with this element.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Volume A.

Quick Reference

Type 44

Twenty-node isoparametric brick (heat transfer element).

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element, and as shown in Figure 3-62.

Geometry

Not applicable.

Coordinates

Three global coordinates in the x, y, and z directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Load Type	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform flux on 1-2-3-4 face (FLUX).

Load Type	Description
2	Uniform body flux.
3	Nonuniform body flux (FLUX).
4	Uniform flux on 6-5-8-7 face.
5	Nonuniform flux on 6-5-8-7 face (FLUX).
6	Uniform flux on 2-1-5-6 face.
7	Nonuniform flux on 2-1-5-6 face (FLUX).
8	Uniform flux on 3-2-6-7 face.
9	Nonuniform flux on 3-2-6-7 face (FLUX).
10	Uniform flux on 4-3-7-8 face.
11	Nonuniform flux on 4-3-7-8 face (FLUX).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (FLUX).

For IBODY=3, the value of P in the user subroutine FLUX is the magnitude of volumetric flux at volumetric integration point NN of element N. For IBODY odd but not equal to 3, P is the magnitude of surface flux at surface integration point NN of element N. Surface flux is positive when heat energy is added to the element.

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid or 27 Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 45

Curved Timoshenko Beam In A Plane

This is a three-node planar beam element which allows transverse shear as well as axial straining. It is based on a quadratic displacement assumption on the global displacements and rotation. The strain-displacement relationships are complete except for large curvature change terms (consistent with the other beam and shell elements in the program). The shear strain is assumed to be quadratic across the thickness. The default cross section is a rectangle. Integration for section properties uses a Simpson rule across the section. Integration for element stiffness and mass uses a two- and three-point Gauss rule along the beam axis. All constitutive relations may be used with this element type.

Quick Reference

Type 45

Curved, planar Timoshenko beam.

Connectivity

Three nodes per element (see Figure 3-64).

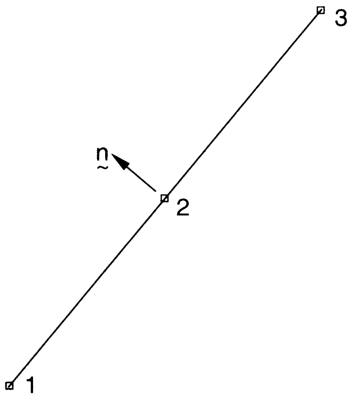


Figure 3-64 Three-Node Timoshenko Beam in a Plane

Geometry

Linear thickness variation along the element. Thickness at first node of the element stored in first data field (EGEOM1).

Thickness at third node of the element stored in third data field (EGEOM3). If EGEOM3=0, constant thickness is assumed.

Beam width (normal to the plane of deformation) stored in second data field, EGEOM2. The default width is unity.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Two coordinates at all nodes:

1 = x
2 = y

Right-handed coordinate set.

Degrees of Freedom

1 = u (global x component of displacement)
2 = v (global y component of displacement)
3 = ϕ_s = rotation of the cross section (right-handed rotation)

Tractions

Distributed loads. Selected with load type are as follows:

Load Type	Description
0	Uniform normal force per beam length as shown in Figure 3-64. Positive pressure is in the negative normal (n) direction.
1	Uniform load (force per beam length) in global x-direction.
2	Uniform load (force per beam length) in global y-direction.
3	Nonuniform load (force per beam length) in global x-direction. (FORCEM).
4	Nonuniform load (force per beam length) in global y-direction. (FORCEM).
5	Nonuniform normal force per beam length as shown in Figure 3-64 (FORCEM). Positive pressure is in the negative normal (n) direction.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For nonuniform loads (types 3, 4, 5) the magnitude is supplied at each of the three Gauss points via user subroutine FORCEM.

Concentrated loads and moments may be applied at the nodes.

Output of Strain and Stress

Two values of strains and stresses are stored at each of the integration points through the thickness:

1 = axial
2 = transverse shear

The first point of the section is on the surface up the positive normal (opposite to the positive pressure), the last point is on the opposite surface.

Transformation

Allowable in x-y plane.

Output Points

Centroidal section or the two Gauss points. The first Gauss point is closest to the first node.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points. This number must be odd.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain as for total Lagrangian approach. Thickness will be updated, but beam width is assumed to be constant.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 65. See Element 65 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness (beam height) and/or the beam width can be considered as design variables.

■ Element 46

Eight-Node Plane Strain Rebar Element

This element is isoparametric, plane strain, 8-node hollow quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node plane strain continuum element (e.g., element 27 or 32) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-65), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points. At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

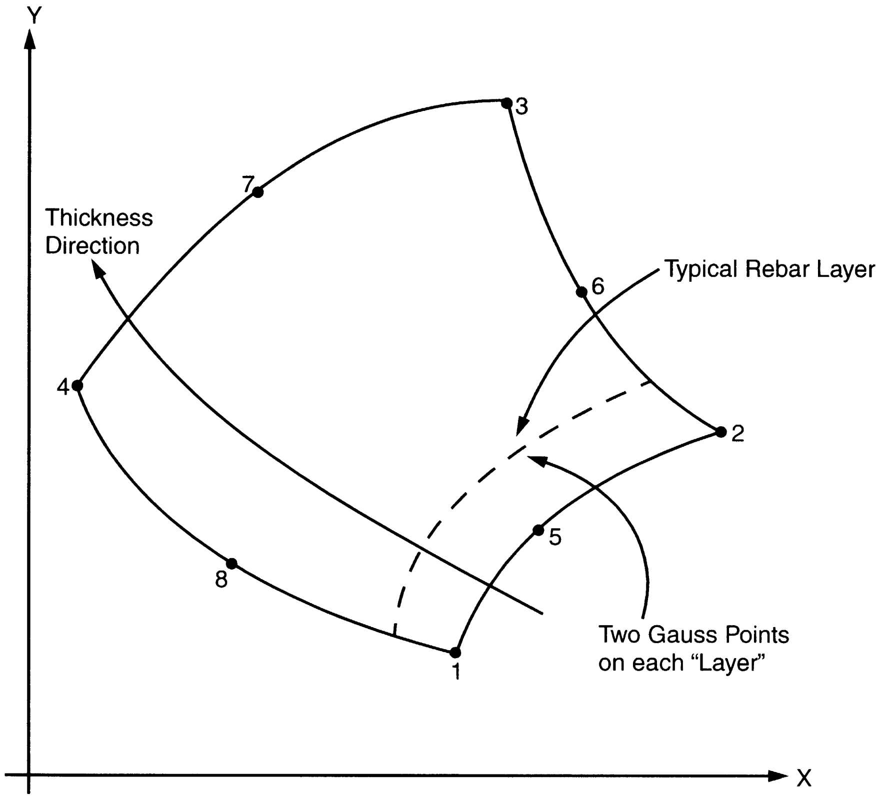


Figure 3-65 Eight-Node Rebar Element Conventions

Quick Reference

Type 46

Eight-node, isoparametric rebar element to be used with 8-node plane strain continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 27 or 32.

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the "thickness" concept associated with rebar layers. If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, set to either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-65).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node plane strain elements (e.g., element types 27 or 32).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Any local set (u,v) may be used at any node.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 47

Generalized Plane Strain Rebar Element

This element is similar to element 46, but is written for generalized plane strain conditions. It is a hollow ten-node planar quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the corresponding generalized plane strain continuum element (e.g., element types 29 or 34) to represent a reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-66), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points. At each such integration point on each layer, you must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

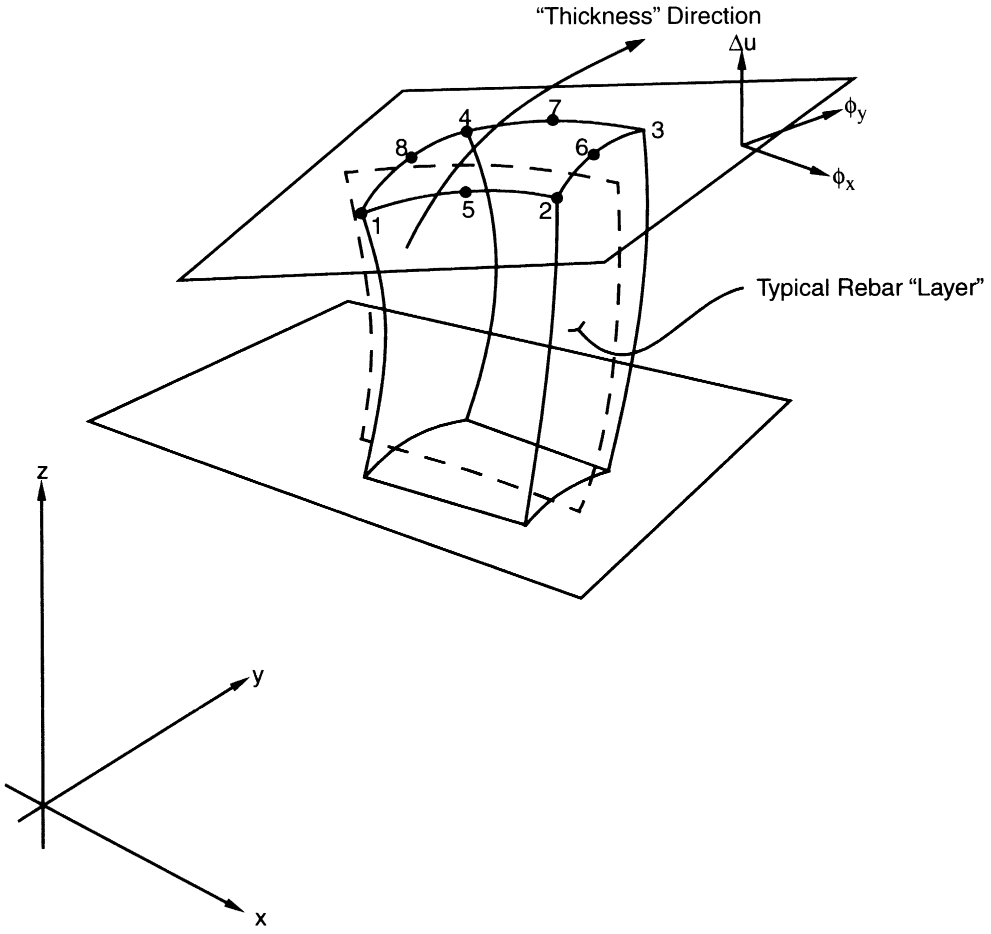


Figure 3-66 Ten-Node Generalized Plane Strain Rebar Element Conventions

Quick Reference

Type 47

Ten-node, generalized plane strain rebar element.

Connectivity

Node numbering of the element is same as that for element 29 or 34.

Geometry

Element thickness (in the z-direction) in first field. Default thickness is unity. Note, this should not be confused with "thickness" concept associated with rebar layers.

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, set to either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-66).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates in x- and y-directions.

Degrees of Freedom

At the first 8 nodes, global x and y displacements.

At node 9, the relative translation of the top surface of the element with respect to the bottom surface.

At node 10, the relative rotations of the top surface of the element with respect to the bottom surface.

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding generalized plane strain elements (e.g., element types 29 or 34).

Output Of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Any local set (u,v) may be used at the any node at first 8 nodes.

Special Considerations

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 48

Eight-Node Axisymmetric Rebar Element

This element is similar to element 46, but is written for axisymmetric conditions. It is a hollow, isoparametric 8-node quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element (e.g., element 28 or 33) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by the user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-67), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

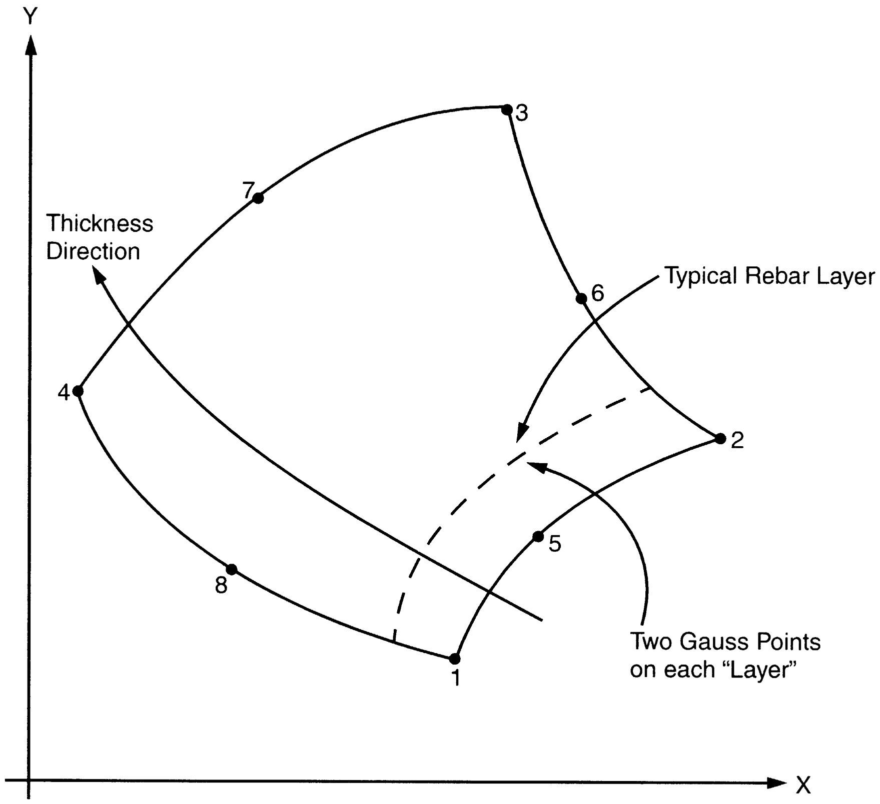


Figure 3-67 Eight-Node Rebar Element Conventions

Quick Reference

Type 48

Eight-node, isoparametric rebar element to be used with 8-node axisymmetric continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 28 or 33.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, set to either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-67).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates in z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r

Tractions

Point loads may be applied at the nodes but no distributed loads are available. Distributed loads are applied only to corresponding 8-node axisymmetric elements (e.g., element types 28 or 33).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformations

Any local set (u,v) may be used in the (z-r) plane at any node.

Special Considerations

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 49

Finite Rotation Linear Thin Shell Element

Element type 49 is a finite rotation, six-node, thin shell element. The in-plane displacements and the in-plane coordinates are linearly interpolated while the out-of-plane displacement and coordinate are quadratically interpolated. This quadratic interpolation provides the possibility to model slightly curved elements for which the influence of the (changes of) curvature on the membrane deformations is taken into account. This influence is especially important in cases where pressure loads have to be carried mainly by membrane forces and in cases of (nearly) inextensional bending. By default, this influence is taken into account. The element can also be used as a flat plate element by entering a nonzero value on the fifth (EGEOM5) geometry data field. In that case, the influence of the curvature on the membrane deformations is not taken into account, and the coordinates of the midside nodes are calculated as the average of the corresponding corner nodes. The degrees of freedom consist of three global translational degrees of freedom for the corner nodes and one local rotational degree of freedom at the midside nodes. These rotational degrees of freedom represent the average rotation of the surface normal about the element edges (see Figure 3-68).

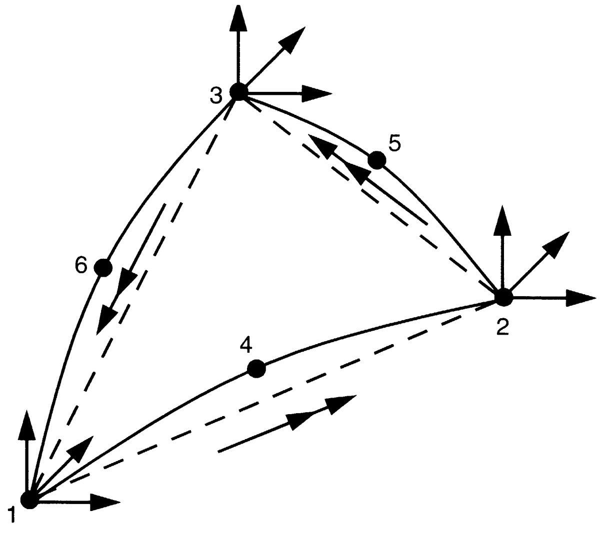


Figure 3-68 Element Type 49 Degrees of Freedom

This element does not suffer from the restriction that the incremental rotations must remain small provided that the LARGE DISP parameter is used.

Element type 49 has only one integration point. Together with the relatively small number of degrees of freedom, this element is very effective from a computational point of view.

All constitutive relations may be used with this element.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system. With respect to the set of base vectors (V_1, V_2, V_3), the generalized as well as the layer stresses and strains are given for the Gaussian integration point which coordinates readily follow from the average of the corner node coordinates (see Figure 3-69). The triangle determined by the corner nodes is called the basic triangle.

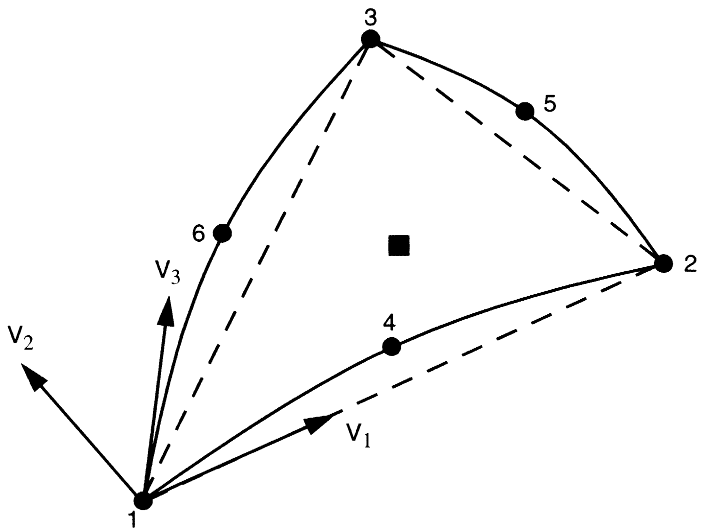


Figure 3-69 Element Type 49 Local Base Vectors and Integration Point Position

Degrees of Freedom

The nodal degrees of freedom are as follows:

At the three corner nodes:	u, v, w Cartesian displacement components.
At the three midside nodes:	ϕ, rotation of the surface normal about the element edge. The positive rotation vector points from the corner with the lower (external) node number to the node with the higher (external) node number.

Using these degrees of freedom, modeling of intersecting plates can be done without special tying types.

Quick Reference

Type 49

Linear, six-node shell element.

Connectivity

Six nodes. Corners given first, proceeding continuously around the element.

Then the midside nodes are given as follows:

4 = Between corners 1 and 2
5 = Between corners 2 and 3
6 = Between corners 3 and 1

Geometry

Linear thickness variation can be specified in the plane of the element. Internally, the average thickness is used. Thicknesses at first, second, and third node are stored for each element in the first (EGEOM1), second (EGEOM2), and third (EGEOM3) geometry data field, respectively. If EGEOM2-EGEOM3 are zero, then a constant thickness (EGEOM1) is assumed for the element.

Alternatively, the NODAL THICKNESS model definition option can be used for the input of the element thickness.

If a nonzero value is entered on the fifth (EGEOM5) geometry data field, the element is considered to be flat.

Coordinates

(x, y, z) global Cartesian coordinates are given. If the coordinates of the midside nodes are not given, they will be calculated as the average of the coordinates of the corresponding corner nodes.

Degrees of Freedom

At the three corner nodes:

1 = u = global Cartesian displacement in x-direction
2 = v = global Cartesian displacement in y-direction
3 = w = global Cartesian displacement in z-direction

At the three midside nodes:

1 = ϕ = rotation of surface normal about the edge. Positive rotation vector points from the corner with the lower (external) node number to the corner with the higher (external) node number.

Distributed Loading

Types of distributed loading are as follows:

Load Type	Description
1	Uniform gravity load per surface area in -z direction.
2	Uniform pressure; positive magnitude in $-V_3$ direction.
3	Uniform gravity load per surface area in -z direction; magnitude given in user subroutine FORCEM.
4	Nonuniform pressure; magnitude given in user subroutine FORCEM, positive magnitude given in $-V_3$ direction.
5	Nonuniform pressure; magnitude and direction given in user subroutine FORCEM.
11	Uniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge.
12	Uniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge.
13	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge; magnitude given in user subroutine FORCEM.
14	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge; magnitude given in user subroutine FORCEM.
15	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge; magnitude and direction given in user subroutine FORCEM.
21	Uniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge.

Load Type	Description
22	Uniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge.
23	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge; magnitude given in user subroutine FORCEM.
24	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge; magnitude given in user subroutine FORCEM.
25	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge; magnitude and direction given in user subroutine FORCEM.
31	Uniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge.
32	Uniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge.
33	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge; magnitude given in user subroutine FORCEM.
34	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge; magnitude given in user subroutine FORCEM.
35	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge; magnitude and direction given in user subroutine FORCEM.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity load in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Output Of Strains

Generalized strain components are as follows:

middle surface stretches ε_{11}, ε_{22}, ε_{12}
middle surface curvatures κ_{11}, κ_{22}, κ_{12}

in local (V_1, V_2, V_3) system.

Output Of Stress

Generalized stress components are as follows:

Tangential stress resultants σ_{11}, σ_{22}, σ_{12}
Tangential stress couples μ_{11}, μ_{22}, μ_{12}

all in local (V_1, V_2, V_3) system.

Stress components:

σ_{11}, σ_{22}, σ_{12} in local (V_1, V_2, V_3) system at equally spaced layers through the thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement components at corner nodes may be transformed to local directions.

Tying

Use subroutine UFORMS.

Output Points

Output occurs at the centroid of the element.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points. This number must be odd.

Beam Stiffeners

For small rotational increments, element type 49 is compatible with beam element types 76 and 77.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output for stresses and strains as for total Lagrangian approach. Notice that if only UPDATE is used, the subsequent increments are based upon linear strain-displacement relations. If both UPDATE and LARGE DISP are used, the full nonlinear strain-displacement relations are used for each increment.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 50. See Element 50 for a description of the convention used for entering the flux and film data for this element.

Element 50

Three-Node Linear Heat Transfer Shell Element

This is a three-node heat transfer shell element with temperatures as nodal degrees of freedom. A linear interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see parameter HEAT).

In the plane of the shell, a one-point Gaussian integration is used for the evaluation of the conductivity matrix and a three-point Gaussian integration for the evaluation of the heat capacity matrix. In the thickness direction of the shell, Simpson's rule is used where the number of point can be given by the SHELL SECT parameter (the default number is 11).

This element is compatible with element type 49 in a thermal-stress analysis and can be used in conjunction with three-dimensional heat transfer brick elements through a tying for heat transfer analyses.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the three corner nodes. The thickness is specified using either the GEOMETRY or NODAL THICKNESS option. The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) stands perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2 and V_3 form a right-hand system (see Figure 3-70).

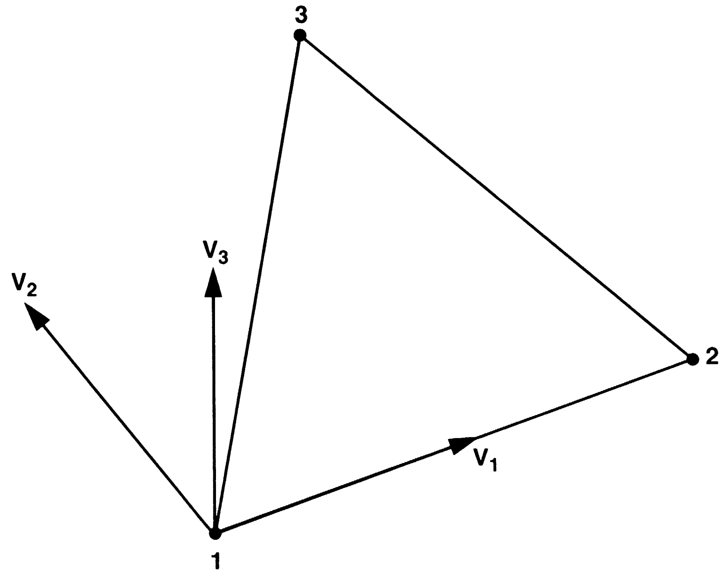


Figure 3-70 Element Type 50, Local Base Vectors

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Volume A.

Quick Reference

Type 50

Three-node linear heat transfer shell element.

Connectivity

Three nodes per element.

Geometry

Linear thickness variation is allowed in the plane of the element. Internally, the average thickness is used. Thicknesses at the first, second, and third node are stored for each element in the first (EGEOM1), second (EGEOM2), and third (EGEOM3) geometry data field, respectively.

Alternatively, the NODAL THICKNESS model definition option can be used for the input of the element thickness.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

N degrees of freedom per node (temperatures):

N = 2: linear distribution through the thickness.
1 = Top Surface Temperature
2 = Bottom Surface Temperature

N = 3: quadratic distribution through the thickness
1 = Top Surface Temperature
2 = Bottom Surface Temperature
3 = Midsurface Temperature

Fluxes

Two types of fluxes:

Volumetric Fluxes

Load Type	Description
1	Uniform flux per unit volume on whole element.
3	Uniform flux per unit volume on whole element; magnitude of flux id defined in subroutine FLUX.

Surface Fluxes

Load Type	Description
5	Uniform flux per unit surface area on top surface.
6	Uniform flux per unit surface area on top surface; magnitude of flux is defined in subroutine FLUX.
2	Uniform flux per unit surface area on bottom surface.
4	Uniform flux per unit surface area on bottom surface; magnitude of flux is defined in subroutine FLUX.

Point fluxes may also be applied at nodal degrees of freedom.

Films

Same specification as **Fluxes**.

Tying

Standard tying types 85 and 86 with three-dimensional heat transfer brick elements.

SHELL SECT – Integration through the thickness

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. The default is 11 points.

Output Points

Temperatures are printed out at the centroid of the element through the thickness of the shell. The first point in thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specification as **Fluxes**.

Element 51

Cable Element

This element is the sag cable element (Figure 3-71). The assumptions for this element are small strain, large displacement and constant strain through the element. This element allows linear elastic behavior only. This element cannot be used with CONTACT.

Notes: All distributed loads are formed on the basis of the current geometry.

Whenever this element is included in the structure, the distributed load magnitude given in user subroutine FORCEM must be the total magnitude to be reached at the end of the current increment and not the incremental magnitude.

Wind load magnitude is based on the unit projected distance and not projected cable length. If there is a big difference between cable length and the distance between the two nodes, it is recommended to subdivide the element along the cable.

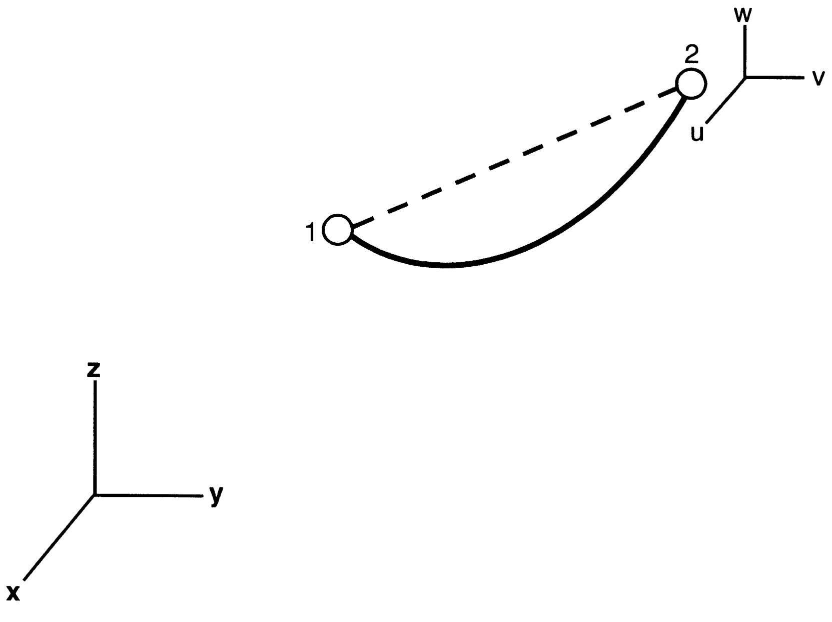


Figure 3-71 Three-Dimensional Cable

Quick Reference

Type 51

Three-dimensional, two-node, sag cable.

Connectivity

Two node per element.

Geometry

The cross-section area is entered in the first data field (EGEOM1). The cable length is entered in the second data field (EGEOM2). If the cable length is unknown and the initial stress is known, then enter a zero in the second data field and enter the initial stress in the third data field (EGEOM3). If the cable length is equal to the distance between the two nodes, only the first data (cross-sectional area) is required.

Coordinates

Three coordinates per node in the global X, Y, and Z direction.

Degrees of Freedom

Global displacement degree of freedom:

1 = u displacement
2 = v displacement
3 = w displacement

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform gravity load (force per unit cable length) in the arbitrary direction.
1	Wind load (force per unit projected distance to the normal plane with respect to wind vector).
2	Arbitrary load (force per unit length); use user subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Uniaxial in the cable member.

Output of Stresses

Uniaxial in the cable member.

Transformation

The three global degrees of freedom for any node may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Point

Constant value through the element.

Element 52

Elastic Beam

This is a straight, Euler-Bernoulli beam in space with only linear elastic material response. Large curvature changes are neglected in the large displacement formulation. Linear interpolation is used along the axis of the beam (constant axial force) with cubic displacement normal to the beam axis (constant beam curvature). This element may be used for nonlinear elasticity (HYPOELAS) where the material behavior is given in subroutine UBEAM (see Volume D). No other material nonlinearity is allowed with this element.

Geometric Basis

The element uses a local (x,y,z) set for section properties. Local x and y are the principal axes of the cross section. Local z is along the beam axis (Figure 3-72). The element is defined geometrically in the GEOMETRY fields 4, 5, and 6. Using the GEOMETRY option, a vector in the plane of the local x-axis and the beam axis must be specified. If no vector is defined here, the local coordinate system may alternatively be defined by the global (x,y,z) coordinates at the two nodes and by (x_1, x_2, x_3), a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam axis toward the point. The local x-axis is normal to the beam axis.

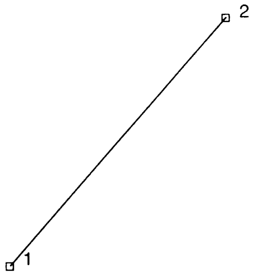


Figure 3-72 Elastic Beam Element

The local z-axis goes from node 1 to node 2, and the local y-axis forms a right-handed set with local x and z.

Quick Reference

Type 52

Elastic straight beam. Linear interpolation axially, cubic normal displacement interpolation.

Connectivity

Two nodes. Local z-axis from first to second node.

Geometry

First geometry data field – A – area
Second geometry data field – I_{xx} – moment of inertia of section about local x-axis
Third geometry data field – I_{yy} – moment of inertia of section about local y-axis

The bending stiffnesses of the section are calculated as EI_{xx} and EI_{yy}. The torsional stiffness of the section is calculated as $\frac{E}{2(1+\upsilon)}(I_{xx} + I_{yy})$. Here E and υ are Young's modulus and Poisson's ratio, calculated as functions of temperature.

If a zero is entered in the first geometry field, the program will use the beam section data corresponding to the section number given in the second geometry field. (Sections are defined using the BEAM SECT parameter.) This allows specification of the torsional stiffness factor K unequal to $I_{xx} + I_{yy}$.

EGEOM4-EGEOM6: Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis will lie on the same side as the specified vector.

Coordinates

First three coordinates - (x, y, z) global Cartesian coordinates.

Fourth, fifth, and sixth coordinates at each node – global Cartesian coordinates of a point in space which locates the local x-axis of the cross section: this axis lies in the plane defined by the beam nodes and this point, pointing from the beam towards this point. The local x-axis is normal to the beam axis. The fourth, fifth and sixth coordinates will only be used if the local x-axis direction is not specified in the GEOMETRY block.

Degrees of Freedom

1 = u_x = global Cartesian x-direction displacement
2 = u_y = global Cartesian y-direction displacement
3 = u_z = global Cartesian z-direction displacement
4 = ϕ_x = rotation about global x-direction
5 = ϕ_y = rotation about global y-direction
6 = ϕ_z = rotation about global z-direction

Tractions

The four types of distributed loading are as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via user subroutine FORCEM.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads and moments may be applied at the nodes.

Output of Strains

Generalized strain components are as follows:

Axial stretch ε
Curvature about local x-axis of cross section K_{xx}
Curvature about local y-axis of cross section K_{yy}
Twist about local z-axis of cross section K_{zz}

Output of Section Forces

Section forces are output as:

Axial force
Bending moment about x-axis of cross section
Bending moment about y-axis of cross section
Torque about beam axis

Transformation

Displacements and rotations may be transformed to local directions.

Tying

For interacting beams use tying type 100 for fully moment carrying joint, tying type 52 for pinned joint.

Output Points

Centroidal section or three Gauss integration sections.

For all beam elements, the default printout gives section forces and moments.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available for this element.

The finite strain capability does not apply to this element.

Note: Nonlinear elasticity can be implemented with the HYPOELAS option and user subroutine UBEAM.

Design Variables

The cross sectional area (A) and moments of inertia (I_{xx}, I_{yy}) can be considered as design variables.

Element 53

Plane Stress, Eight-Node Distorted Quadrilateral with Reduced Integration

Element type 53 is an eight-node, isoparametric, arbitrary quadrilateral written for plane stress applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 3, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution.

All constitutive models may be used with this element.

Quick Reference

Type 53

Second-order, isoparametric, distorted quadrilateral with reduced integration. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then fifth node between first and second; the sixth node between second and third, etc. See Figure 3-73.

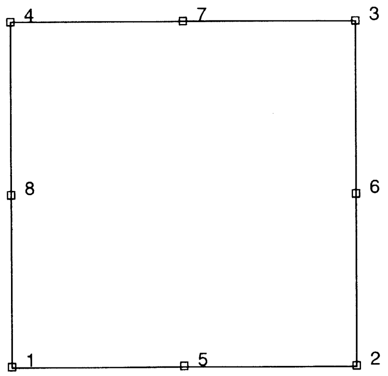


Figure 3-73 Nodes of Eight-Node, 2D Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement.
2 = v = global y-direction displacement.

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.

Load Type (IBODY)	Description
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

A nine-point integration scheme is used for the integration of body forces (see Figure 3-74).

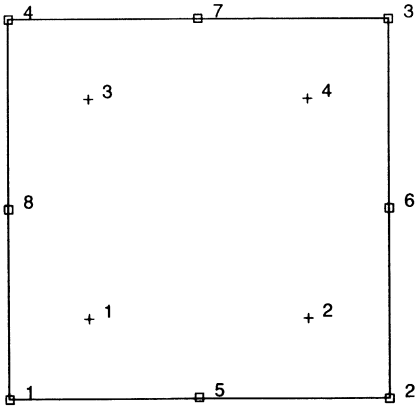


Figure 3-74 Integration Points of Eight-Node, 2D Element with Reduced Integration

Output of Strains

Output of strains at the centroid of element or four integration points (see Figure 3-74 and **Output Points** below) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = γ_{xy}, shear

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 3-74. This is the usual option for a second-order element with reduced integration.

Note: Because this is a reduced element, it is possible to excite so-called "hourglass" or "breathing" modes. This mode, shown in Figure 3-75, makes no contribution to the strain energy of the element.

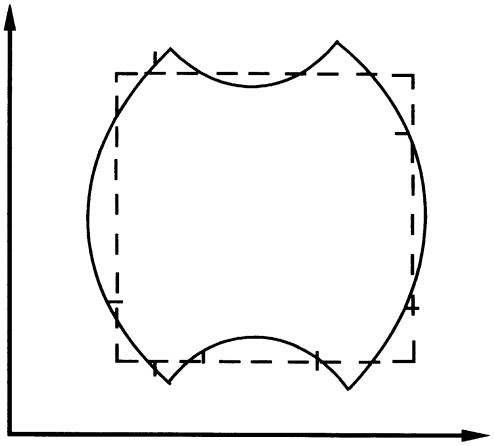


Figure 3-75 Breathing Mode of Reduced Integration

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates directions. Thickness will be updated.

Note: Distortion of element during analysis may cause bad results. Element type 3 is to be preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable for this element.

Element 54

Plane Strain, Eight-Node Distorted Quadrilateral with Reduced Integration

Element type 54 is an eight-node, isoparametric, arbitrary quadrilateral written for plane strain applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 11, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 58 instead. Element type 58 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 54

Second-order, isoparametric, distorted quadrilateral with reduced integration. Plane strain.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-76.

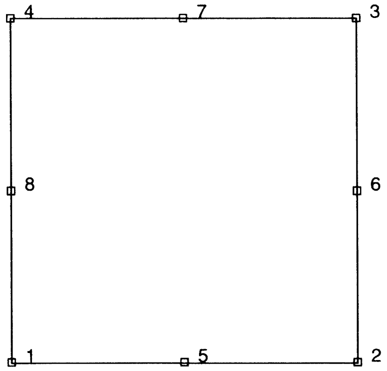


Figure 3-76 Eight-Node, Plane Strain Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees Of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.

Load Type (IBODY)	Description
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

A nine-point integration scheme is used for the integration of body forces (see Figure 3-77).

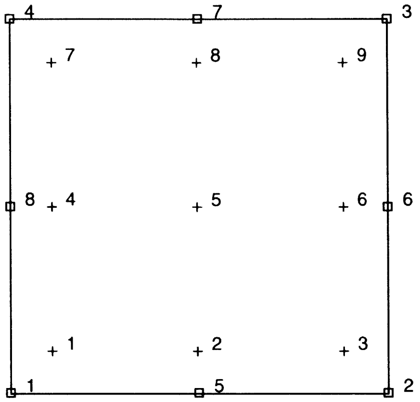


Figure 3-77 Integration Points for Body Forces Calculation

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-78 and **Output Points** on the following page) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy}, shear

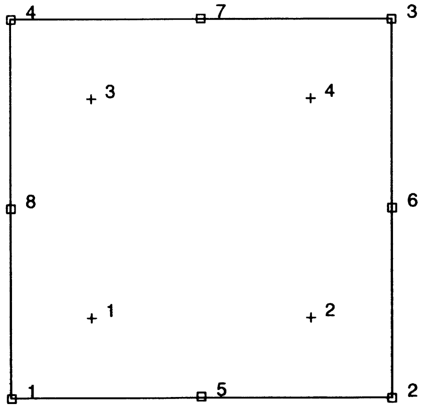


Figure 3-78 Integration Points for Reduced Integration Planar Element

Output of Stresses

Output of stresses is the same as **Strains**.

Transformation

Only in the x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 3-78. This is the usual option for a second-order element with reduced integration.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note: Distortion or element during analysis may cause bad results. Element type 6 or 11 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Element 55

Axisymmetric, Eight-Node Distorted Quadrilateral with Reduced Integration

Element type 55 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements, hence the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 10, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 59 instead. Element type 59 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 55

Second-order, isoparametric, distorted quadrilateral with reduced integration. Axisymmetric formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth between second and third, etc. (see Figure 3-79).

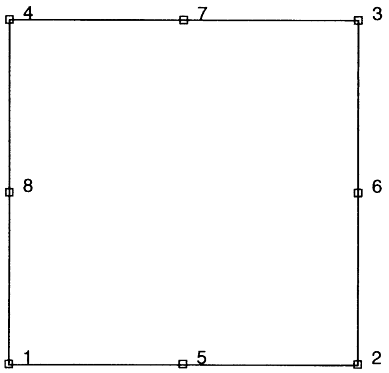


Figure 3-79 Nodes of Eight-Node Axisymmetric Element with Reduced Integration

Geometry

No geometry in input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Tractions

Surface Forces. Pressure and shear forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.

Load Type (IBODY)	Description
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

A nine-point integration scheme is used for the integration of body forces (see Figure 3-80).

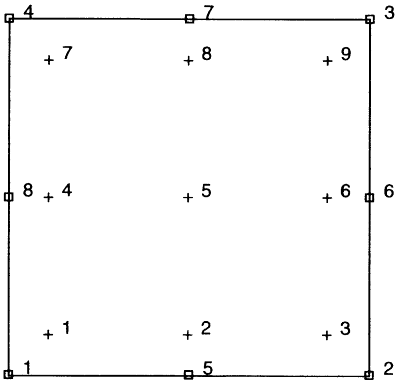


Figure 3-80 Integration Points for Body Force Calculation

Concentrated nodal loads must be the value of the load integrated around the circumference.

Output of Strains

Output of strains at the centroid or element integration points (see **Output Points** on the following page and Figure 3-81) in the following order:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direct
4 = γ_{zr}, shear in the section

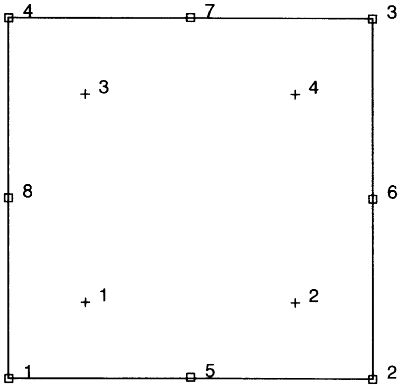


Figure 3-81 Integration Points of Elght-Node, Axisymmetric Element with Reduced Integration

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 3-81. This is the usual option for a second-order element with reduced integration.

Updating Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note: Distortion or element during analysis may cause bad results. Element type 2 or 10 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 70. See Element 70 for a description of the conventions used for entering the flux and film data for this element.

Element 56

Generalized Plane Strain, Distorted Quadrilateral with Reduced Integration

This element is an extension of the plane strain isoparametric quadrilateral (element type 54) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 19, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 60 instead. Element type 60 is also preferable for small strain incompressible elasticity.

This element cannot be used with the element-by-element iterative solver.

Quick Reference

Type 56

Second-order, isoparametric, distorted quadrilateral with reduced integration. Generalized plane strain formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in x-y plane). Then the fifth node between first and second; the sixth node between second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each of the ten nodes. Note that the ninth and tenth nodes may be anywhere in the (x, y) plane.

Degrees of Freedom

Two at each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

One at the ninth node:

1 = Δz = relative z-direction displacement of front and back surfaces. See Figure 3-82.

Two at the tenth node:

1 = $\Delta\theta_x$ = Relative rotation of front and back surfaces about global x-axis.

2 = $\Delta\theta_y$ = Relative rotation of front and back surfaces about global y-axis. See Figure 3-82.

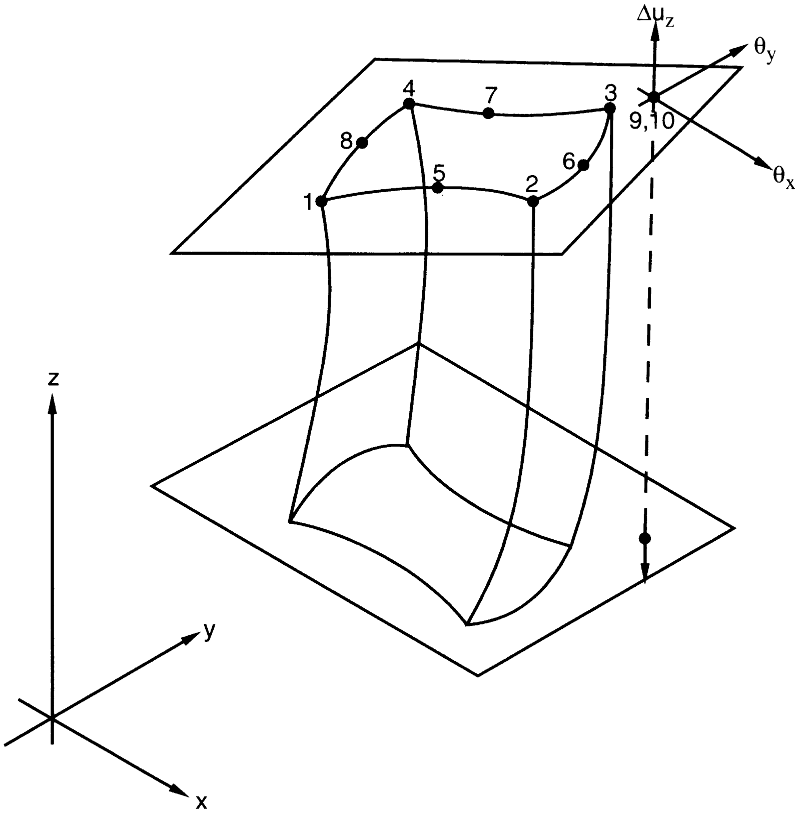


Figure 3-82 Generalized Plane Strain Distorted Quadrilateral with Reduced Integration

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.

Load Type (IBODY)	Description
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

A nine-point integration scheme is used for the integration of body forces (Figure 3-83).

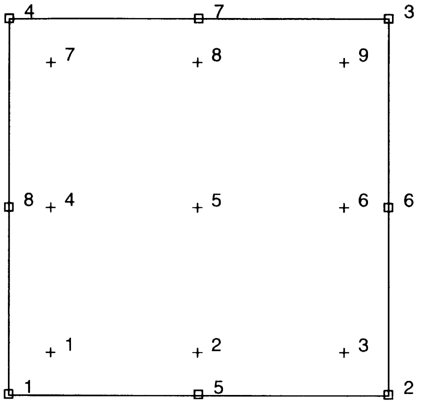


Figure 3-83 Integration Points for Body Force Calculation

Output of Strains

Output of strains at the centroid or element integration points (see **Output Points** on the following page and Figure 3-83) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction direct
4 = γ_{xy}, shear in the (x-y) plane

No γ_{xz}, γ_{yz} or shear – relative rotations of front and back surfaces.

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given as shown in Figure 3-84. This is the usual option for a second-order element with reduced integration.

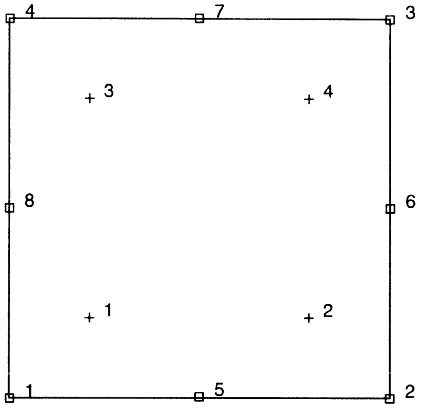


Figure 3-84 Integration Points for Stiffness Matrix

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates. Thickness will be updated.

Note: Distortion or element during analysis may cause bad results. Element type 19 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable for this element.

Element 57

Three-Dimensional 20-Node Brick with Reduced Integration

Element type 57 is a 20-node, isoparametric, arbitrary hexahedral using reduced integration.This element uses triquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 7, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 61 instead. Element type 61 is also preferable for small strain incompressible elasticity.

Note: Reduction to Wedge or Tetrahedron – By simply repeating node numbers on the same spatial position, the element may be reduced as far as a tetrahedron. Element type 127 is preferred for tetrahedrals

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of 20 nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in counterclockwise order when viewed from inside the element. Nodes 5, 6, 7, 8 are corners of the opposite face. Node 5 shares an edge with 1; node 6 shares an edge with 2, etc. Nodes 9, 11, and 12 are the middle of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc.

Note that in most normal cases, the elements will be generated automatically, so that the user need not concern himself with the node numbering scheme.

Integration

The element is integrated numerically using eight points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 3-85). A similar plane follows, moving toward the 5, 6, 7, 8 face. If the CENTROID parameter is used, output occurs at the centroid of the element. If the ALL POINTS parameter is used, the integration points are used for stress output. That is the usual option for a second-order element with reduced integration.

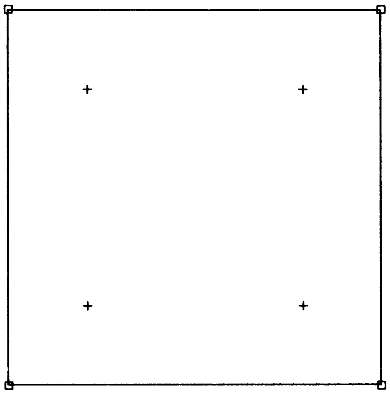


Figure 3-85 Element 57 Integration Plane

The subroutine FORCEM will be called once per integration point when flagged. The magnitude of load defined by DIST LOADS will be ignored and the FORCEM value will be used instead. For nonuniform body force, force values must be provided for 27 integration points, as specified in Figure 3-86 since the reduced integration scheme is not used for the body forces. For nonuniform surface pressures, values need only be supplied for the nine integration points on the face of application.

Nodal (concentrated) loads may also be supplied.

Quick Reference

Type 57

Twenty-nodes, isoparametric arbitrary distorted cube.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and is shown in Figure 3-87.

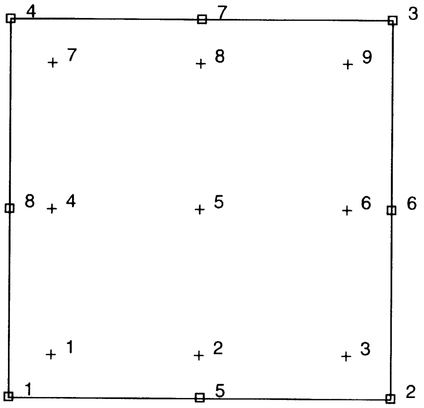


Figure 3-86 Integration Points for Distributed Loads

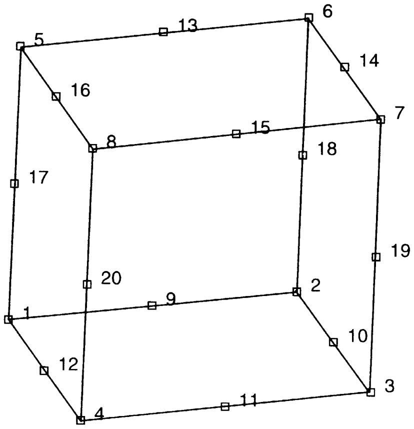


Figure 3-87 Form of Element 57

Geometry

Generally not required. The first field contains the transition thickness if the automatic brick to shell transition constraints are to be used (see Figure 3-88).

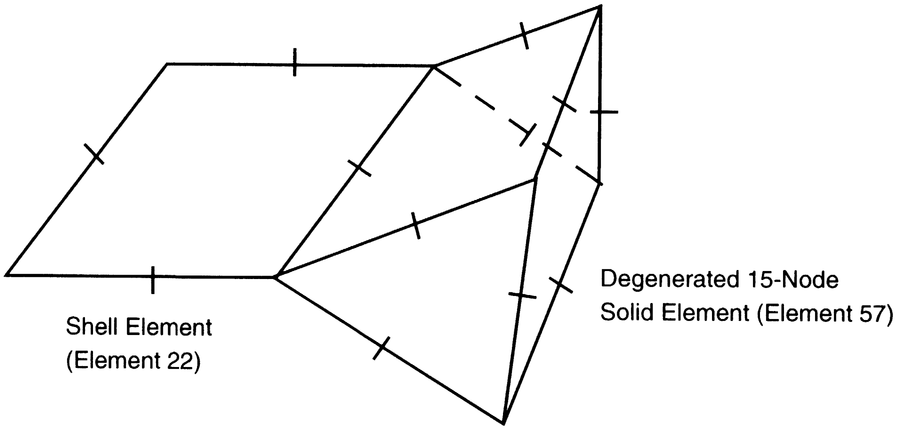


Figure 3-88 Shell-to-Solid Automatic Constraint

Coordinates

Three global coordinates in the x, y and z directions.

Degrees of Freedom

Three global degrees of freedom: u, v and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.

Load Type	Description
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through user subroutine FORCEM).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in FORCEM.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in FORCEM.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in FORCEM.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in FORCEM.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in FORCEM.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.

Load Type	Description
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

1 ε_{xx}
2 ε_{yy}
3 ε_{zz}
4 ε_{xy}
5 ε_{yz}
6 ε_{zx}

Output of Stresses

Same as for **Output of Strains**.

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS. An automatic constraint is available for brick to shell transition meshes. (See **Geometry**.)

Output Points

Centroid or eight Gaussian integration points (see Figure 3-89).

Note: A large bandwidth results in long run times. Optimize as much as possible.

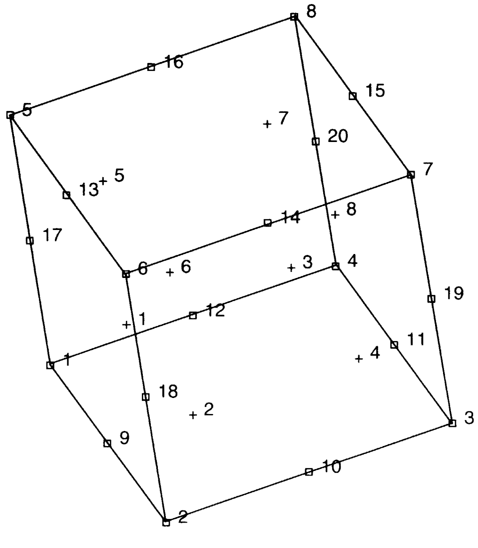


Figure 3-89 Integration Points for 20-Node Brick with Reduced Integration

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates.

Note: Distortion of element during analysis may cause bad results. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 71. See Element 71 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic work is specified with type 101.

Element 58

Plane Strain Eight-Node Distorted Quadrilateral with Reduced Integration Herrmann Formulation

Element type 58 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible plane strain applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 80, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 11 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 58

Second-order, isoparametric, distorted quadrilateral with reduced integration. Plane strain. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-90.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each node.

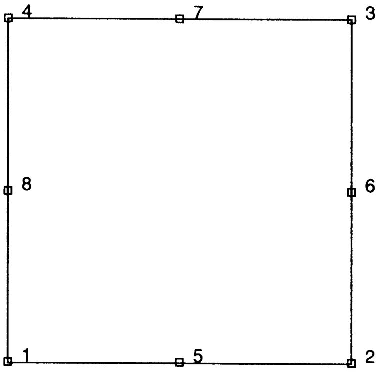


Figure 3-90 Eight-Node, Planar Element

Degrees of Freedom

1 = u = global x-direction displacement
2 = v = global y-direction displacement; additional degree of freedom at corner nodes only
3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.

Load Type	Description
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Note that a nine-point scheme is used for the integration or body forces.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-91 and **Output Points**) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy} shear
5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative pressure (for Mooney or Ogden)

Output of Stresses

Four stresses corresponding to first four **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

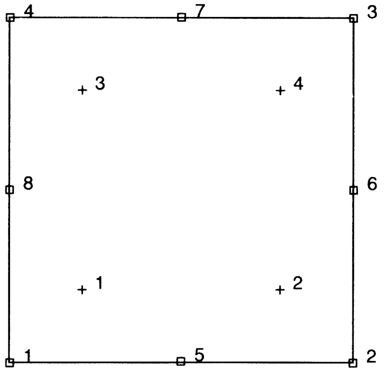


Figure 3-91 Integration Points for Eight-Node Reduced Integration Element

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 3-91. This is the usual option for a second-order element with reduced integration.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Element 59

Axisymmetric, Eight-Node Distorted Quadrilateral with Reduced Integration, Herrmann Formulation

Element type 59 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 82, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 55 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 59

Second-order, isoparametric, distorted quadrilateral with reduced integration axisymmetric formulation. Hybrid formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-92.

Geometry

No geometry input is necessary for this element.

Coordinates

Two global coordinates, z and r, at each node.

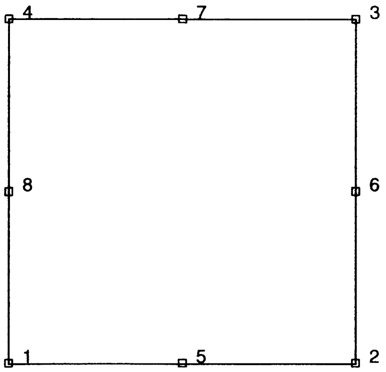


Figure 3-92 Eight-Node Axisymmetric Element

Degrees of Freedom

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Additional degree of freedom at each corner node:

3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Tractions

Surface Forces. Pressure and shear surface forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.

Load Type (IBODY)	Description
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Note that a nine-point scheme is used for integration of body forces.

Output of Strains

Output of strains at the centroid of element integration points (see Figure 3-93 and **Output Points**) in the following order:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direct
4 = γ_{zr} shear in the section
5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative pressure (for Mooney or Ogden)

Output of Stresses

Output of stresses is the same as first four **Output of Strains**.

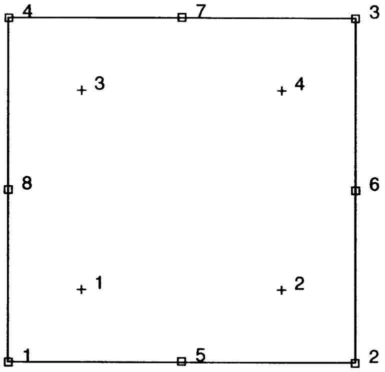


Figure 3-93 Integration Points for Eight-Node Reduced Integration Element

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given as shown in Figure 3-93. This is the usual option for a second-order element with reduced integration.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available. Finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 70. See Element 70 for a description of the conventions used for entering the flux and film data for this element.

Element 60

Generalized Plane Strain Distorted Quadrilateral with Reduced Integration – Herrmann Formulation

This element is an extension of the plane strain isoparametric quadrilateral (element type 58) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 81, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 56 when other material behavior, such as plasticity, must be represented.

This element cannot be used with the element-by-element iterative solver.

Quick Reference

Type 60

Second-order, isoparametric, distorted quadrilateral with reduced integration, generalized plane strain, hybrid formulation. See Volume A for generalized plane strain theory, see Volume F for incompressible and nearly incompressible theory, and for Mooney material formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in x-y plane). Then the fifth node between first and second; the sixth node between second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

At all ten nodes, two global coordinates, x and y. Note that the ninth and tenth nodes may be anywhere in the (xi) plane.

Degrees Of Freedom

At each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Additional degree of freedom at the first four (corner) nodes:

3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

One at the ninth node:

1 = Δz = relative z-direction displacement of front and back surfaces (see Figure 3-94).

Two at the tenth node:

1 = $\Delta\theta_x$ = relative rotation of front and back surfaces about global x-axis.
2 = $\Delta\theta_y$ = relative rotation of front and back surfaces about global y-axis (see Figure 3-94).

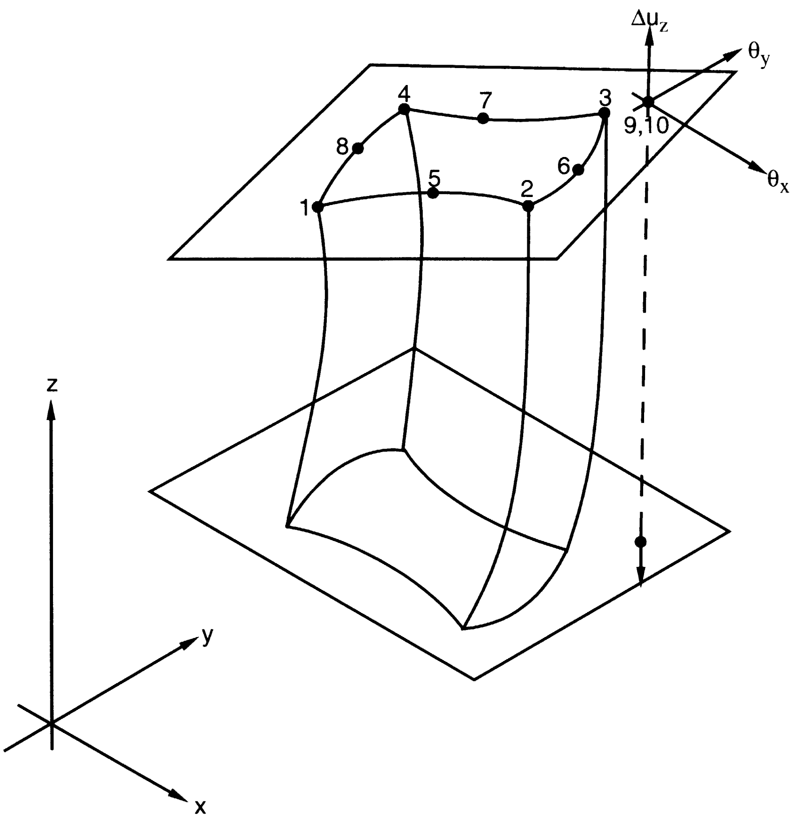


Figure 3-94 Generalized Plane Strain Distorted Quadrilateral

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.

Load Type	Description
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

Note that a nine-point scheme is used for the integration or body forces (see Figure 3-95).

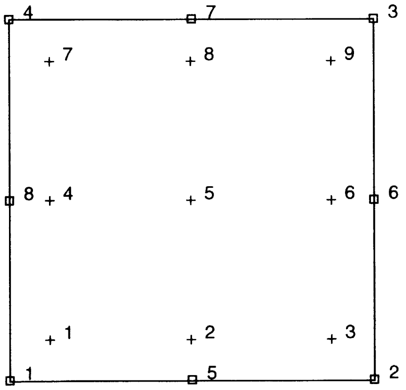


Figure 3-95 Integration Points for Body Force Calculations

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-96 and **Output Points**) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy} shear in the (x-y) plane

No γ_{xz} or γ_{yz} shear.

5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Output of stresses is the same as first four **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

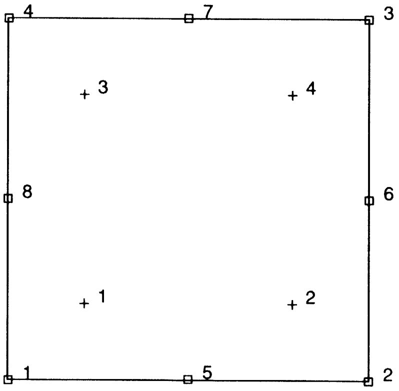


Figure 3-96 Integration Points for Eight-Node Reduced Integration Element

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given as shown in Figure 3-96. This is the usual option for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness (beam height) and/or the beam width can be considered as design variables.

Element 61

Three-Dimensional, 20-Node Brick with Reduced Integration – Herrmann Formulation

Element type 61 is a 20-node, isoparametric, arbitrary hexahedral written for incompressible applications using reduced integration. This element uses triquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using trilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 84, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 57 when other material behavior, such as plasticity, must be represented.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of 20 nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in counterclockwise order when viewed from inside the element. Nodes 5, 6, 7, 8 are the corners of the opposite face; node 5 shares an edge with 1, 6 with 2, etc. Nodes 9, 10, 11, 12 are the middle of the edges of the 1, 2, 3, 4 face; node between 1 and 2, 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc.

Note that in most normal cases, the elements will be generated automatically, so that the user need not concern himself with the node numbering scheme.

Reduction to Wedge or Tetrahedron

The element may be reduced as far as a tetrahedron, simply be repeating node numbers. Element type 130 would be preferred for tetrahedrals.

Integration

The element is integrated numerically using eight points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 3-97). A similar plane follows, moving toward the 5, 6, 7, 8 face. The "centroid" of the element is used for stress output if the CENTROID parameter is flagged.

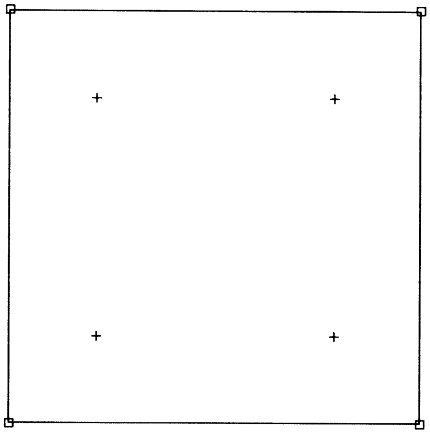


Figure 3-97 Plane of Integration Points for Reduced Integration Brick Element

The subroutine FORCEM will be called once per integration point when flagged. The magnitude of load defined by TRACTIONS will be ignored and the FORCEM value will be used instead.

Note that for integration of body forces, a 27-point integration scheme is used. Hence, for nonuniform body force, values must be provided for 27 points. Similarly, for nonuniform surface pressures, values need be supplied for nine integration points on the face of application.

Nodal (concentrated) loads may also be supplied.

Quick Reference

Type 61

Twenty-nodes, isoparametric arbitrary distorted cube with reduced integration. Herrmann formulation. See Volume F, for details on this theory.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and is shown in Figure 3-98.

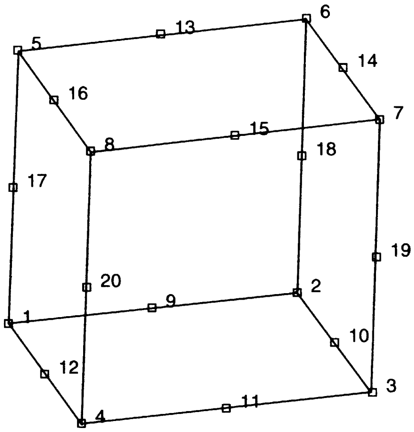


Figure 3-98 Twenty-Node Brick Element

Geometry

Not required.

Coordinates

Three global coordinates in the x, y, and z directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w, at all nodes. Additional degrees of freedom at corner nodes (first 8 nodes) for Herrmann or Mooney is as follows:

σ_{kk}/E; i.e., mean pressure variable (for Herrmann) or
-p; i.e. negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through user subroutine FORCEM).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in FORCEM.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in FORCEM.
28	Uniform pressure on 3-2-6-7 face.

Load Type	Description
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in FORCEM.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in FORCEM.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in FORCEM.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.

Load Type	Description
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Stress-strain output in global components:

1 – ε_{xx}
2 – ε_{yy}
3 – ε_{zz}
4 – γ_{xz}
5 – γ_{yz}
6 – γ_{zx}
7 – $\sigma_{kk}/2G(1 + \upsilon)$ = mean pressure variable (for Herrmann)
h = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Output for stresses is the same as the first six **Output of Strains.**

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or eight Gaussian integration points (see Figure 3-97).

Note: A large bandwidth results in long run times – optimize.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 71. See Element 71 for a description of the conventions used for entering the flux and film data for this element.

Element 62

Axisymmetric, Eight-Node Quadrilateral for Arbitrary Loading (Fourier)

Element type 62 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This Fourier element may only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 62

Second-order, isoparametric, distorted quadrilateral for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique.

Connectivity

Corner numbered first in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between third, etc. See Figure 3-99.

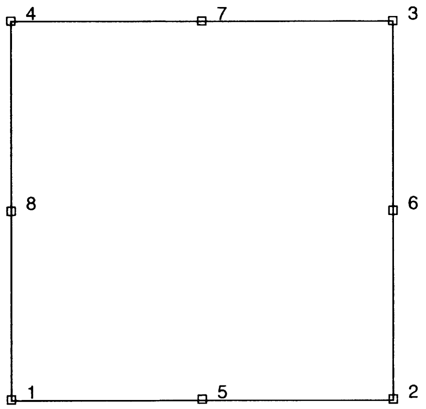


Figure 3-99 Eight-Node Fourier Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Three at each node:

1 = u = global z-direction (axial)
2 = v = global r-direction (radial)
3 = θ = global θ-direction (circumferential displacement)

Tractions

Surface Forces. Pressure and shear surface forces available for this element are listed below:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒→2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.

Load Type (IBODY)	Description
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Body forces (per unit volume). Load type 2 is uniform body in the z-direction (axial), load type 3 is nonuniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial), load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, user subroutine FORCEM must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force may be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-100 and **Output Points**) in the following order.

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, direct
4 = γ_{zr}, in-plane shear
5 = $\gamma_{r\theta}$, out-of-plane shear
6 = $\gamma_{\theta z}$, out-of-plane shear

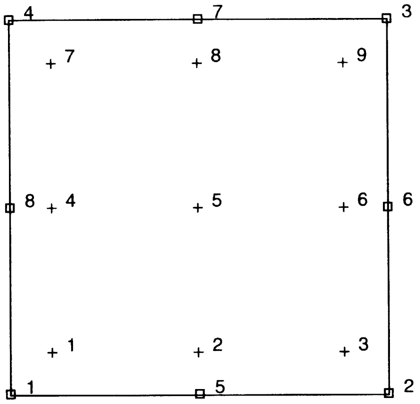


Figure 3-100 Integration Points of Eight-Node 2D Element

Output of Stresses

Same as for **Output of Strains**.

Transformations

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, the output is given for all nine integration points.

Element 63

Axisymmetric, Eight-Node Distorted Quadrilateral for Arbitrary Loading, Herrmann Formulation (Fourier)

Element type 63 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using nine-point Gaussian integration.

This Fourier element may only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 63

Second-order, isoparametric quadrilateral for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique. Hybrid formulation for incompressible or nearly incompressible materials. See Volume F for details.

Connectivity

Corners numbered first in counterclockwise order. Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-101 and Figure 3-102.

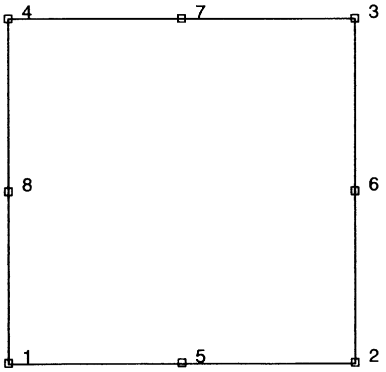


Figure 3-101 Eight-Node Fourier Herrmann Element

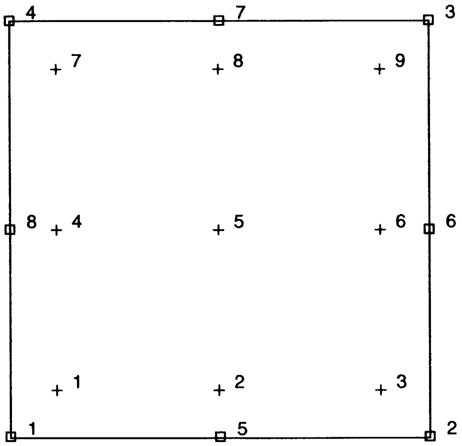


Figure 3-102 Integration Points for Eight-Node Fourier Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each of the eight nodes.

Degrees of Freedom

1 = u = global z-direction (axial)
2 = v = global r-direction (radial)
3 = θ = global θ-direction (circumferential displacement)
Additional degree of freedom at each corner node.
4 = σ_{kk}/E = mean pressure variable (for Herrmann formulation).

Tractions

Surface Forces. Pressure and shear forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Body forces (per unit volume). Load type 2 is uniform body in the z-direction (axial), load type 3 is nonuniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial), load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, user subroutine FORCEM must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force may be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-102 and **Output Points**) in the following order:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, direct
4 = γ_{zr}, in-plane shear
5 = $\gamma_{r\theta}$, out-of-plane shear
6 = $\gamma_{\theta z}$, out-of-plane shear
7 = $\sigma_{kk}/2G(1 + \upsilon)$, mean pressure variable (for Herrmann)

Output of Stresses

Same as for **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, shown as point 5 in Figure 3-102.

If the ALL POINTS parameter is used, the output is given for all nine integration points.

Element 64

Isoparametric, Three-Node Truss

This element is a quadratic, three-node truss with constant cross section. The strain-displacement relations are written for large strain, large displacement analysis. Three-point Gaussian integration is used along the element. The degrees of freedom are the u, v, and w displacements at the three nodes of the element.

This element is very useful as reinforcement element in conjunction with the two- and three-dimensional second order isoparametric elements in the program. Possible applications include the use as a string in membrane or as discrete reinforcement string in composite materials. All constitutive relations may be used with this element.

Quick Reference

Type 64

Three-dimensional, three-node, isoparametric truss.

Connectivity

Three nodes per element (see Figure 3-103).

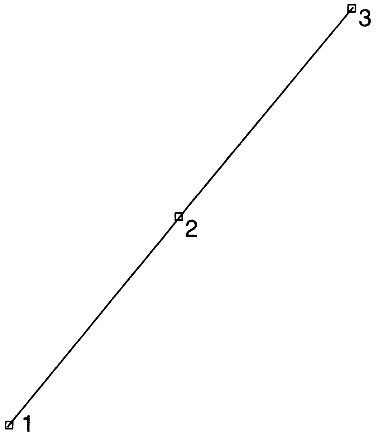


Figure 3-103 Isoparametric Truss Element

Geometry

The cross-sectional area is input in the first data field (EGEOM1). The other two data fields are not used. If not specified, the cross-sectional area defaults to 1.0.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = w displacement

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform load in the direction of the global x-axis per unit volume.
1	Uniform load in the direction of the global y-axis per unit volume.
2	Uniform load in the direction of the global z-axis per unit volume.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Strains

Uniaxial in the truss member.

Output of Stresses

Uniaxial in the truss member.

Transformation

The three global degrees of freedom for any node may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or three Gaussian integration points along the truss if the ALL POINTS parameter is used. First point is closest to first node given; second point is centroid; third point is closest to third node of truss. See Figure 3-104.

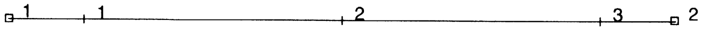


Figure 3-104 Integration Points for Element Type 64

Updated Lagrange and Finite Strain Plasticity

Capability is available; area will be updated.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 65. See Element 65 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The cross-sectional area can be considered as a design variable.

Element 65

Heat Transfer Element, Three-Node Link

This element is a quadratic heat link with constant cross-sectional area. It is the heat-transfer equivalent of element type 64.

Quick Reference

Type 65

Three-dimensional, three-node, heat transfer link.

Connectivity

Three nodes per element (see Figure 3-105).

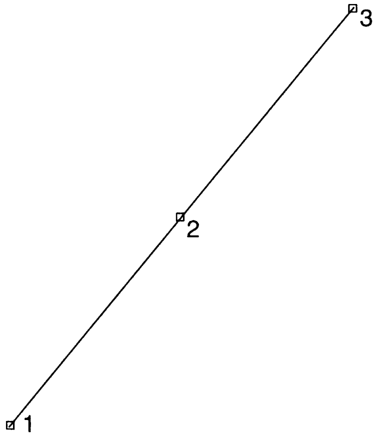


Figure 3-105 Three-Node Heat Transfer Element

Geometry

The cross-sectional area is input in the first data field (EGEOM1); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

One degree of freedom per node:

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)

Fluxes

Distributed fluxes according to value of IBODY are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross sectional area)
1	Uniform flux on last node (per cross sectional area)
2	Volumetric flux on entire element (per volume)

Films

Same specifications as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is not available.

Current

Same specification as **Fluxes**.

Output Points

Centroid or three Gaussian integration points along the truss if the ALL POINTS parameter is used. First point is closest to first node given; second point is centroid; third point is closest to third node. See Figure 3-106.

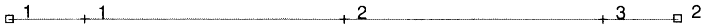


Figure 3-106 Integration Points for Element Type 65

Element 66

Eight-Node Axisymmetric Herrmann Quadrilateral with Twist

The modified axisymmetric (includes a twist mode of deformation), eight-node distorted quadrilateral – suitable for materially linear, elastic, and incompressible or nearly incompressible deformation (Herrmann formulation) as well as nonlinear elastic incompressible Mooney-Rivlin or Ogden behavior and/or some other higher order forms of hyperelastic deformation.

Quick Reference

Type 66

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation modified to include a twist mode of deformation. Hybrid (Herrmann) formulation of incompressible or nearly incompressible materials. See Volume F.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane), then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-107.

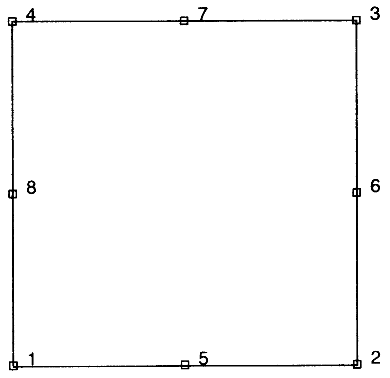


Figure 3-107 Eight-Node Axisymmetric Herrmann with Twist

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

1 = u_z, global z-direction displacement (axial).
2 = u_R, global R-direction displacement (radial). See Figure 3-108.
3 = u_θ, global θ-direction displacement (tangential) in radians. See Figure 3-108.
4 = P, hydrostatic pressure variable – only at the corner nodes.

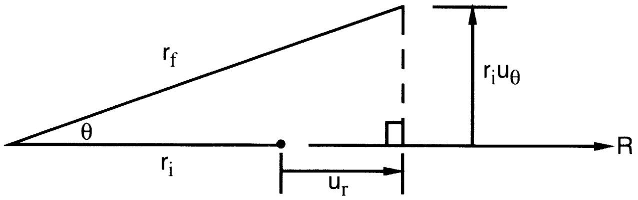


Figure 3-108 Radial and Tangent Displacements

Tractions

Surface Forces. Pressure and shear (in the z-r plane) forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force per unit volume in the z-direction (axial).
3	Nonuniform body force per unit volume in the z-direction (axial).
4	Uniform body force per unit volume in the r-direction (radial).
5	Nonuniform body force per unit volume in the r-direction (radial).
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.

Load Type (IBODY)	Description
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Note that a nine-point scheme is used for the integration or body forces.

Note that "loads" associated with the third degree of freedom, can only be specified, at this time, as concentrated nodal values. These "loads" actually correspond to a torque in the θ - Z plane. Also note that in specifying any concentrated nodal load, the value to be used should be that obtained by integration around the entire circumference defined by the radius of the nodal point.

Output of Strains

The total components of strain are printed out at the centroid or element integration points (see Figure 3-109 and **Output Points**) in the following order:

1 = E_{ZZ}, direct axial
2 = E_{RR}, direct radial
3 = $E_{\theta\theta}$, direct hoop
4 = $2E_{ZR}$ (= γ_{ZR}, the engineering definition of strain for small deformations), shear in the section
5 = $2E_{R\theta}$ (= $\gamma_{R\theta}$, for small deformations), warping of radial lines
6 = $2E_{\theta Z}$ (= $\gamma_{\theta Z}$, for small deformations), twist deformation

where E_{ij} are the physical components of the Green's tensor referred to the initial cylindrical reference system (i.e., Lagrangian strain).

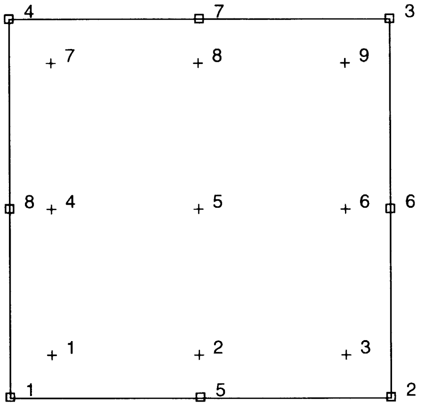


Figure 3-109 Integration Points for Element 66

Output Of Stresses

The stress components that are conjugate to the Green's strain components (listed above), are also printed. These are the physical components of the symmetric second Piola-Kirchhoff stress, S_{ij}. In addition, the physical components of the Cauchy stress tensor are also printed.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-109. If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-109. This is the usual option for a second-order element, particularly when material and/or geometric nonlinearities are present.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case. See description of MARC element type 67 for a compressible (i.e., conventional isoparametric axisymmetric formulation) element that is compatible (i.e., includes the twist deformation) with this Herrmann type element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 42. See Element 42 for a description of the conventions used for entering the flux and film data for this element.

Element 67

Eight-Node Axisymmetric Quadrilateral with Twist

Element type 67 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications with torsional strains. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 20, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 66 instead. Element type 66 is also preferable for small strain incompressible elasticity.

Notice that there is no friction contribution in the torsional direction when the CONTACT option is used.

Quick Reference

Type 67

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation modified to include a twist mode of deformation. Conventional counterpart to incompressible element type 66 described above.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-110.

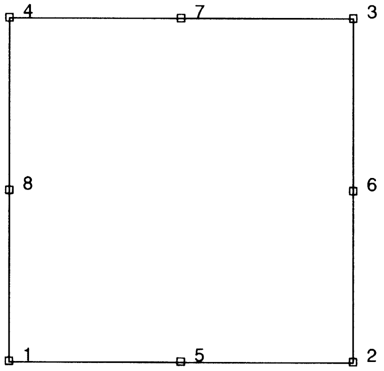


Figure 3-110 Eight-Node Axisymmetric with Twist

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

$1 = u_z$, global z-direction displacement (axial).
$2 = u_R$, global R-direction displacement (radial). See Figure 3-111.
$3 = u_\theta$, global θ-direction displacement (tangential) in radians. See Figure 3-111.

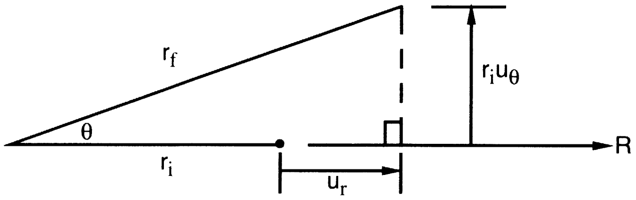


Figure 3-111 Radial and Tangent Displacements

Tractions

Surface forces. Pressure and shear (in the r-z plane) forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force per unit volume in the z-direction (axial).
3	Nonuniform body force per unit volume in the z-direction (axial).
4	Uniform body force per unit volume in the r-direction (radial).
5	Nonuniform body force per unit volume in the r-direction. (radial)
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.

Load Type (IBODY)	Description
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Note that "loads" associated with the third degree of freedom, θ, can only be specified, at this time, as concentrated nodal values. These "loads" actually correspond to a torque in the θ - z plane. Also note that in specifying any concentrated nodal load, the value to be used should be that obtained by integration around the entire circumference defined by the radius of the nodal point.

Output of Strains

The total components of strain are printed out at the centroid or element integration points (see Figure 3-112 and **Output Points**) in the following order:

1 = E_{ZZ}, direct axial
2 = E_{RR}, direct radial
3 = $E_{\theta\theta}$, direct hoop
4 = $2E_{ZR}$ (= γ_{ZR}, the engineering definition of strain for small deformations), shear in the section
5 = $2E_{R\theta}$ (= $\gamma_{R\theta}$ for small deformations), warping of radial lines
6 = $2E_{\theta Z}$ (= $\gamma_{\theta Z}$, for small deformations), twist deformation

where E_{ij} are the physical components of the Green's strain tensor referred to the initial cylindrical reference system (i.e., Lagrangian strain).

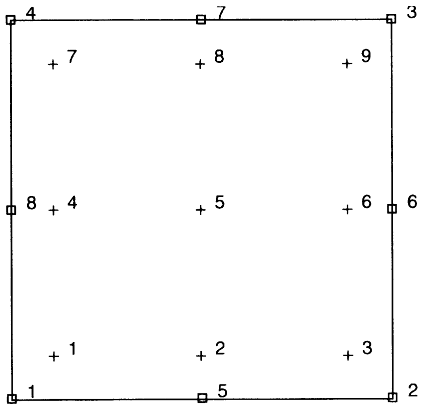


Figure 3-112 Integration Points for Element 67

Output of Stresses

The stress components, that are conjugate to the Green's strain components (listed above), are also printed. These are the physical components of the symmetric second Piola-Kirchhoff stress, S_{ij}.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-112. If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-112. This is the usual option for a second-order element, particularly when material and/or geometric nonlinearities are present.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 42. See Element 42 for a description of the conventions used for entering the flux and film data for this element.

Element 68

Elastic, Four-Node Shear Panel

This is a linear-elastic shear panel of arbitrary shape. A shear panel is an idealized model of an elastic sheet. If there are stiffeners present, the panel resists the shearing forces and the stiffeners resist normal and bending forces. The generalization to an arbitrary shape is due to S. J. Garvey. Due to the simplifications involved, the response of the element is restricted to linear materials and large displacement effects are neglected. The stiffness matrix is found in closed form.

Geometric Basis

The element is formulated in a local plane defined by the two diagonals. If they do not intersect, the plane is located midway between the diagonals.

Displacements

The displacements at each node are:

u, v, w global Cartesian components

Connectivity Specification

The element has four nodes. They can be listed in clockwise or counterclockwise order.

Quick Reference

Type 68

Linear-elastic shear panel.

Connectivity

Four nodes per element (see Figure 3-113).

Geometry

Thickness in first data field (EGEOM1).

---

[1] Garvey, S. J., "The Quadrilateral Shear Panel", *Aircraft Engineering*, p. 134, May 1951.

Coordinates

1 = x
2 = y
3 = z

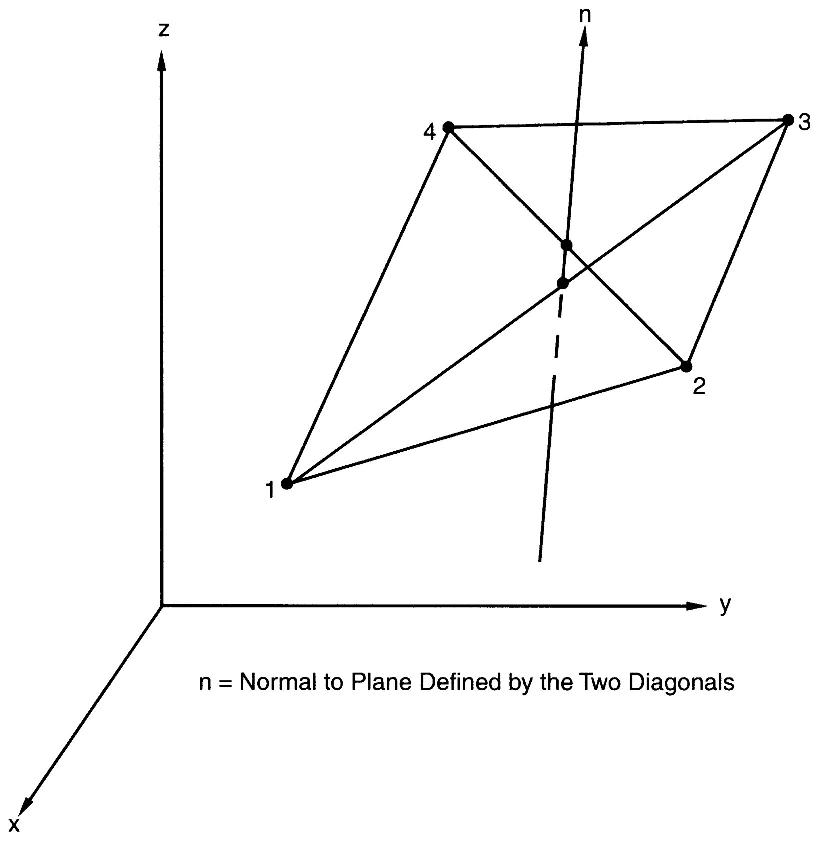


Figure 3-113 Shear Panel

Degrees of Freedom

1 = u
2 = v
3 = w

Tractions

Only concentrated forces at nodal points.

Output of Stresses

Shear stress at all four nodal points. Average shear stresses and maximum shear stresses are printed.

$$\tau_{avg} = 1/4(\tau_1 + \tau_2 + \tau_3 + \tau_4)$$

$$\tau_{max} = \max(|\tau_1|, |\tau_2|, |\tau_3|, |\tau_4|)$$

Transformation

The degrees of freedom may be transformed to local directions.

Tying

Use tying (UFORMS) needed to formulate constraints between (u, v, w) degrees of freedom of beams and shear panels.

Updated Lagrangian Procedure and Finite Strain Plasticity

Capability is not available.

Element 69

Eight-Node Planar Biquadratic Quadrilateral with Reduced Integration (Heat Transfer Element)

Element type 69 is an eight-node, isoparametric, arbitrary quadrilateral written for planar heat transfer applications using reduced integration. This element may also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the **Quick Reference**.

The conductivity of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

Quick Reference

Type 69

Second-order, distorted heat quadrilateral with reduced integration.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes starting with node 5 in between 1 and 2, and so on (see Figure 3-114).

Geometry

Thickness stored in EGEOM1 field. If not specified, unit thickness is assumed.

Coordinates

Two global coordinates per node – x and y.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

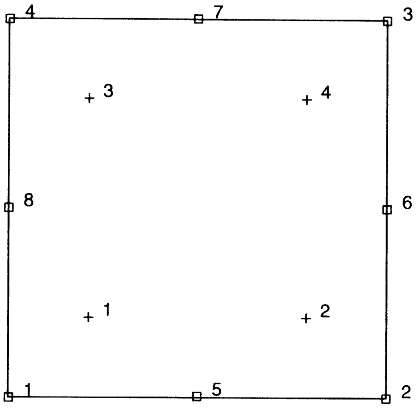


Figure 3-114 Eight-Node Planar Heat Quadrilateral with Reduced Integration

Fluxes

Surface Fluxes

Surface fluxes are specified below. All are per unit surface area. All nonuniform fluxes are specified through subroutine FLUX.

Flux Type (IBODY)	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.

Flux Type (IBODY)	Description
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through subroutine FLUX.

Films

Same specification as **Fluxes.**

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes.**

Charge

Same specifications as **Fluxes.**

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
2	(4 - 8 - 1) to (2 - 6 - 3)

Element 70

Eight-Node Axisymmetric Biquadrilateral with Reduced Integration (Heat Transfer Element)

Element type 70 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric heat transfer applications using reduced integration. This element may also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the Quick Reference.

The conductivity of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

Quick Reference

Type 70

Second-order, distorted axisymmetric heat quadrilateral with reduced integration.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes with node 5 located between 1 and 2, etc. (see Figure 3-115).

Geometry

Not applicable.

Coordinates

Two global coordinates per node, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

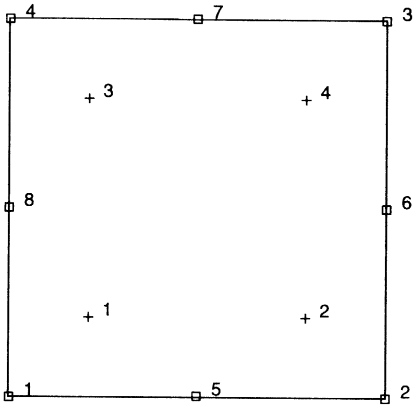


Figure 3-115 Eight-Node Axisymmetric Heat Quadrilateral with Reduced Integration

Fluxes

Surface Fluxes

Surface fluxes are specified below. Surface flux magnitudes are input per unit surface area. The magnitude of nonuniform surface fluxes must be specified through subroutine FLUX.

Flux Type	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.

Flux Type	Description
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume), type 3 is nonuniform flux per unit volume, with magnitude given through subroutine FLUX.

Films

Same specifications as **Fluxes**.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
2	(4 - 8 - 1) to (2 - 6 - 3)

View Factors Calculation For Radiation

Capability is available.

Element 71

Three-Dimensional 20-Node Brick with Reduced Integration (Heat Transfer Element)

Element type 71 is a 20-node, isoparametric, arbitrary quadrilateral written for three-dimensional heat transfer applications using reduced integration. This element may also be used for electrostatic applications.

This element uses triquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the **Quick Reference**.

The conductivity of this element is formed using eight-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, and 4 are corners of one face, given in a counterclockwise direction when viewed from inside the element. Nodes 5, 6, 7, and 8 are the corners of the opposite face; node 5 shares an edge with 1; node 6 with 2, etc. Nodes 9, 10, 11, and 12 are the middles of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, and 16 are midpoints on the 5, 6, 7, and 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc. (see Figure 3-116).

Integration

The element is integrated numerically using eight points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 3-117). One similar plane follows, closest to the 5, 6, 7, 8 face.

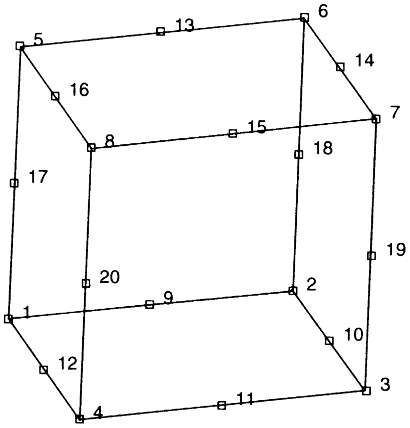


Figure 3-116 Form of Element 71

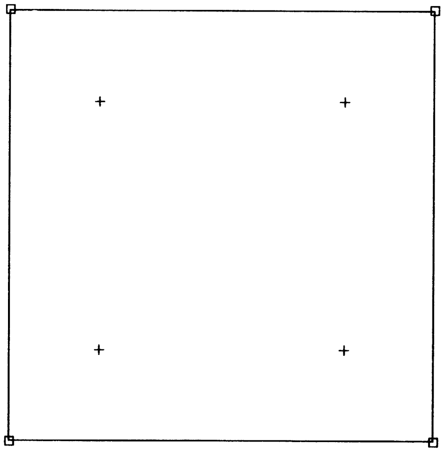


Figure 3-117 Points of Integration in a Sample Integration Plane

Reduction to Wedge or Tetrahedron

The element may be reduced as far as a tetrahedron, simply by repeating node numbers on the same spatial node position. Element type 122 is preferred for tetrahedrals.

Notes: A large bandwidth results in long run times. Optimize as much as possible.

The lumped specific heat option gives poor results with this element at early times in transient solutions. If accurate transient analysis is required, the user should not use the lumping option with this element.

Quick Reference

Type 71

Twenty-node isoparametric brick (heat transfer element) with reduced integration.

Connectivity

Twenty-nodes numbered as described in the connectivity write-up for this element, and as shown in Figure 3-116.

Geometry

Not applicable.

Coordinates

Three global coordinates in x-, y-, z-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Distributed fluxes given by a type specification are as follows:

Load Type	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform flux on 1-2-3-4 face (FLUX).
2	Uniform body flux.
3	Nonuniform body flux (FLUX).
4	Uniform flux on 6-5-8-7 face.

Load Type	Description
5	Nonuniform flux on 6-5-8-7 face (FLUX).
6	Uniform flux on 2-1-5-6 face.
7	Nonuniform flux on 2-1-5-6 face (FLUX).
8	Uniform flux on 3-2-6-7 face.
9	Nonuniform flux on 3-2-6-7 face (FLUX).
10	Uniform flux on 4-3-7-8 face.
11	Nonuniform flux on 4-3-7-8 face (FLUX).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (FLUX).

For type=3, the value of P in the user subroutine FLUX is the magnitude of volumetric flux at volumetric integration point NN of element N. For type odd but not equal to 3, P is the magnitude of surface flux at surface integration point NN of element N. Surface flux is positive when heat energy is added to the element.

Films

Same specifications as **Fluxes**.

Tying

Use subroutine UFORMS.

Output Points

Centroid or eight Gaussian integration points.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 72

Bilinear Constrained Shell Element

This is an eight-node, thin-shell element with zero-order degrees of freedom. Bilinear interpolation is used both for global displacements and coordinates. Global rotations are interpolated quadratically from the rotation vectors at the centroid and at the midside nodes. At these midside nodes, constraints are imposed on the rotations by relating the rotation normal to the boundary as well as the rotation about the surface normal to the local displacements. In addition, all three rotation components are related to the local displacements at the centroid. In this way a very efficient and simple element is obtained. The element can be used in curved shell analysis but also for the analysis of complicated plate structures. For the latter case, this element is easier to use than the usual plate elements, since the absence of local higher order degrees of freedom will allow for direct connections between folded plates without tying requirements along the folds.

Due to its simple formulation when compared to the standard higher-order shell elements, it is less expensive, and therefore, very attractive in nonlinear analyses. The element is fairly insensitive to distortion, particularly if the corner nodes lie in the same plane. In that case, all constant bending modes are represented exactly. The element can be degenerated to a triangle by collapsing one of the sides. The midside node on that side then becomes a dummy node and can be given the same number as the end nodes of that side. Thus, the element degenerates to a constant shear – constant bending plane triangle. All constitutive relations may be used with this element.

Geometric Basis

The element is defined geometrically by the (x, y, z) coordinates of the four corner nodes. Although coordinates may be specified at the midside nodes, they will be ignored and the edges treated as straight lines.

Due to the bilinear interpolation, the surface will form a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option. The stress-strain output is given in local orthogonal surface directions, V_1, V_2, and V_3, which for the centroid are defined in the following way (see Figure 3-118).

First, the vectors tangent to the curves with constant isoparametric coordinates ξ and η are normalized:

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s/\sqrt{2}\,|s|, \quad \bar{d} = d/\sqrt{2}\,|d|$$

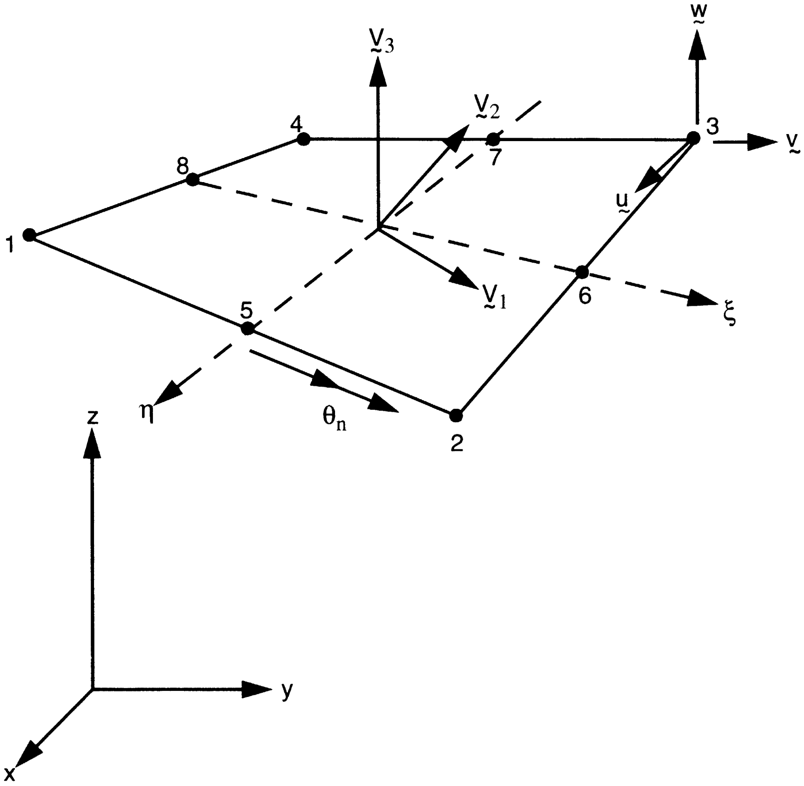


Figure 3-118 Bilinear Constrained Shell Element 72

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}$$

$$V_2 = \bar{s} - \bar{d}$$

and

$$V_3 = V_1 \times V_2$$

In this way the vectors $\frac{\partial x}{\partial \xi}, \frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions for the Gaussian integrations points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The nodal displacement variables are as follows:

At the four corner nodes:	u, v, w Cartesian displacement components.
At the four midside nodes:	ϕ_n, rotation of the element edge about itself. The positive rotation vector points from the corner with the lower (external) node number to the corner with the higher (external) node number.

Due to the ease in modeling intersecting plates (pin joint or moment carrying joint), no special tying types have been developed for this element type.

Connectivity Specification

The four corner nodes of the element are input first, proceeding continuously around the element edges. Then the node between corners 1 and 2 is given, followed by the midside nodes between corners 2 and 3, 3 and 4, and 4 and 1.

In case the element is degenerated to a triangle by collapsing one of the sides, the midside node on the collapsed side no longer has any stiffness associated with it. The midside node is given the same node number as the two corner nodes. Hence, the connectivity of the triangle in Figure 3-119 can for instance be specified as:

1, 72, 1, 2, 3, 3, 4, 5, 3, 6,

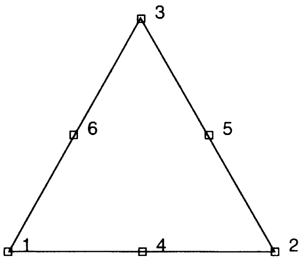


Figure 3-119 Collapsed Shell Element Type 72

Quick Reference

Type 72

Bilinear, constrained eight-node shell element.

Connectivity

Eight nodes: corners nodes given first, proceeding continually around the element.

Then the midside nodes are given as:

5 = between corners 1 and 2
6 = between corners 2 and 3
7 = between corners 3 and 4
8 = between corners 4 and 1

The element may be collapsed to a triangle. The midside node on the collapsed edge has no associated stiffness.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

(x, y, z) global Cartesian coordinates are given. Note that since the element is assumed to have straight edges, the coordinates associated with the midside nodes are not required by the program.

Degrees of Freedom

At four corner nodes:

1 = u = global Cartesian x-direction displacement
2 = v = global Cartesian y-direction displacement
3 = w = global Cartesian z-direction displacement

At midside nodes:

1 = ϕ_n = rotation of edge about itself. The positive rotation vector points from the corner with the lower (external) node number to the corner with the higher external node number.

Tractions

Types of distributed loading are as follows:

Load Type	Direction
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction.
5	Nonuniform load per surface area in arbitrary direction.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to the edge.
12	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in FORCEM on the 1-2 edge.
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to the edge.
22	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in FORCEM on the 2-3 edge.

Load Type	Direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to the edge.
32	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction given in FORCEM on the 3-4 edge.
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to the edge.
42	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in FORCEM on the 4-1 edge.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads and moments may also be applied at the nodes.

Output of Strains

Generalized strain components are:

Middle surface stretches ε_{11} ε_{22} ε_{12}
Middle surface curvatures κ_{11} κ_{22} κ_{12}

in local (V_1, V_2, V_3) system.

Output of Stresses

σ_{11}, σ_{22}, σ_{12} in local V_1, V_2, V_3, system given at equally spaced layers through the thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement components at corner nodes may be transformed to local direction.

The TRANSFORMATION option should not be invoked on the midside nodes.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-120. If the ALL POINTS parameter is used, the output is given in the first four points as shown in Figure 3-120.

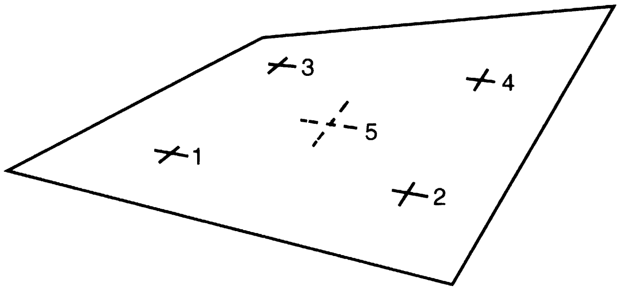


Figure 3-120 Integration Points for Element 72

Section Stress Integration

Integration through the shell thickness is performed numerically using Simpson's rule.

Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Three points are enough for linear response; seven points are enough for simple plasticity or creep analysis; eleven points are enough for complex plasticity or creep (e.g., dynamic plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with beam element types 76 and 77.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output for stresses and strains as for total Lagrangian approach.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 73

Axisymmetric, Eight-Node Quadrilateral for Arbitrary Loading with Reduced Integration (Fourier)

Element type 73 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This Fourier element may only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 73

Second-order, isoparametric, distorted quadrilateral, for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique, using reduced integration.

Connectivity

Corner numbered first in counterclockwise order (right-handed convention). The fifth node between first and second; the sixth node between second and third, etc. See Figure 3-121.

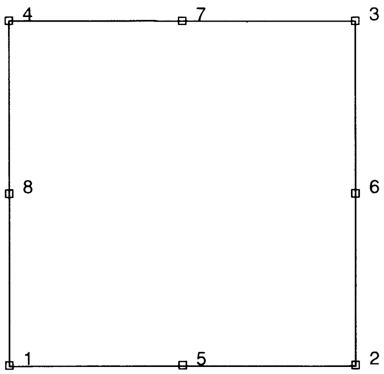


Figure 3-121 Nodal Numbering of Element 73

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Three at each node:

1 = u = displacement in axial (z) direction
2 = v = displacement in radial (r) direction
3 = ϕ = circumferential displacement (θ) direction

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.

Load Type (IBODY)	Description
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Body forces (per unit volume). Load type 2 is uniform body force in the z-direction (axial); load type 3 is nonuniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial); load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, user subroutine FORCEM must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force may be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-122 and **Output Points**) in the following order.

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, direct
4 = γ_{zr}, in-plane shear
5 = $\gamma_{r\theta}$, out-of-plane shear
6 = $\gamma_{\theta z}$, out-of-plane shear

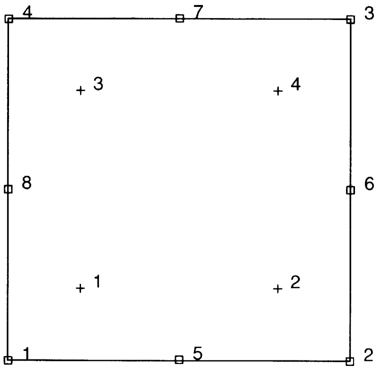


Figure 3-122 Integration Points of Eight-Node 2D Element

Output of Stresses

Same as for **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, the output is given for all four integration points.

■ Element 74

Axisymmetric, Eight-Node Distorted Quadrilateral for Arbitrary Loading, Herrmann Formulation, with Reduced Integration (Fourier)

Element type 74 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element which may exhibit hourglass modes. This element should be used with caution.

This Fourier element may only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 74

Second-order, isoparametric quadrilateral for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique. Mixed formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first in counterclockwise order. Then fifth node between first and second; the sixth node between second and third, etc. See Figure 3-123.

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

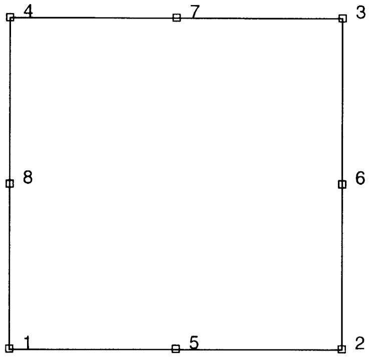


Figure 3-123 Eight-Node Axisymmetric, Herrmann Fourier Element

Degrees of Freedom

1 = u = displacement in axial (z) direction
2 = v = displacement in radial (r) direction
3 = ϕ = circumferential displacement (j) direction.

Additional degree of freedom at each corner node:

4 = σ_{kk}/E = mean pressure variable.

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.

Load Type (IBODY)	Description
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Body forces (per unit volume). Load type 2 is uniform body force in the z-direction (axial); load type 3 is non-uniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial); load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, user subroutine FORCEM must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force may be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-124 and **Output Points**) in the following order:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, direct
4 = γ_{zr}, in-plane shear
5 = $\gamma_{z\theta}$, out-of-plane shear
6 = $\gamma_{\theta z}$, out-of-plane shear
7 = σ_{kk}/E, mean pressure variable (for Herrmann)

Output of Stresses

Same as for **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

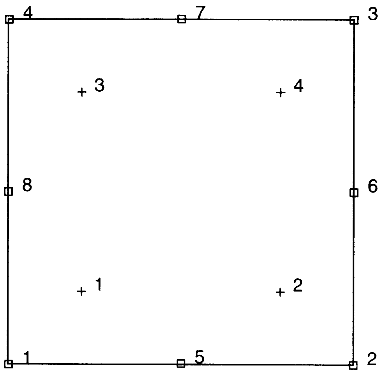


Figure 3-124 Integration Points for Reduced Integration Element

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, the output is given for all four integration points.

Element 75

Bilinear Thick-Shell Element

This is a four-node, thick-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. In this way, a very efficient and simple element is obtained which exhibits correct behavior in the limiting case of thin shells. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion, particularly if the corner nodes lie in the same plane. All constitutive relations may be used with this element.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface will form a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, V_1, V_2, and V_3, which for the centroid are defined in the following way (see Figure 3-125).

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s/\sqrt{2}|s| \quad \bar{d} = d/\sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}$$

$$V_2 = \bar{s} - \bar{d}$$

and

$$V_3 = V_1 \times V_2$$

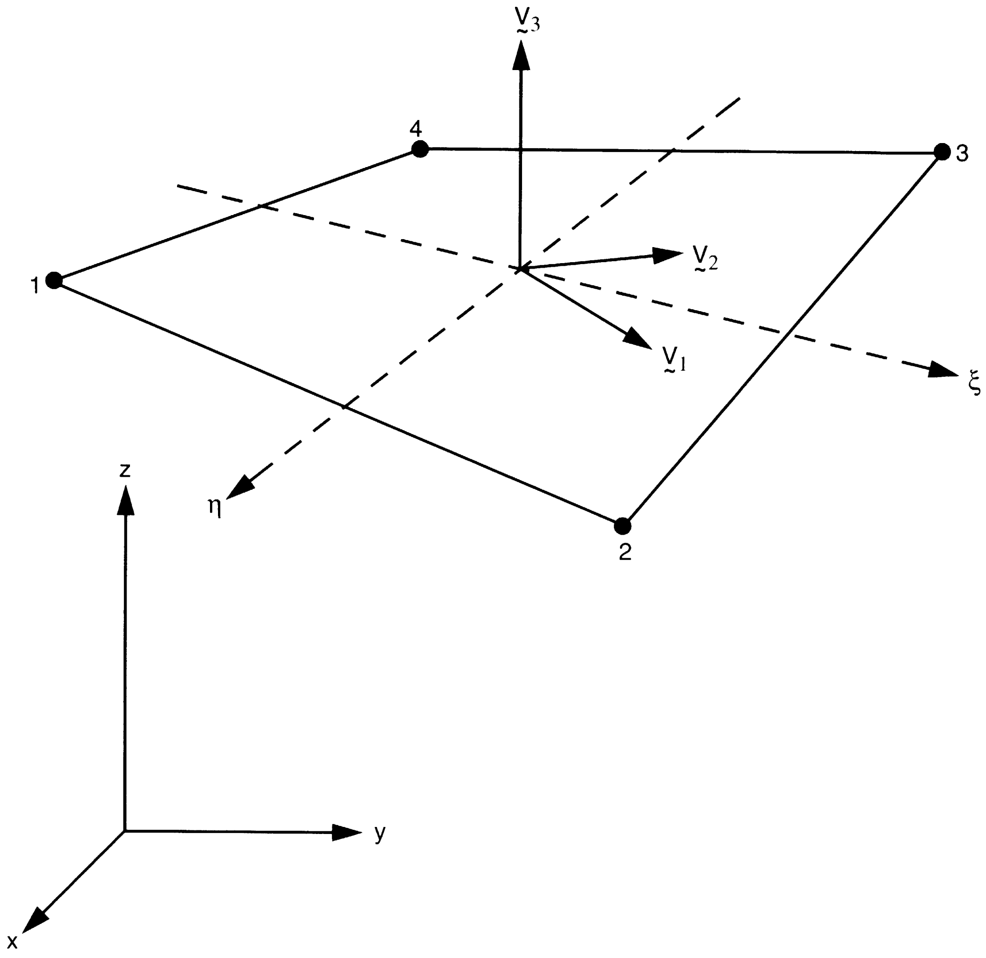


Figure 3-125 Form of Element 75

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

Quick Reference

Type 75

Bilinear, four-node shell element including transverse shear effects.

Connectivity

Four nodes per element. The element may be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_y = rotation about global x-axis
5 = ϕ_z = rotation about global z-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction, magnitude given in user subroutine FORCEM.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude given in user subroutine FORCEM.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in user subroutine FORCEM.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in FORCEM on 1-2 edge.
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in FORCEM on 2-3 edge.
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 3-4 edge.

Load Type	Description
33	Nonuniform edge load magnitude and direction given in FORCEM on 3-4 edge.
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in FORCEM on 4-1 edge.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches: $\varepsilon_{11}\ \varepsilon_{22}\ \varepsilon_{12}$
Middle surface curvatures: $\kappa_{11}\ \kappa_{22}\ \kappa_{12}$
Transverse shear strains: $\gamma_{23}\ \gamma_{31}$

in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system.

Output Of Stresses

$\sigma_{11}, \sigma_{22}, \sigma_{12}, \sigma_{23}, \sigma_{31}$ in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at corner nodes may be transformed to local direction.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, the user has to select his load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.q., thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

■ Element 76

Thin-Walled Beam in Three Dimensions without Warping

This is a simple straight beam element with no warping of the section, but including twist. The default cross section is a thin-walled circular closed section beam; the user may specify alternative closed cross sections through the BEAM SECT parameter.

The degrees of freedom associated with the end nodes are three global displacements and three global rotations, all defined in a right-handed convention. The midnode has only one degree of freedom, rotation along the beam axis, to be compatible with shell element 49 or 72. The generalized strains are stretch, two curvatures, and twist. Stresses are direct (axial) and shear given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at the end nodes, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross-section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. Alternatively, this point may also be specified by the fourth, fifth, and sixth entry on the GEOMETRY data if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

For other than the default (circular) section, the stress points are defined by the user in the local x-y set through the BEAM SECT parameter set. For the circular hollow section, EGEOM1 is the wall thickness and EGEOM2 is the radius. Otherwise, EGEOM2 gives the section choice from the BEAM SECT input. Section properties are obtained by numerical integration over the stress points of the section. The default is a circular cross section.

All constitutive relations may be used with this element.

Notes: For noncircular sections, the BEAM SECT parameter must be used to describe the section.

For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98) the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout may be changed via the PRINT ELEMENT option.

This element may be used in combination with shell element 72 and open-section beam element 77 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 76

Closed-section beam, Euler-Bernoulli theory.

Connectivity

Three nodes per element. (Nodes 1 and 3 are the end nodes. See Figure 3-126)

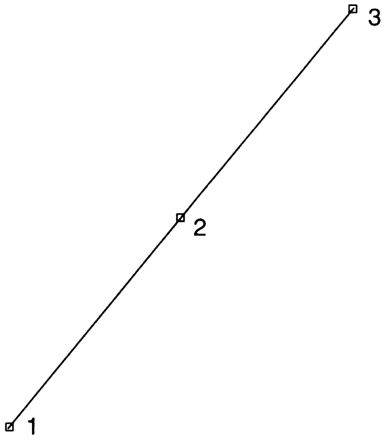


Figure 3-126 Closed-Section Beam

Geometry

In the default section of a hollow circular cylinder, the first data field is for the thickness (EGEOM1).

For noncircular section, set EGEOM1 to 0.

For circular section, set EGEOM2 to radius.

For noncircular section, set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECTION parameter.)

Optional specification of cross-section direction with EGEOM4, EGEOM5 and EGEOM6 (see **Coordinates**).

Coordinates

Six coordinates at the end nodes. The first three coordinates are global (x,y,z). The fourth, fifth, and sixth coordinates are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates may also be given as EGEOM4, EGEOM5 and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

At End Nodes:	At Middle Node:
1 = u	1 = ϕ_t (rotation around the beam axis; positive from the lower to the higher external end node number.)
2 = v	
3 = w	
4 = ϕ_x	
5 = ϕ_y	
6 = ϕ_z	

Distributed Loads

Distributed load types are as follows:

Load type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via user subroutine FORCEM.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may be applied at the end nodes; a twisting moment may be applied at the midnode.

Output of Strains

Generalized strains:

1 = ε_{zz} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = γ = twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-movement
4 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the end nodes may be transformed to a local coordinate system. Transformations should not be invoked for the midnode.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is near the third node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange is available. Finite is not recommended since the cross section is assumed to remain constant.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and/or the radius can be considered as design variables.

Element 77

Thin-Walled Beam in Three Dimensions including Warping

This is a simple straight beam element that includes warping and twist of the section. The user must specify open cross sections through the BEAM SECT parameter. The section number is given in the GEOMETRY data field. Primary warping effects are included, but twisting is assumed to be elastic.

The degrees of freedom associated with the end nodes are three global displacements and three global rotations, all defined in a right-handed convention and the warping. The midnode has only one degree of freedom, rotation along the beam axis, to be compatible with shell element 72. The generalized strains are stretch, two curvatures, warping, and twist. Stresses are direct (axial) given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at the end nodes, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross-section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. Alternatively, this point may also be specified by the fourth, fifth, and sixth entry on the GEOMETRY data if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

All constitutive relations may be used with this element.

Notes: The BEAM SECT parameter must be used to describe the section.

For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout may be changed via the PRINT ELEMENT option.

This element may be used in combination with shell element type 72 and closed-section beam element type 76 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 77

Open section beam including warping.

Connectivity

Three nodes per element (see Figure 3-127).

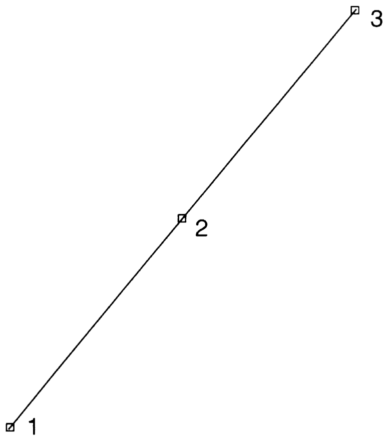


Figure 3-127 Open-Section Beam

Geometry

Set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with EGEOM4, EGEOM5, and EGEOM6 (see **Coordinates**).

Coordinates

Six coordinates at the end nodes; the first three are global (x,y,z), the fourth, fifth and sixth are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates may also be given as EGEOM4, EGEOM5, and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

At End Nodes:	At Middle Node:
1 = u	1 = ϕ_t (rotation around the beam axis; positive from the lower to the higher external end node number.)
2 = v	
3 = w	
4 = ϕ_x	
5 = ϕ_y	
6 = ϕ_z	
7 = η = warping degree-of-freedom	

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via user subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may be applied at the end nodes; a twisting moment may be applied at the midnode.

Output of Strains

Generalized strains:

1 = ε_{xx} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = η = warping
5 = γ = twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-movement
4 = bimoment
5 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the end nodes may be transformed to a local coordinate reference.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is near the third node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange is available. Finite is not recommended since the cross section is assumed to remain constant.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Element 78

Thin-Walled Beam in Three Dimensions without Warping

This is a simple, straight beam element with no warping of the section, but includes twist. The default cross section is a thin-walled, circular, closed-section beam. The user may specify alternative closed cross sections through the BEAM SECT parameter.

The degrees of freedom associated with each node are three global displacements and three global rotations, all defined in a right-handed convention. The generalized strains are stretch, two curvatures, and twist. Stresses are direct (axial) and shear given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at each node, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. This point may also be specified by the fourth, fifth and sixth entry on the GEOMETRY data if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

For other than the default (circular) section, the stress points are defined by the user in the local x-y set through the BEAM SECT parameter. For the circular hollow section, EGEOM1 is the wall thickness, EGEOM2 is the radius. Otherwise, EGEOM2 gives the section choice from the BEAM SECT input. Section properties are obtained by numerical integration over the stress points of the section. The default is a circular cross section.

All constitutive relations may be used with this element.

Notes: For noncircular sections, the BEAM SECT parameter must be used to describe the section.

For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout may be changed via the PRINT ELEMENT option.

This element may be used in combination with shell element type 75 and open-section beam element type 79 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 78

Closed-section beam, Euler-Bernoulli theory.

Connectivity

Two nodes per element (see Figure 3-128).

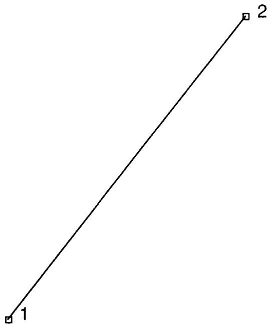


Figure 3-128 Closed Section Beam

Geometry

In the default section of a hollow, circular cylinder, the first data field is for the thickness (EGEOM1).

For noncircular section, set EGEOM1 to 0.

For circular section, set EGEOM2 to radius.

For noncircular section, set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with EGEOM4, EGEOM5, and EGEOM6 (see **Coordinates**).

Coordinates

Six coordinates per node. The first three coordinates are global (x,y,z). The fourth, fifth, and sixth coordinates are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates may also be given as EGEOM4, EGEOM5, and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

1 = u
2 = v
3 = w
4 = ϕ_x
5 = ϕ_y
6 = ϕ_z

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via user subroutine FORCEM.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may be applied at the end nodes.

Output Of Strains

Generalized strains:

1 = ε_{zz} = axial stretch.
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = γ = twist per unit length.

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-movement
4 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the nodes may be transformed to a local coordinate system.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is near the second node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange is available. Finite is not recommended since the cross section is assumed to remain constant.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and/or the radius can be considered as design variables.

Element 79

Thin-Walled Beam in Three Dimensions including Warping

This is a simple, straight beam element that includes warping and twisting of the section. The user must specify open cross sections through the BEAM SECT parameter. The section number is given in the GEOMETRY data field. Primary warping effects are included, but twisting is assumed to be elastic.

The degrees of freedom associated with the nodes are three global displacements and three global rotations, all defined in a right-handed convention and the warping. The generalized strains are stretch, two curvatures, warping, and twist. Stresses are direct (axial) given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at the end nodes, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. Alternatively, this point may also be specified by the fourth, fifth, and sixth entry on the GEOMETRY data if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

All constitutive relations may be used with this element.

Notes: The BEAM SECT parameter must be used to describe the section.

For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout may be changed via the PRINT ELEMENT option.

This element may be used in combination with shell element 75 and closed section beam element 78 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 79

Open-section beam including warping.

Connectivity

Two nodes per element (see Figure 3-129).

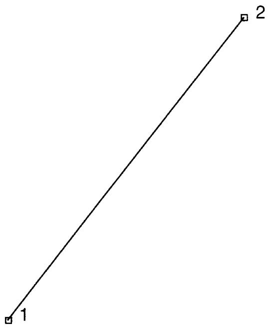


Figure 3-129 Open Section Beam

Geometry

Set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with EGEOM4, EGEOM5, and EGEOM6 (see **Coordinates**).

Coordinates

Six coordinates at each node. The first three coordinates are global (x,y,z). The fourth, fifth, and sixth coordinates are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates may also be given as EGEOM4, EGEOM5, and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

$1 = u$
$2 = v$
$3 = w$
$4 = \phi_x$
$5 = \phi_y$
$6 = \phi_z$
$7 = \eta$ = warping degree-of-freedom

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via user subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may be applied at the end nodes.

Output of Strains

Generalized strains:

1 = ε_{zz} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = η = warping.
5 = γ = twist.

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-movement
4 = bimoment
5 = axial torque

Layer stresses are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the nodes may be transformed to a local coordinate reference.

Tying

Use tying type 100 for fully moment-carrying joints, tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the CONNECTIVITY description of the element. The second point is near the second node in the CONNECTIVITY description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available. Finite strain is not recommended since it is assumed that the cross section does not change.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

■ Element 80

Arbitrary Quadrilateral Plane Strain, Herrmann Formulation

Element type 80 is a four-node, isoparametric, arbitrary quadrilateral written for plane strain incompressible applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

In general, one needs more of these lower-order elements than the higher-order elements such as types 32 or 58. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 11 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 80

Plane strain quadrilateral, Herrmann formulation.

Connectivity

Five nodes per element. Node numbering for the corners follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note: Avoid reducing this element into a triangle.

Geometry

The thickness is entered in the first data field (EGEOM1). Defaults to unit thickness.

Coordinates

Two coordinates in the global x and y directions for the corner nodes. No coordinates are necessary for the fifth hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = u displacement (x-direction)
2 = v displacement (y-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node.

1 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).

Load Type	Description
25	Nonuniform shear force on side 3 - 4.
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates are:

1 $= \varepsilon_{xx}$
2 $= \varepsilon_{yy}$
3 $= \varepsilon_{zz} = 0$
4 $= \gamma_{xy}$
5 $= \sigma_{kk}/E$ = mean pressure variable (for Herrmann)
$= -p$ = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Four stresses corresponding to the first four **Output of Strains**.

Transformation

The two global degrees of freedom at the corner nodes may be transformed into local coordinates.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 3-130.

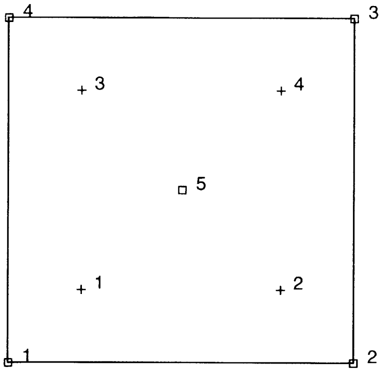


Figure 3-130 Gaussian Integration Points for Element Type 80

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available - finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Element 81

Generalized Plane Strain Quadrilateral, Herrmann Formulation

This element is an extension of the plane strain isoparametric quadrilateral (element 80) to the generalized plane strain case, and modified for the Herrmann variational principle. This is an easy element for use in incompressible analysis. The fifth node, which should not be shared by other elements, represents the hydrostatic pressure. The generalized plane strain condition is obtained by allowing two extra (sixth and seventh) nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions), and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generalized plane strain part of the assembled stiffness matrix; considerable c.p. time savings are achieved if these two nodes are given the highest node numbers in that part of the structure. The generalized plane strain formulation follows that of Volume A.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

In general, one needs more of these lower-order elements than the higher-order elements such as types 34 or 60. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 19 when other material behavior, such as plasticity, must be represented.

Note: Because this element will most likely be used in geometrically nonlinear analysis, it should not be used in conjunction with the CENTROID parameter.

Quick Reference

Type 81

Generalized plane strain quadrilateral, Herrmann formulation.

Connectivity

Seven nodes per element (see Figure 3-131). Node numbering is as follows:

> The first four nodes are the corners of the element in the x-y plane. The numbering must proceed counterclockwise around the element when the x-y plane is viewed from the positive z-side.
>
> The fifth node has only a pressure degree of freedom and is not shared with other elements. The sixth and seventh nodes are shared by all generalized plane strain elements in this part of the structure. These two nodes should have the highest node numbers in the generalized plane strain part of the structure, to reduce matrix solution time.

Note: Avoid reducing this element into a triangle.

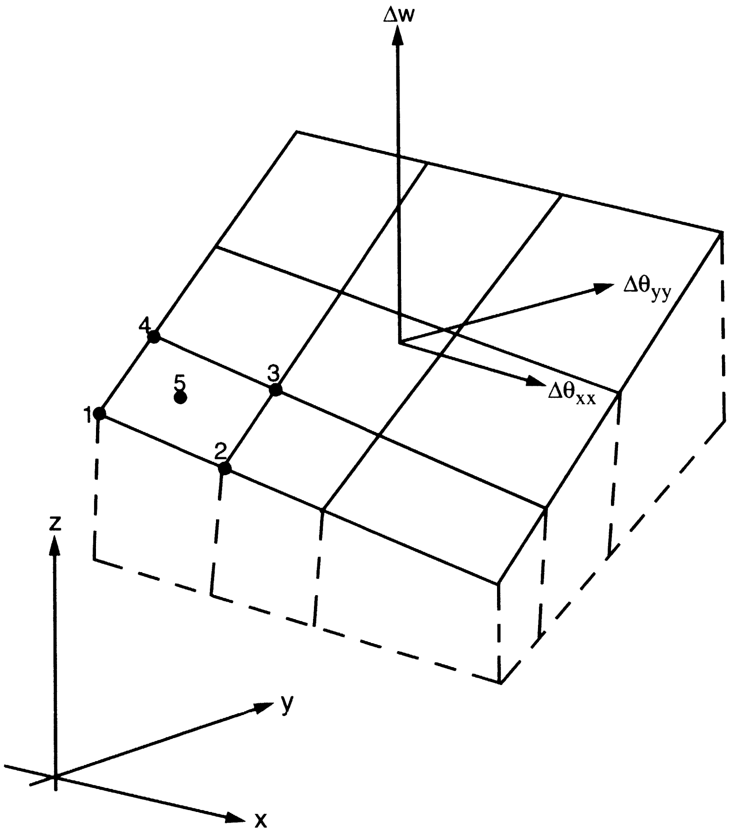


Figure 3-131 Generalized Plane Strain Elements

Geometry

The thickness is entered in the first data field (EGEOM1). Default is unit thickness.

Coordinates

Coordinates are X and Y at all nodes. The coordinates of node 5 do not need to be defined. Note the position of the first shared node (node 6 of each element) determines the point where the thickness change will be measured. The user chooses the location of nodes 6 and 7. These nodes should be at the same location in space.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement (parallel to x-axis)
2 = v displacement (parallel to y-axis)

at the four corner nodes.

For node 5:

1 = negative hydrostatic pressure
= σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

For the first shared node (node 6 of each element):

1 = Δw = thickness change at that point
2 – is not used.

For the second shared node (node 7 of each element):

1 = $\Delta\theta_{xx}$ = relative rotation of top surface of generalized plane strain section of structure, with respect to its bottom surface, about the x-axis.

2 = $\Delta\theta_{yy}$ = relative rotation of the top surface with respect to the bottom surface about the y-axis.

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in first coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in first coordinate direction.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element.
* 8	Uniform pressure on 3-4 face of the element.

Load Type	Description
* 9	Nonuniform pressure on 3-4 face of the element.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element.
* 20	Uniform shear force on 1-2 face in the 1-2 direction.
* 21	Nonuniform shear force on side 1 - 2.
* 22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2 - 3.
* 24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3 - 4
* 26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \sigma_{kk}/E$ = mean pressure variable (for Herrmann)
$= -p$ = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Four stresses, corresponding to the first four **Output of Strains**.

Transformation

In the x-y plane for the corner nodes only.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 82

Arbitrary Quadrilateral Axisymmetric Ring, Herrmann Formulation

Element type 82 is a four-node, isoparametric arbitrary quadrilateral written for axisymmetric incompressible applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element. In general, one needs more of these lower order elements than the higher-order elements such as types 33 or 59. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element may be used in conjunction with other elements such as type 10 when other material behavior, such as plasticity, must be represented.

Note: Because this element will most likely be used in geometrically nonlinear analysis, it should not be used in conjunction with the CENTROID parameter.

Quick Reference

Type 82

Axisymmetric, arbitrary ring with a quadrilateral cross section, Herrmann formulation.

Connectivity

Five nodes per element. Node numbering for the corner nodes follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note: Avoid reducing this element into a triangle.

Coordinates

Two coordinates in the global z- and r-directions for the corner nodes. No coordinates are necessary for the hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = axial displacement (in z-direction)
2 = radial displacement (in r-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node.

1 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4

Load Type	Description
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates is as follows:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{rz}$
$5 = \sigma_{kk}/E$ = mean pressure variable (for Herrmann)
$= -p$ = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Four stresses corresponding to the first four **Output of Strains**.

Transformation

The global degrees of freedom at the corner nodes may be transformed into local coordinates. No transformation for the pressure node.

Tying

May be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid or at the 4 Gaussian points shown in Figure 3-132.

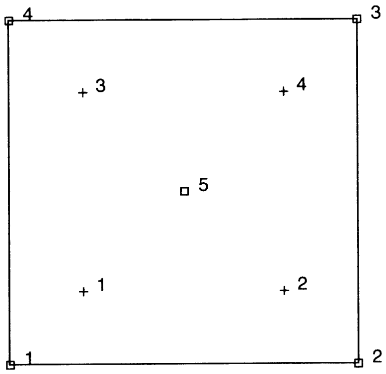


Figure 3-132 Integration Points for Element 82

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

■ Element 83

Axisymmetric Torsional Quadrilateral, Herrmann Formulation

Element type 83 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric incompressible applications with torsional strains. It is assumed that there are no variations in the circumferential direction.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element. In general, one needs more of these lower-order elements than the higher-order elements such as type 66. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 20 when other material behavior, such as plasticity, must be represented.

Notice that there is no friction contribution in the torsional direction when the CONTACT option is used.

Note: Because this element will most likely be used in geometrically nonlinear analysis, it should not be used in conjunction with the CENTROID parameter.

Quick Reference

Type 83

Axisymmetric, arbitrary, ring with a quadrilateral cross section, Herrmann formulation.

Connectivity

Five nodes per element. Node numbering for the corner nodes in a right-handed manner (counterclockwise). The fifth node has only a pressure degree of freedom and is not to be shared with other elements.

Note: Avoid reducing this element into a triangle.

Coordinates

Two coordinates in the global z and r directions respectively. No coordinates are necessary for the hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom for the corner nodes:

1 = axial displacement (in z-direction)
2 = radial displacement (in r-direction)
3 = angular rotation about the symmetry axis (θ-direction, measured in radians)

For node 5:

1 = negative hydrostatic pressure
1 = σ_{kk}/E for Herrmann formulation
= -p for Mooney or Ogden formulation

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1 - 2.

Load Type	Description
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads and torques may be applied at the nodes. The magnitude of concentrated loads must correspond to the load integrated around the circumference.

Output of Strains

Output of strains at the centroid of the element in global coordinates.

1 = ε_{zz}
2 = ε_{rr}
3 = $\varepsilon_{\theta\theta}$
4 = γ_{zr}
5 = $\gamma_{r\theta}$
6 = $\gamma_{\theta z}$
7 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Output for stress is the same direction as for **Output of Strains**.

Transformation

First two degrees of freedom at the corner nodes may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points (see Figure 3-133).

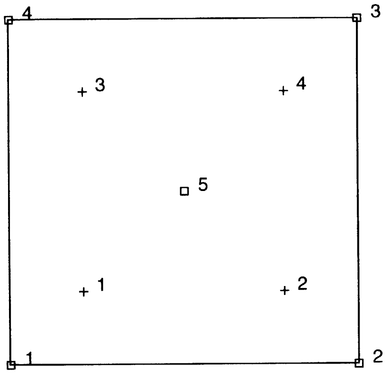


Figure 3-133 Gaussian Integration Points for Element Type 83

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 84

Three-Dimensional Arbitrarily Distorted Brick, Herrmann Formulation

This is an eight-node, isoparametric element with an additional ninth node for the pressure. The element is based on the following type of displacement assumption and mapping into a cube in the (g-h-r) space:

$$x = a_0 + a_1 g + a_2 h + a_3 r + a_4 gh + a_5 hr + a_6 gr + a_7 ghr$$
$$u = b_0 + b_1 g + b_2 h + b_3 r + b_4 gh + b_5 hr + b_6 gr + b_7 ghr$$

The pressure is assumed constant throughout the element. The 24 generalized displacements are related to the u-v-w displacements (in global coordinates) at the eight corners of the distorted cube. The stiffness of the element is formed by numerical integration using eight points defined in the (g-h-r) space.

Quick Reference

Type 84

Nine-node, three-dimensional, first-order isoparametric element (arbitrarily distorted brick) with mixed formulation.

Connectivity

Nine nodes per element (Figure 3-134).

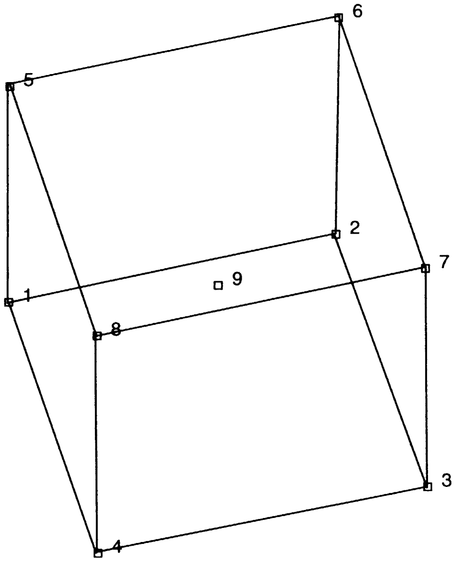


Figure 3-134 Element 84 with 9 Nodes

Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4. The node with the pressure degree of freedom is the last node in the connectivity list, and should not be shared with other elements.

Note: Avoid reducing this element into a tetrahedron or a wedge.

Coordinates

Three coordinates in the global x-, y-, and z-directions for the first eight nodes. No coordinates are necessary for the pressure node.

Degrees of Freedom

Three global degrees of freedom u, v, and w at the first eight nodes. One degree of freedom (negative hydrostatic pressure) at the last node.

1 = σ_{kk}/E = mean pressure variable (for Herrmann)
= $-p$ = negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Distributed loads chosen by value of IBODY as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM).
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in FORCEM.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in FORCEM.
28	Uniform pressure on 3-2-6-7 face.

Load Type	Description
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in FORCEM.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in FORCEM.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in FORCEM.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure forces are positive into element face.

Output of Strains

1 = ε_{xx} = global xx strain
2 = ε_{yy} = global yy strain
3 = ε_{zz} = global zz strain
4 = γ_{xy} = global xy strain
5 = γ_{yz} = global yz strain
6 = γ_{zx} = global zx strain
7 = σ_{kk}/E = mean pressure variable (for Herrmann)
= $-p$ = negative hydrostatic pressure (for Mooney or Ogden)

Output Of Stresses

Six stresses, corresponding to first six **Output of Strains**.

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom at the corner nodes. No transformation for the pressure node.

Tying

No special tying available.

Output Points

Centroid or the eight integration points as shown in Figure 3-135.

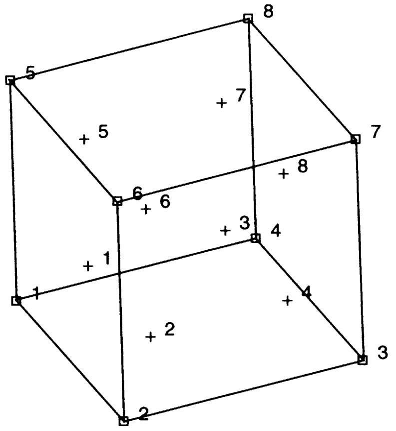


Figure 3-135 Gaussian Integration for Element Type 84

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available. Finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 43. See Element 43 for a description of the conventions used for entering the flux and film data for this element.

Element 85

Four-Node Bilinear Shell (Heat Transfer Element)

This is a four-node heat transfer shell element with temperatures as nodal degrees of freedom. Bilinear interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see parameter HEAT). A four-point Gaussian integration is chosen for the element in the plane of the shell and a eleven-point Simpson's rule is used in the thickness direction. This element is compatible with stress shell element types 72 and 75 in thermal-stress analysis and can be used in conjunction with three-dimensional heat transfer brick elements through tying for heat transfer analysis.

Geometric Basis

Similar to element types 72 and 75, the element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation the surface will form a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

Local orthogonal surface directions, V_1, V_2, and V_3, with the centroid of the element as origin, are defined below (see Figure 3-136):

At the centroid, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = (s/\sqrt{2})\ |s|, \quad \bar{d} = (d/\sqrt{2})\ |d|$$

the local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}$$
$$V_2 = \bar{s} - \bar{d}$$

and

$$V_3 = V_1 \times V_2$$

In this way the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

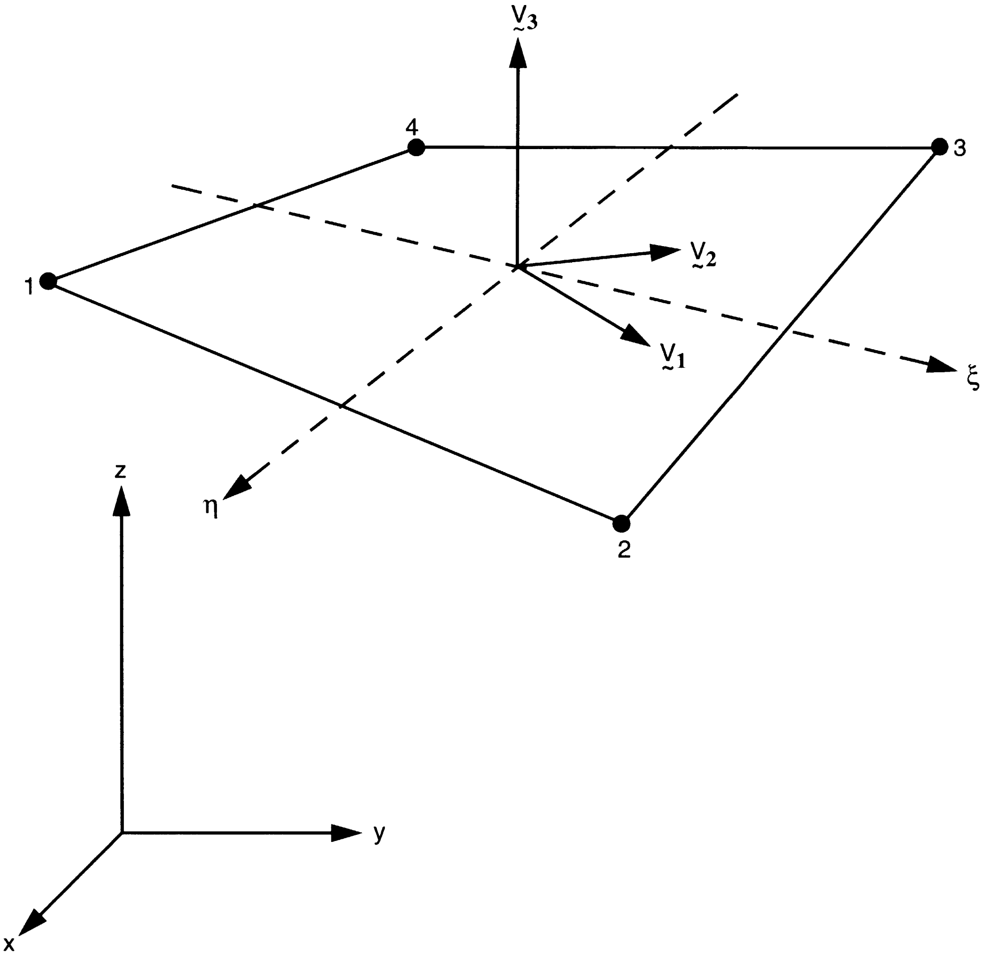


Figure 3-136 Four-Node Heat Transfer Shell Element

The local directions for the Gaussian integrations points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Volume A.

Quick Reference

Type 85

Four-node bilinear, heat transfer shell element.

Connectivity

Four nodes per element. The element may be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

N degrees of freedom per node – temperatures

N = 2 – Linear distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature

N = 3 – Quadratic distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature
3 = Mid Surface Temperature

The first field of the HEAT parameter is used to specify whether a linear or quadratic variation will be used through the thickness.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform FLUX per unit volume on whole element.
3	Nonuniform FLUX per unit volume on whole element; magnitude of flux is defined in subroutine FLUX.

Surface Fluxes

Flux Type	Description
5	Uniform FLUX per unit surface area on top surface.
6	Nonuniform FLUX per unit surface area on top surface; magnitude of flux is in subroutine FLUX.
2	Uniform FLUX per unit surface area on bottom surface.
4	Nonuniform FLUX per unit surface area on bottom surface, magnitude of flux is defined in subroutine FLUX.

Point fluxes may also be applied at nodal degrees-of-freedom.

Films

Same specification as **Fluxes**.

Tying

Standard types 85 and 86 with three-dimensional heat transfer brick elements.

Shell Sect – Integration Through The Shell Thickness Direction

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Three points are enough for linear response. The default is 11 points.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell, at the centroid or Gaussian integration points in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as **Fluxes**.

Element 86

Eight-Node Curved Shell (Heat Transfer Element)

This is a eight-node heat transfer shell element with temperatures as nodal degrees of freedom. Quadratic interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see parameter HEAT). A nine-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. This element is compatible with stress shell element type 22 in thermal-stress analysis and can be used in conjunction with three-dimensional heat transfer brick elements through tying for heat transfer analysis. Note that only centroid or four Gaussian points are used for the output of element temperatures.

Geometric Basis

Similar to element type 22, the element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. Local orthogonal surface directions, V_1, V_2, and V_3, for each integration point, are defined below (see Figure 3-137):

At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} / \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s/(\sqrt{2}|s|) \qquad \bar{d} = d/(\sqrt{2}|d|),$$

the local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}$$

$$V_2 = \bar{s} - \bar{d}$$

and

$$V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

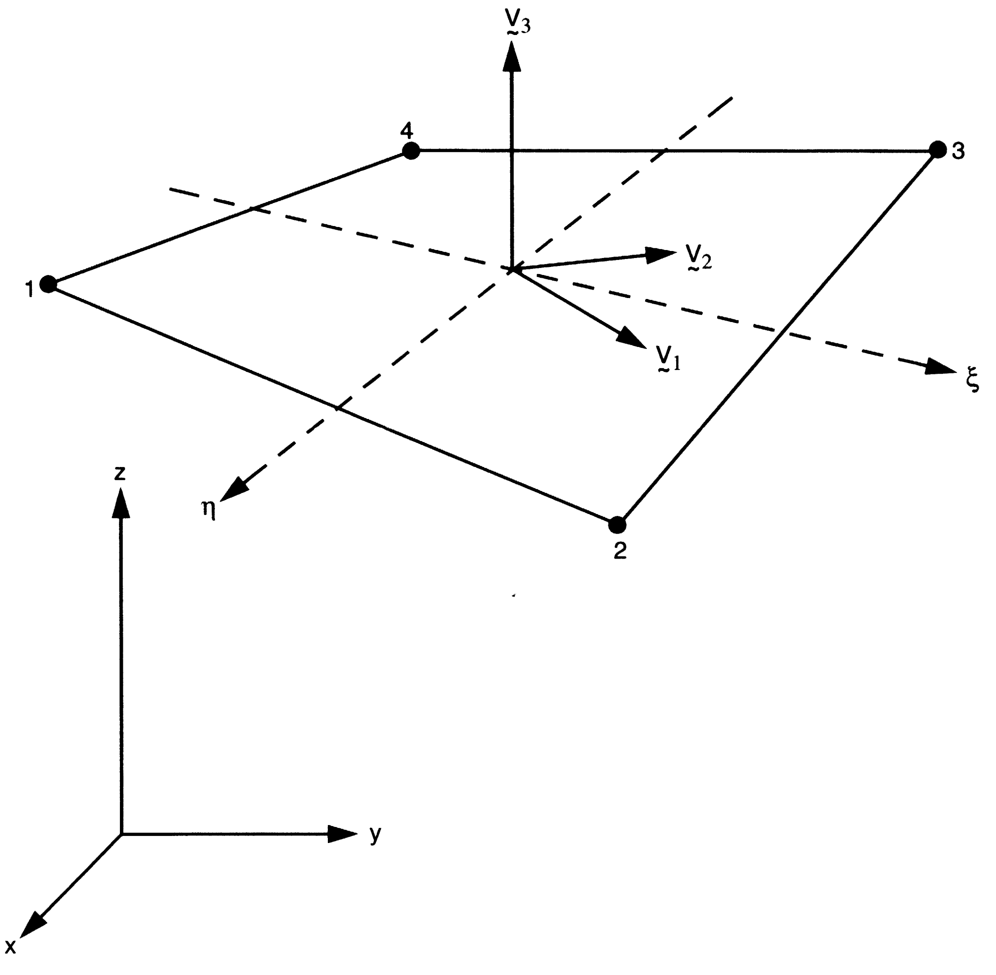


Figure 3-137 Form of Element 86

Quick Reference

Type 86

Eight-node curved heat transfer shell element.

Connectivity

Eight nodes per element.

The connectivity is specified as follows: nodes 1, 2, 3, 4 form the corners of the element, then node 5 lies at the middle of the 1-2 edge; node 6 at the middle of the 2-3 edge, etc.
See Figure 3-137.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

N degrees of freedom per node – temperatures

N = 2 – Linear Distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature

N = 3 – Quadratic distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature
3 = Mid Surface Temperature

The first field of the HEAT parameter is used to specify whether a linear or quadratic variation will be used through the thickness.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform FLUX per unit volume on whole element.
3	Nonuniform FLUX per unit volume on whole element; magnitude of flux is defined in subroutine FLUX.

Surface Fluxes

Flux Type	Description
5	Uniform FLUX per unit surface area on top surface.
6	Nonuniform FLUX per unit surface area on top surface, magnitude of flux is in subroutine FLUX.
2	Uniform FLUX per unit surface area on bottom surface.
4	Nonuniform FLUX per unit surface area on bottom surface, magnitude of flux is defined in subroutine FLUX.

Point fluxes may also be applied at nodal degrees-of-freedom.

Films

Same specification as **Fluxes**.

Tying

Standard types 85 and 86 with three-dimensional heat transfer brick elements.

Shell Sect – Integration Through The Shell Thickness Direction

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Three points are enough for linear response. The default is 11 points.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell at the centroid or Gaussian integration points in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as **Fluxes**.

Element 87

Three-Node Axisymmetric Shell (Heat Transfer Element)

This is a three-node, axisymmetric shell, heat transfer element with temperatures as nodal degrees of freedom. Quadratic interpolation is used for the temperatures in the plane of the shell, and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see parameter HEAT). A three-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. This element is compatible with stress shell element type 89 in thermal-stress analysis and can be used in conjunction with axisymmetric heat transfer elements through tying for heat transfer analysis. Note that only the centroid or two Gaussian points are used for the output of element temperatures.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Volume A.

Quick Reference

Type 87

Three-node axisymmetric, curved heat transfer-shell element.

Connectivity

Three nodes per element (see Figure 3-138).

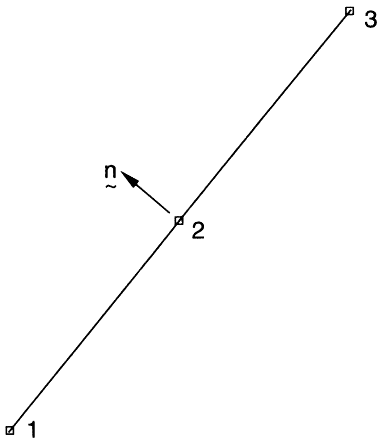


Figure 3-138 Three-Node Axisymmetric, Heat Transfer-Shell Element

Geometry

Linear thickness variation is allowed along the length of the element. For each element, thickness at first node of the element is stored in the first data field (EGEOM1); thickness at third node is stored in the third data field (EGEOM3). If EGEOM3=0, a constant thickness is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

1 = z
2 = r

Degrees of Freedom

N degrees of freedom per node – temperatures

N = 2 – Linear distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature

N = 3 - Quadratic distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature
3 = Mid Surface Temperature

The first field of the HEAT parameter is used to specify whether a linear or quadratic variation will be used through the thickness.

Fluxes

Two types of distributed fluxes are as follows:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in subroutine FLUX.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface area on bottom surface.
4	Nonuniform flux per unit surface area on bottom surface; magnitude of flux is defined in subroutine FLUX.
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface; magnitude of flux is defined in subroutine FLUX.

Point fluxes may also be applied at the degrees of freedom and must be integrated around the circumference.

Films

Same specification as **Fluxes**.

Tying

Standard types 85 and 86 with axisymmetric heat transfer elements.

Shell Sect – Integration Through The Shell Thickness Direction

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers. This number must be odd. Three points are enough for linear response. The default is 11 points.

Output Points

Temperatures are printed out at the layer points through the thickness of the shell, at the centroid or three Gaussian integration points. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as **Fluxes**.

Element 88

Two-Node Axisymmetric Shell (Heat Transfer Element)

This is a two-node, axisymmetric shell, heat transfer element with temperatures as nodal degrees of freedom. Linear interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see parameter HEAT). A three-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. This element is compatible with stress shell element type 1 in thermal-stress analysis and can be used in conjunction with axisymmetric heat transfer elements through tying for heat transfer analysis. Note that only the centroid is used for the output of element temperatures.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Volume A.

Quick Reference

Type 88

Two-node axisymmetric, heat transfer shell element.

Connectivity

Two nodes per element.

Geometry

Constant thickness of the shell is stored in the first data field (EGEOM1).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Two coordinates are required, z and r, as shown in Figure 3-139.

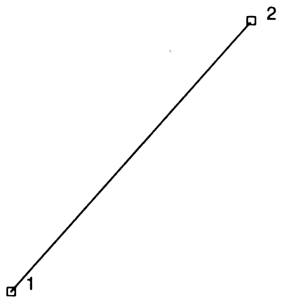


Figure 3-139 Two-Node Axisymmetric Heat Transfer Shell Element

Degrees of Freedom

N degrees of freedom per node - temperatures

N = 2 – Linear distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature

N = 3 – Quadratic distribution through thickness

1 = Top Surface Temperature
2 = Bottom Surface Temperature
3 = Mid Surface Temperature

The first field of the HEAT parameter is used to specify whether a linear or quadratic variation will be used through the thickness.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in subroutine FLUX.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface area on bottom surface.
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface; magnitude of flux is defined in subroutine FLUX.
4	Nonuniform flux per unit surface area on bottom surface; magnitude of flux is defined in subroutine FLUX.

Point fluxes may also be applied at the degrees of freedom and must be integrated around the circumference.

Films

Same specification as **Fluxes**.

Tying

Standard types 85 and 86 with axisymmetric heat transfer elements.

Shell Sect – Integration Through The Shell Thickness Direction

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points. This number must be odd. Three points are enough for linear response. The default is 11 points.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell at the centroid in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as **Fluxes**.

Element 89

Thick Curved Axisymmetric Shell

This is a three-node, axisymmetric, thick-shell element, with a quadratic displacement assumption based on the global displacements and rotation. The strain-displacement relationships used are suitable for large displacements with small strains. Two-point Gaussian integration is used along the element for the stiffness calculation and three-point integration is used for the mass and pressure determination. All constitutive relations may be used with this element.

Quick Reference

Type 89

Axisymmetric, curved, thick-shell element.

Connectivity

Three nodes per element (see Figure 3-140).

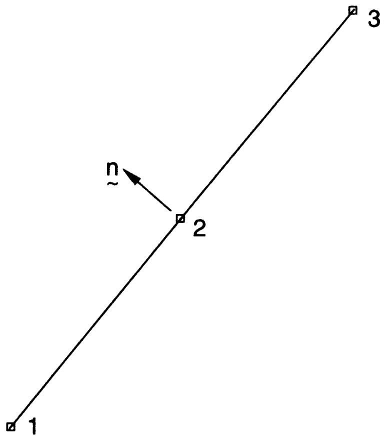


Figure 3-140 Axisymmetric, Curved Thick-Shell Element

Geometry

Linear thickness variation along length of the element. Thickness at first node of the element store in the first data field (EGEOM1).

Thickness at third node in the third data field (EGEOM3).

If EGEOM3=0, constant thickness assumed. Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used since, in the other case, section properties formed at the centroid of the element are used for all integration points.

The second data field is not used (EGEOM2).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)
3 = ϕ = right hand rotation

Tractions

Distributed loads selected with IBODY are as follows:

Load Type	Description
0	Uniform pressure.
5	Nonuniform pressure.

Pressure assumed positive in the negative normal (-n) direction.

Load Type	Description
1	Uniform load in 1 direction (force per unit area).
2	Uniform load in 2 direction (force per unit area).
3	Nonuniform load in 1 direction (force per unit area).
4	Nonuniform load in 2 direction (force per unit area).

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

In the nonuniform cases (IBODY = 3, 4, or 5), the load magnitude must be supplied by user subroutine FORCEM.

Concentrated loads applied at the nodes must be integrated around the circumference.

Output of Strains

Generalized strains are:

1 = ε_s = meridional membrane (stretch)
2 = $\varepsilon\theta$ = circumferential membrane (stretch)
3 = γ_t = transverse shear strain
4 = χ_s = meridional curvature
5 = χ_θ = circumferential curvature

Output Of Stresses

Stresses are output at the integration points through the thickness of the shell. The first point is on the surface of the positive normal. The positive normal is opposite to the direction of positive pressure as shown in Figure 3-140.

1 = meridional stress
2 = circumferential stress
3 = transverse shear.

Transformation

The degrees of freedom may be transformed to local directions.

Output Points

Centroid or two Gaussian integration points. The first Gaussian integration point is close to the first node as defined in the connectivity data. The second integration point is close to the third node as defined in the connectivity data.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers. This number must be odd. Three points are enough for linear material response. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.g., thermal plasticity). The default is 11 points.

Updated Lagrange Procedure And Finite Strain Plasticity

Capability is available – output of stress and strain in meridional and circumferential direction. Thickness will be updated.

Note: Shell theory only applies if strain variation through the thickness is small.

Note, however, that since the curvature calculation is linearized, the user has to select his load steps such that the rotations increments remain small within a load step.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 87. See Element 87 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 90

Thick Curved Axisymmetric Shell – for Arbitrary Loading (Fourier)

This is a three-node, thick-shell element for the analysis of arbitrary loading of axisymmetric shells. Quadratic interpolation functions are used on the global displacements and rotations. Two-point Gaussian integration is used along the element for the stiffness calculation and three-point integration is used for the mass and pressure determination. This Fourier element may only be used for linear elastic analysis. No contact is permitted with this element.

Quick Reference

Type 90

Second-order isoparametric, curved, thick-shell element for arbitrary loading of axisymmetric shells, formulated by means of the Fourier expansion technique.

Connectivity

Three nodes per element (see Figure 3-141).

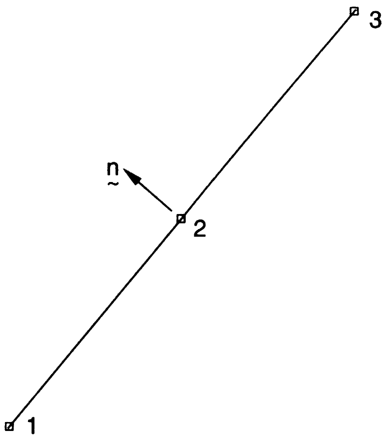


Figure 3-141 Axisymmetric, Curved Thick-Shell Element for Arbitrary Loading

Geometry

Linear thickness variation along length of the element. Thickness at first node of the element is stored in the first data field (EGEOM1).

Thickness at third node in the third data field (EGEOM3).

If EGEOM3=0, constant thickness assumed. Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used since, in the other case, section properties formed at the centroid of the element are used for all integration points.

The second data field is not used (EGEOM2).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)
3 = w = circumferential displacement
4 = ϕ = right handed rotation in the z-r plane
5 = ψ = right handed rotation about meridian

Tractions

Distributed loads selected with IBODY are as follows:

Load Type	Description
0	Uniform pressure
5	Nonuniform pressure

Pressure assumed positive in the negative normal (-n) direction (see Figure 3-141).

Load Type	Description
1	Uniform load in 1 direction (force per unit area).
2	Uniform load in 2 direction (force per unit area).
3	Nonuniform load in 1 direction (force per unit area).

Load Type	Description
4	Nonuniform load in 2 direction (force per unit area).
6	Uniform shear in 3 direction (moment per unit area).
7	Nonuniform shear in 3 direction (moment per unit area).
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

In the nonuniform cases (IBODY = 3, 4, or 5), the load magnitude must be supplied by user subroutine FORCEM.

Concentrated loads applied at the nodes must be integrated around the circumference.

For loads varying in the 3 (circumferential) direction, each distributed load or concentrated force may be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

Output of Strains

Generalized strains are:

$1 = \varepsilon_s$ = meridional membrane (stretch)
$2 = \varepsilon_\theta$ = circumferential membrane (stretch)
$3 = \gamma_{s\theta}$ = in plane shear
$4 = \gamma_{\theta n}$ = transverse shear strain in meridional direction
$5 = \gamma_{ns}$ = transverse shear strain in circumferential direction
$6 = \chi_s$ = meridional curvature
$7 = \chi_\theta$ = circumferential curvature
$8 = \chi_{s\theta}$ = shear curvature (twist)

Output of Stresses

Stresses are output at the integration points through the thickness of the shell. The first point is on the surface of the positive normal. The positive normal is opposite to the direction of positive pressure as shown in Figure 3-141.

1 = meridional stress
2 = circumferential stress
3 = out of plane shear
4 = circumferential transverse shear
5 = meridional transverse shear

Transformation

The degrees of freedom may be transformed to local directions.

Output Points

Centroid or two Gaussian integration points. The first Gaussian integration point is close to the first node as defined in the connectivity data. The second integration point is close to the third node as defined in the connectivity data.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers. This number must be odd. Three points are enough for linear material response. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.g., thermal plasticity). The default is 11 points.

Element 91

Linear Plane Strain Semi-infinite Element

This is a six-node, plane strain element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-5-3 direction. Mappings are such that the element will expand to infinity. Displacements at infinity are implied to be zero; it is unnecessary to put boundary conditions at these nodes. This element does not have nonlinear capability. This element may not be used in CONTACT.

Quick Reference

Type 91

Plane strain semi-infinite element (see Figure 3-142).

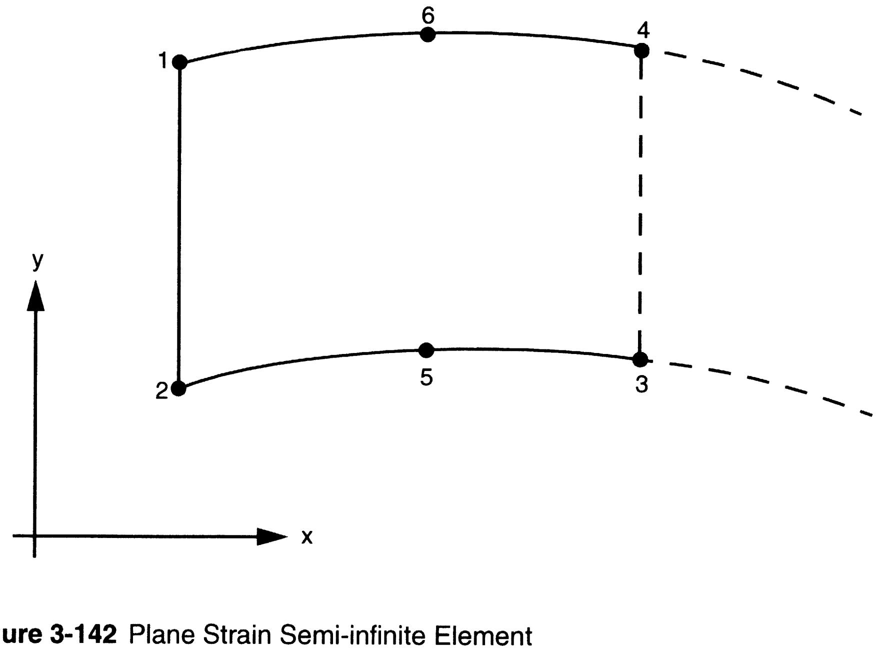


Figure 3-142 Plane Strain Semi-infinite Element

Connectivity

Six nodes per element. Counterclockwise numbering. 1-2 face should be connected to a standard element. 2-3 and 4-1 face should be either connected to another semi-infinite element or no other element. 3-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global x and y direction.

Geometry

The thickness is given in the first field EGEOM1.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Tractions

Distributed loads are listed below.

Load Type	Description
0	Uniform pressure on 1-2 face.
1	Nonuniform pressure on 1-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in y-direction.
6	Uniform shear force in direction 1 to 2 on 1-2 face.
7	Nonuniform shear force in direction 1 to 2 on 1-2 face.
8	Uniform pressure on 2-5-3 face.
9	Nonuniform pressure on 2-5-3 face.
12	Uniform pressure on 3-6-1 face.
13	Nonuniform pressure on 3-6-1 face.

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Stress and Strains

1=xx
2=yy
3=zz
4=xy

Output Points

Centroid or six Gaussian integration points (see Figure 3-143).

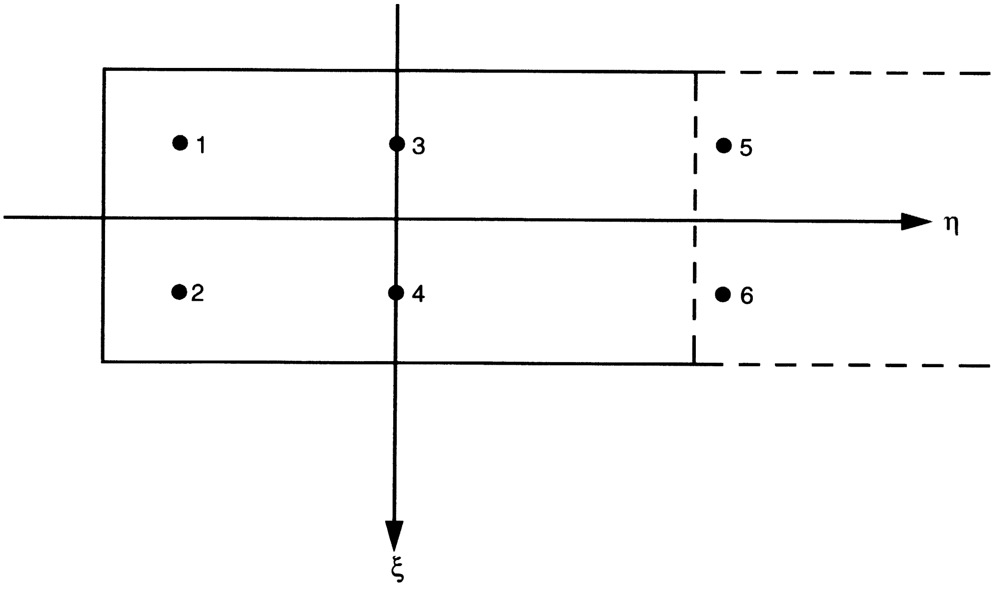


Figure 3-143 Integration Point Locations

Transformations

Two global degrees of freedom may be transformed into a local system.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

Capability is not available.

Note: No boundary conditions at infinity are required. Locations of nodes 5 and 6 express the decay of functions.

Element 92

Linear Axisymmetric Semi-infinite Element

This is a six-node, axisymmetric element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction and cubic in the 2-5-3 direction. Mappings are such that the element will expand to infinity. Displacements at infinity are implied to be zero; it is unnecessary to put boundary conditions at these nodes. This element does not have nonlinear capability. This element may not be used in CONTACT.

Quick Reference

Type 92

Axisymmetric semi-infinite element (see Figure 3-144).

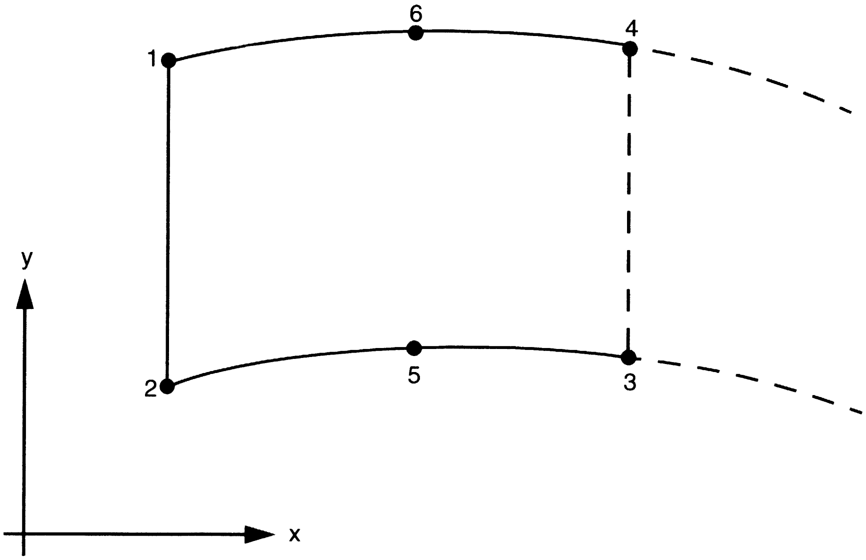


Figure 3-144 Axisymmetric Semi-infinite Element

Connectivity

Six nodes per element. Counterclockwise numbering. 1-2 face should be connected to a standard element. 3-4 face should not be connected to any elements.

Coordinates

Two coordinates in the global z and r directions.

Geometry

Not necessary for this element.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (z-direction)
2 = v displacement (r-direction)

Tractions

Distributed loads are listed below:

Load Type	Description
0	Uniform pressure on 1-2 face.
1	Nonuniform pressure on 1-2 face.
2	Uniform body force in z-direction.
3	Nonuniform body force in z-direction.
4	Uniform body force in r-direction.
5	Nonuniform body force in r-direction.
6	Uniform shear force in direction 1 to 2 on 1-2 face.
7	Nonuniform shear force in direction 1 to 2 on 1-2 face.
8	Uniform pressure on 2-5-3 face
9	Nonuniform pressure on 2-5-3 face.
12	Uniform pressure on 3-7-1 face.
13	Nonuniform pressure on 3-6-1 face.

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Stress and Strain

1=zz
2=rr
3=θθ
4=zr

Output Points

Centroid or six Gaussian integration points (see Figure 3-145).

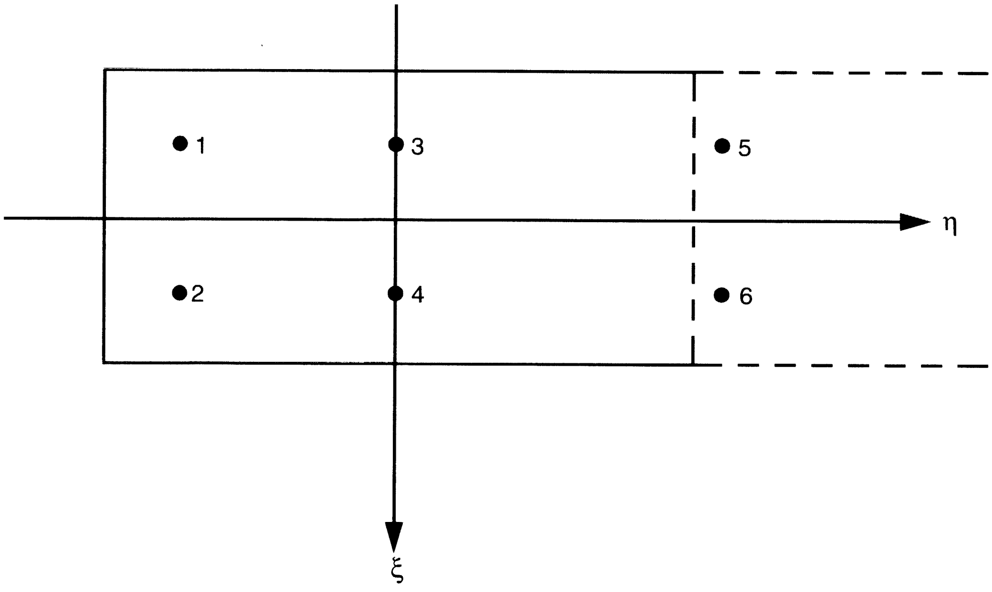


Figure 3-145 Integration Point Location

Transformations

Two global degrees of freedom may be transformed into a local system.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

Capability is not available.

Note: No boundary conditions at infinity are required. Locations of nodes 5 and 6 express the decay of functions.

Element 93

Quadratic Plane Strain Semi-infinite Element

This is a nine-node, plane strain, semi-infinite element that may be used with usual quadratic elements to solve the problems involving unbounded domains. Interpolation functions are parabolic in 1-5-2 direction, and cubic in 2-6-3 direction. Mappings are such that the element will expand to infinity. Displacements at the infinity are implied to be zero. This element does not have any nonlinear capability. This element cannot be used with CONTACT.

Quick Reference

Type 93

Plane strain, semi-infinite element (see Figure 3-146).

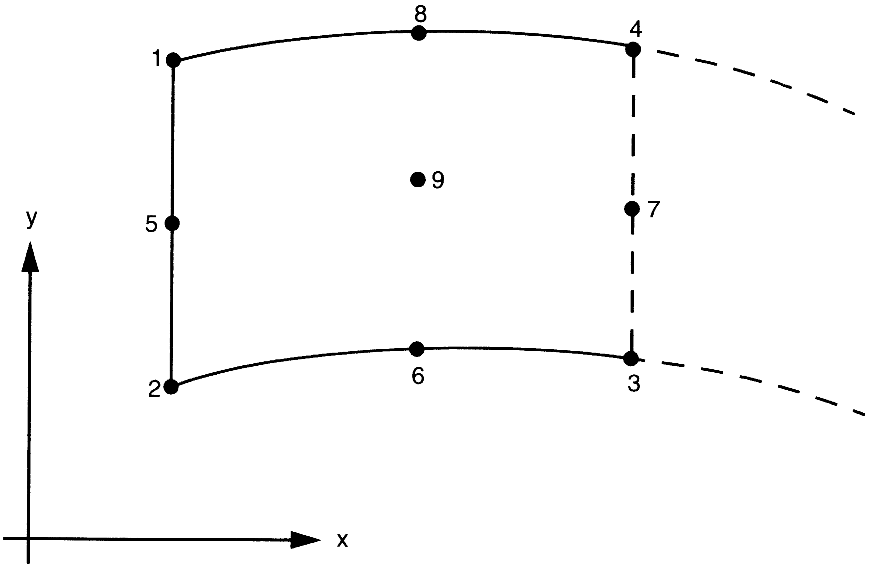


Figure 3-146 Plane Strain Semi-infinite Element

Connectivity

Nine nodes per element. Counterclockwise numbering. 1-5-2 face should be connected to a standard element. 3-7-4 face should not be connected to any elements.

Coordinates

Two coordinates in the global x and y directions.

Geometry

The thickness is given in the first field, EGEOM1.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Tractions

Distributed loads are listed below:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in y-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in y-direction.
6	Uniform shear force in direction 1-5-2 on 1-5-2 face.
7	Nonuniform shear force in direction 1-5-2 on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
12	Uniform pressure on 4-7-1 face.
13	Nonuniform pressure on 4-7-1 face.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Stresses and Strains

1 = xx
2 = yy
3 = zz
4 = xy

Output Points

Centroid or nine Gaussian integration points (see Figure 3-147).

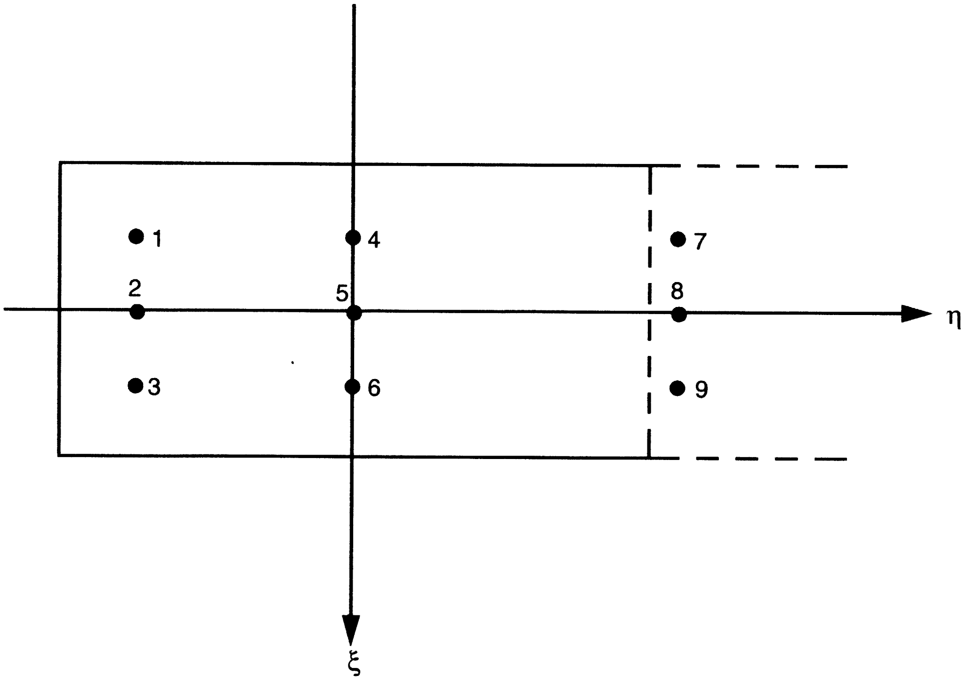


Figure 3-147 Integration Point Locations

Transformations

Two global degrees of freedom may be transformed into a local system.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

Capability is not available.

Note: No boundary conditions at infinity are required. Locations of nodes 7, 8, and 9 express the decay of functions.

Element 94

Quadratic Axisymmetric Semi-infinite Element

This is a nine-node, axisymmetric, semi-infinite element that may be used with the usual quadratic elements to solve the problems involving unbounded domains. Interpolation functions are parabolic in 1-5-2 direction, and cubic in 2-6-3 direction. Mappings are such that the element will expand to infinity. Displacements at the infinity are implied to be zero. This element does not have any nonlinear capability. This element may not be used with CONTACT.

Quick Reference

Type 94

Axisymmetric, semi-infinite element.

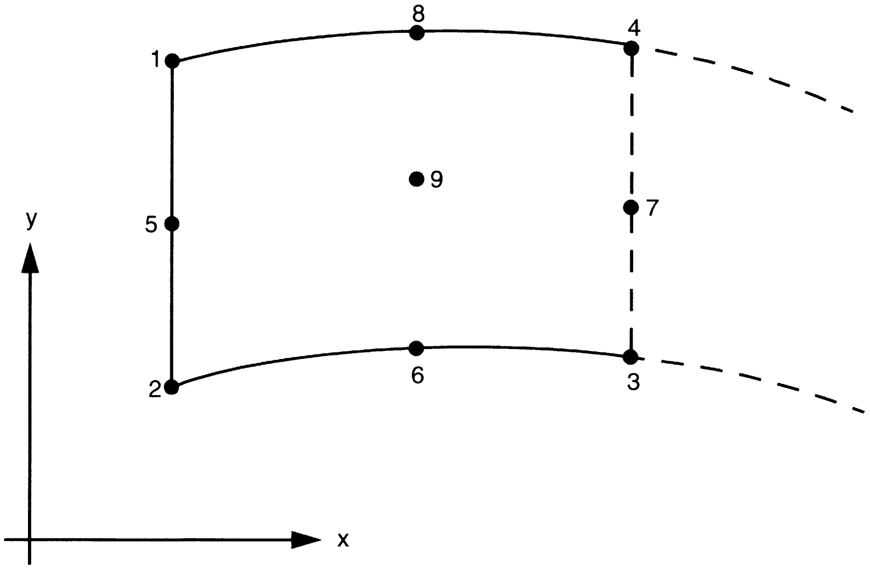


Figure 3-148 Axisymmetric Semi-infinite Element

Connectivity

Nine nodes per element. Counterclockwise numbering. 1-5-2 face should be connected to a standard element. 3-7-4 face should not be connected to any elements.

Coordinates

Two coordinates in the global z and r directions.

Geometry

No geometry option is necessary.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (z-direction)
2 = v displacement (r-direction)

Tractions

Distributed loads are listed below:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in z-direction.
3	Nonuniform body force in z-direction.
4	Uniform body force in r-direction.
5	Nonuniform body force in r-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
12	Uniform pressure on 4-7-1 face.
13	Nonuniform pressure on 4-7-1 face.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Output of Stresses and Strains

1 = zz
2 = rr
3 = θθ
4 = zr

Output Points

Centroid or nine Gaussian integration points.

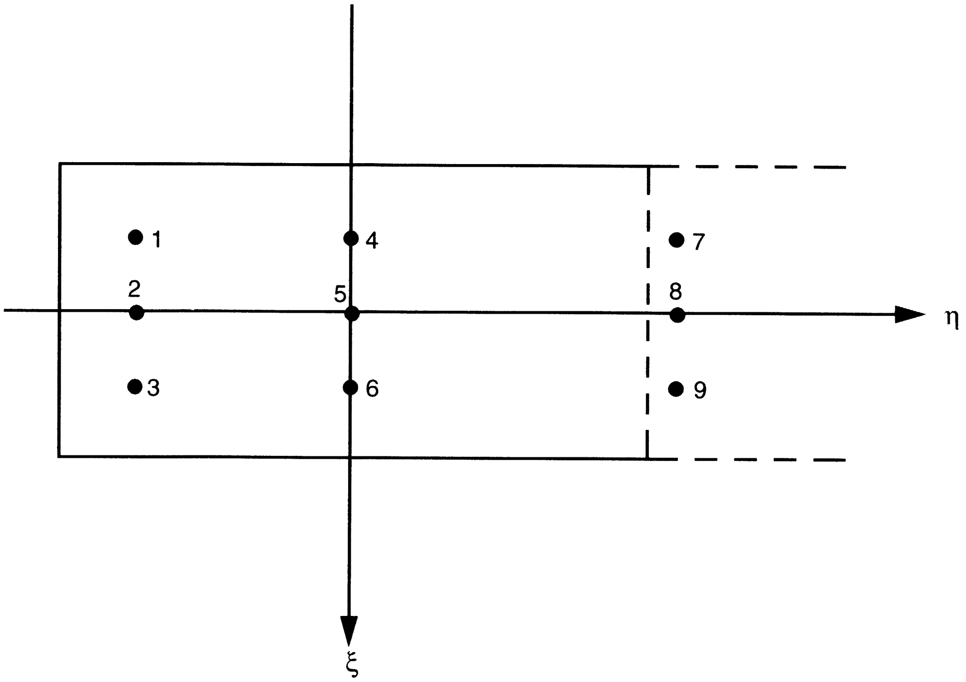


Figure 3-149 Integration Point Locations

Transformations

Two global degrees of freedom may be transformed into a local system.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

Capability is not available.

Note: No boundary conditions at infinity are required. Locations of nodes 7, 8, and 9 express the decay of functions.

Element 95

Axisymmetric Quadrilateral with Bending

This is the same formulation as element type 10, with bending effects included. Element type 95 provides a capability to do efficient analysis of axisymmetric structures deforming axisymmetrically and in bending. The elements are based on the usual (isoparametric) displacement formulation in the z-r plane, whereas in the circumferential direction sinusoidal variation is assumed, which may be expressed by:

$$
\begin{aligned}
u_z(\theta) &= u_z(1+\cos\theta)/2 + \bar{u}_z(1-\cos\theta)/2 \\
u_r(\theta) &= u_r(1+\cos\theta)/2 + \bar{u}_r(1-\cos\theta)/2 \\
u_\theta(\theta) &= u_\theta \sin\theta
\end{aligned}
$$

The element is integrated numerically in the z-r plane using the usual Gaussian quadrature formulas, whereas numerical integration with an equidistant scheme is used along the circumference. The number of points along the circumference is chosen with the SHELL SECT parameter, and must be at least equal to 3. For linear elastic material behavior, this element furnishes "exact" results (for the circumferential variation) for axisymmetric and bending deformation even with the minimum number of circumferential integration points.

Because of the numerical integration scheme, the elements can also be used if material nonlinearity (creep or plasticity) plays a role. It should be noted that the exact solution does not necessarily contain the sinusoidal variation as given above, and, in that sense, the solution obtained is an approximate one. However, experience obtained so far indicates that for thick-walled members, where ovalization of the cross section does not play a significant role, the solution is sufficiently accurate for most practical purposes. Note that if nonlinear effects are present, the number of integration points along the circumference should be at least 5, and more if desired.

This element may not be used with the CONTACT option; use gap element type 97 instead.

Quick Reference

Type 95

Axisymmetric, arbitrary ring with a quadrilateral cross section and bending effects included. This is achieved by including additional degrees of freedom representing the displacements at the point 180° along the circumference.

Connectivity

Four nodes per element. Node numbering follows right-handed convention (counterclockwise). See Figure 3-150.

Geometry

No geometry input is required for this element.

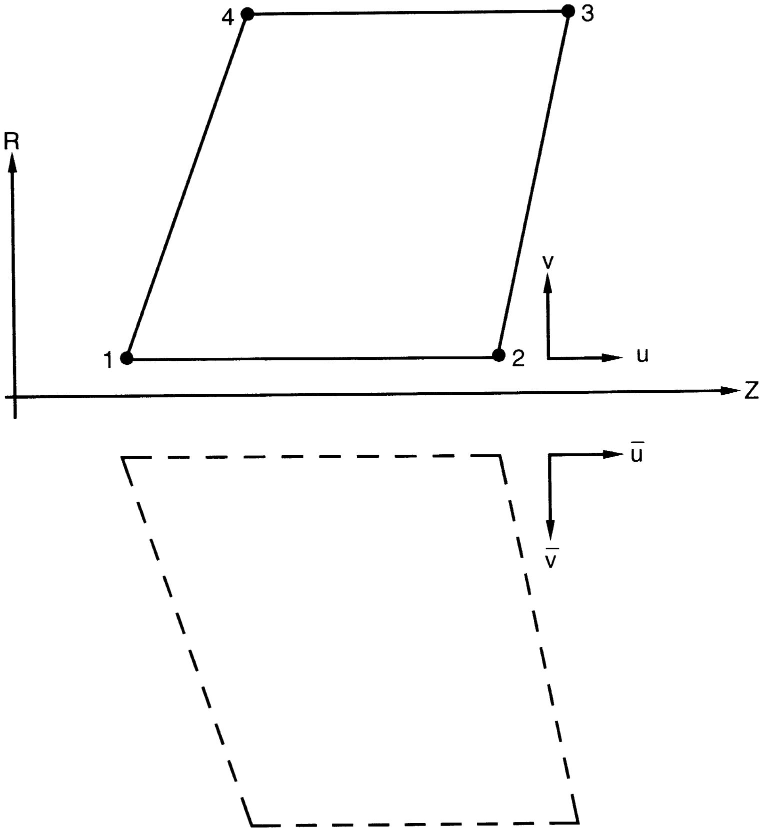


Figure 3-150 Axisymmetric Ring with Bending

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u = z displacement (along symmetry axis).
2 = v = radial displacement.
3 = $\bar{u}$ = z displacement of reverse side.
4 = $\bar{v}$ = radial displacement of reverse side.
5 = w = circumferential displacement at 90° angle.

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0,50	Uniform pressure distributed on 1-2 face of the element.
1,51	Uniform body force per unit volume in first coordinate direction.
2,52	Uniform body force by unit volume in second coordinate direction.
3,53	Nonuniform pressure on 1-2 face of the element.
4,54	Nonuniform body force per unit volume in first coordinate direction.
5,55	Nonuniform body force per unit volume in second coordinate direction.
6,56	Uniform pressure on 2-3 face of the element.
7,57	Nonuniform pressure on 2-3 face of the element.
8,58	Uniform pressure on 3-4 face of the element.
9,59	Nonuniform pressure on 3-4 face of the element.
10,60	Uniform pressure on 4-1 face of the element.
11,61	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1-2.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4.
26	Uniform shear force on side 4-1 (positive from 4 to 1).

Load Type	Description
37	Nonuniform shear force on side 4-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes. The load types 0-11 correspond to axisymmetric loading. The load type 50-61 correspond to "bending" loads. In that case, the circumferential variation of the distributed load is equal to $p(\theta) = p_o \cos(\theta)$.

Output of Strains

Output of strains at the centroid of the element or at the Gauss points in global coordinates is:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{rz}$
$5 = \gamma_{z\theta}$
$6 = \gamma_{\theta r}$

Output of Stresses

Same as for **Output of Strains.**

Transformation

The transformation on degrees of freedom 3 and 4 are the same as on degrees of freedom 1 and 2. Four global degrees of freedom may be transformed into local coordinates.

Tying

Use subroutine UFORMS.

Output Points

Output is available at the centroid or at the 4 Gaussian points shown in Figure 3-151.

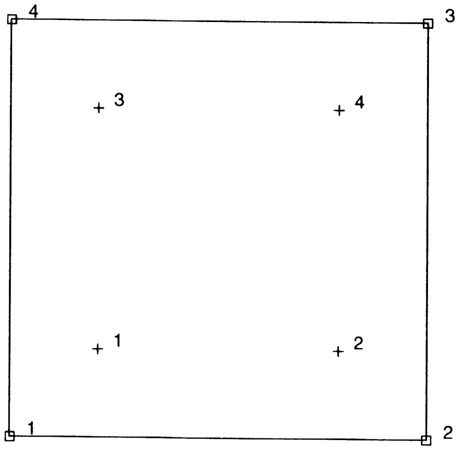


Figure 3-151 Integration Point Locations

Integration Along Circumference

The element is integrated numerically in the circumferential direction. The number of integration points is given on the SHELL SECT parameter. The points are equidistant.

Large Displacement

This element has only geometrically linear behavior. Neither the LARGE DISP or the UPDATE parameter has any effect on this element.

Element 96

Axisymmetric, Eight-Node Distorted Quadrilateral with Bending

This element follows the same second-order isoparametric formulation as the regular axisymmetric element type 28, but has the possibility to deform during bending. Element type 96 provides a capability to do efficient analysis of axisymmetric structures deforming axisymmetrically and in bending. The elements are based on the usual (isoparametric) displacement formulation in the z-r plane, whereas in the circumferential direction sinusoidal variation is assumed, which may be expressed by:

$$
\begin{aligned}
u_z(\theta) &= u_z(1+\cos\theta)/2 + \bar{u}_z(1-\cos\theta)/2 \\
u_r(\theta) &= u_r(1+\cos\theta)/2 + \bar{u}_r(1-\cos\theta)/2 \\
u_\theta(\theta) &= u_\theta \sin\theta
\end{aligned}
$$

The element is integrated numerically in the z-r plane using the usual Gaussian quadrature formulas, whereas numerical integration with an equidistant scheme is used along the circumference. The number of points along the circumference is chosen with the SHELL SECT parameter, and must be at least equal to three. For linear elastic material behavior, this element furnishes "exact" results (for the circumferential variation) for axisymmetric and bending deformation even with the minimum number of circumferential integration points.

Because of the numerical integration scheme, the elements can also be used if material nonlinearity (creep or plasticity) plays a role. It should be noted that the exact solution does not necessarily contain the sinusoidal variation as given above, and in that sense the solution obtained is an approximate one. However, experience obtained so far indicates that for thick-walled members, where ovalization of the cross section does not play a significant role, the solution is sufficiently accurate for most practical purposes Note that if nonlinear effects are present, the number of integration points along the circumference should at least be 5, and more if desired.

This element may not be used with the CONTACT option; use gap element type 97 instead.

Quick Reference

Type 96

Second order, isoparametric, distorted quadrilateral. Axisymmetric formulation with bending deformation.

Connectivity

Eight nodes per element.

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then fifth node between first and second. The sixth node between second and third, etc. See Figure 3-152.

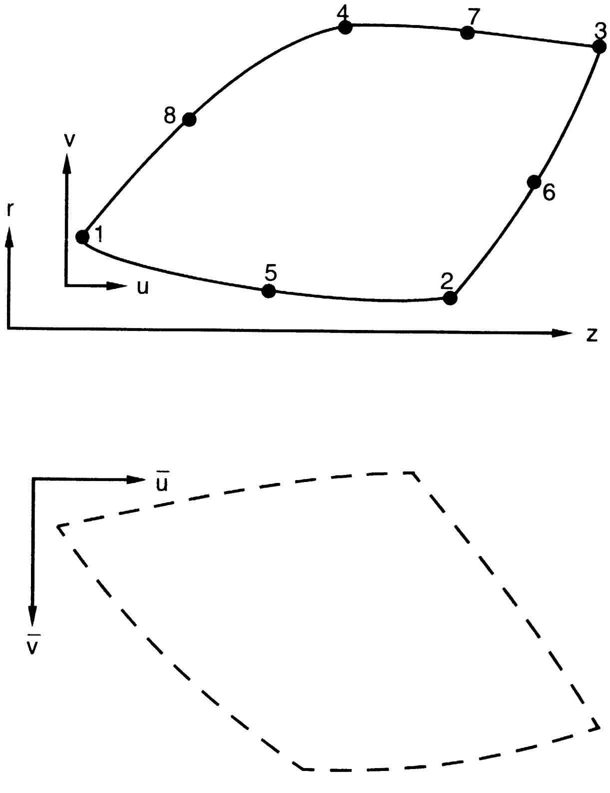


Figure 3-152 Axisymmetric Ring with Bending

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = z-direction displacement (axial)
2 = v = r-direction displacement (radial)
3 = $\bar{u}$ = z-direction displacement at reverse side (axial)
4 = $\bar{v}$ = r-direction displacement at reverse side (radial)
5 = w = circumferential displacement at 90° angle.

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0,50	Uniform pressure on 1-5-2 face.
1,51	Nonuniform pressure on 1-5-2 face.
2,52	Uniform body force in x-direction.
3,53	Nonuniform body force in the x-direction.
4,54	Uniform body force in y-direction.
5,55	Nonuniform body force in the y-direction.
6,56	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7,57	Nonuniform shear in direction 1⇒5⇒2 on 1-5-2 face.
8,58	Uniform pressure on 2-6-3 face.
9,59	Nonuniform pressure on 2-6-3 face.
10,60	Uniform pressure on 3-7-4 face.
11,61	Nonuniform pressure on 3-7-4 face.
12,62	Uniform pressure on 4-8-1 face.
13,63	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.

Load Type	Description
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
90	Torsional load on 1-5-2 face.
91	Nonuniform torsional load on 1-5-2 face.
92	Torsional load on 2-6-3 face.
93	Nonuniform torsional load on 2-6-3 face.
94	Torsional load on 3-7-4 face.
95	Nonuniform torsional load on 3-7-4 face.
96	Torsional load on 4-8-1 face.
97	Nonuniform torsional load on 4-8-1 face.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Concentrated nodal loads must be the value of the load integrated around the circumference.

The load types 0-13 correspond to axisymmetric loading, whereas the load types 50-63 correspond to "bending" loading. In that case, the circumferential variation of the distributed load is equal to $p(\theta) = p_o \cos(\theta)$.

Output of Strains

Six strain components are printed in the order listed below:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direct
4 = γ_{zr}, shear in an axial section
5 = $\gamma_{r\theta}$, shear in a radial section
6 = $\gamma_{\theta z}$, shear in a circumferential section

Output of Stresses

Output for stresses is the same as for **Output of Strains**.

Transformation

Only in z-r plane. The transformation degrees on freedom 3 and 4 is the same as on degrees of freedom 1 and 2.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 3-153.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-153. This is the usual option for a second-order element.

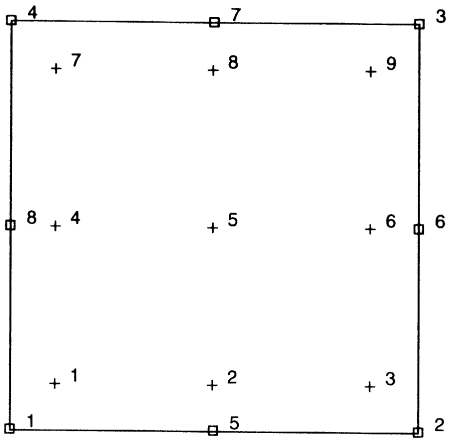


Figure 3-153 Integration Point Locations

Integration Along Circumference

The element is integrated numerically in the circumferential direction. The number of integration points is given on the SHELL SECT parameter. The points are equidistant.

Large Displacement

This element has only geometrically linear behavior. Neither the LARGE DISP or the UPDATE parameter has any effect on this element.

Element 97

Special Gap and Friction Link for Bending

This element provides gap and friction capability for the bending element types 95 and 96. The element works in a similar fashion as the regular fixed direction gap 12; however, extra degrees of freedom have been included to account for independent contact and friction at either side of the bending elements. In contrast to the regular gap element, however, the element cannot be used in full three-dimensional situations, since gap and friction motion can only occur in the 1-2 plane. In all other important aspects, the element is identical to element type 12. Lagrange multipliers are used to enforce the constraint conditions due to contact and friction, and the iterative algorithm is also unchanged. It should be noted that for each element two independent contact and friction conditions can occur, which both must satisfy the convergence criteria.

Quick Reference

Type 97

Four node gap and friction link with double contact and friction conditions. It is designed specifically for use with element types 95 and 96.

Connectivity

Four-nodes per element. Nodes 1 and 4 are the end of the link to connect to the rest of the structure. Node 2 is the "gap" node associated with the contact conditions between the end nodes. Node 3 is the "friction" node associated with the friction conditions between the end nodes.

Coordinates

Nodes 1 and 4 are the physical positions of the end nodes. For use with the axisymmetric bending elements, the first is the z-coordinate and the second the r-coordinate.

Node 2 is the gap direction cosine. Only two values (n_z and n_r) need to be entered because the element is always located in the 1-2 plane. If no values are entered, the program calculates the direction as:

$$\underset{\sim}{n} = (\underset{\sim}{x}_4 - \underset{\sim}{x}_1)/|\underset{\sim}{x}_4 - \underset{\sim}{x}_1|$$

Node 3 is the friction direction cosine. Only two values (t_z and t_r) are used since the element is located in the 1-2 plane. It is not necessary to enter these directions since they are uniquely determined. The direction must be orthogonal to the gap direction. The program calculates these at $t_z = n_r$, $t_r = -n_z$.

Gap Data

First data field:	The closure distance U_{c1}. Note that since the initial geometry is assumed to be symmetric, the same closure distance is used on either side.
Second data field:	The coefficient of friction μ.
Third data field:	This field is used to define the elastic stiffness (spring stiffness) of the closed gap in the gap direction. If the field is left blank, the gap is assumed rigid if closed.
Fourth data field:	This field is used to define the elastic stiffness of the closed, nonslipping gap in the friction direction. If this entry is left blank, the nonslipping closed gap is assumed rigid in the friction direction. This only applies if nonzero coefficient of friction is used.

Degrees Of Freedom

Nodes 1 and 4

$1 = u_z$ = axial displacement associated with the first gap.
$2 = u_r$ = radial displacement associated with the first gap.
$3 = \bar{u}_z$ = axial displacement associated with second gap (opposite side of axisymmetric structure).
$4 = \bar{u}_r$ = radial displacement associated with second gap.

Node 2

$1 = N$ = normal forces in first gap.
$3 = \bar{N}$ = normal forces in second gap.

Note: Degrees of freedom 2 and 4 are not used.

Node 3

$1 = F$ = friction force in first gap.
$2 = s$ = accumulated slip in first gap.
$3 = \bar{F}$ = friction force in second gap.
$4 = \bar{s}$ = accumulated slip in second gap.

It is assumed that the circumferential variation of the contact force and friction force is of the form:

$$n = N(1 + \cos\theta) + \bar{N}(1 - \cos\theta)$$

$$f = F(1 + \cos\theta) + \bar{F}(1 - \cos\theta)$$

Transformations

The transformation option can be used for nodes 1 and/or 4. Note that the second two degrees of freedom ($\bar{u}_z$ and $\bar{u}_r$) are transformed in the same way as the first two degrees of freedom.

Element 98

Elastic Beam with Transverse Shear

This is a straight beam in space which includes transverse shear effects with elastic material response. Large curvature changes are neglected in the large displacement formulation. Linear interpolation is used along the axis of the beam (constant axial force) with cubic displacement normal to the beam axis (constant beam curvature). This element may be used for nonlinear elasticity (hypoelastic) where the material behavior is given in subroutine UBEAM (see Volume D). No other material nonlinearity is associated with this element.

Geometric Basis

The element uses a local (x,y,z) set for section properties. Local x and y are the principal axes of the cross section. Local z is along the beam axis. The element is defined geometrically in the GEOMETRY fields 4, 5, and 6. Using the GEOMETRY option, a vector in the plane of the local x-axis and the beam axis must be specified. If no vector is defined here, the local coordinate system may alternatively be defined by the global (x,y,z) coordinates at the two nodes and by (x_1, x_2, x_3), a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam axis toward the point. The local x-axis is normal to the beam axis.

The local z-axis goes from node 1 to node 2, and the local y-axis forms a right-handed set with local x and z.

Numerical Integration

The element uses a one-point integration scheme. This point is at the midspan location. This leads to an exact calculation for bending and a reduced integration scheme for shear.

Quick Reference

Type 98

Elastic straight beam. Linear interpolation for displacements and rotations. Transverse shear included.

Connectivity

Two nodes. The local z-axis goes from the first node to the second node (see Figure 3-154).

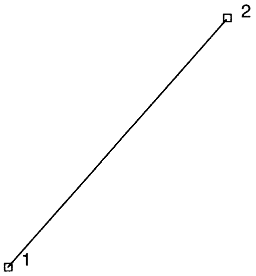


Figure 3-154 Elastic Beam Element

Geometry

First geometry data field	A	area, or enter a zero if beam definition given through the BEAM SECT parameter.
Second geometry data field	I_{xx}	moment of inertia of section about local x-axis, or enter the section number if the BEAM SECT parameter is to be used.
Third geometry data field	I_{yy}	moment of inertia of section about local y-axis

The bending stiffnesses of the section are calculated as EI_{xx} and EI_{yy}.

The torsional stiffness of the section is calculated as $\frac{E}{2(1+\nu)}(I_{xx}+I_{yy})$.

Here, E and υ are Young's modulus and Poisson's ratio calculated as functions of temperature.

If a zero is entered in the first geometry field, the program will use the beam section data corresponding to the section number given in the second geometry field. (Sections are defined using the BEAM SECT parameter set.) This allows specification of the torsional stiffness factor K unequal to $I_{xx} + I_{yy}$, as well as definition of the effective transverse shear areas A_x^s and A_y^s unequal to the area A.

EGEOM4-EGEOM6: Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis will lie on the same side as the specified vector.

Coordinates

First three coordinates – (x, y, z) global Cartesian coordinates.

Fourth, fifth, and sixth coordinates at each node – global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam towards this point. The local x-axis is normal to the beam axis. The fourth, fifth, and sixth coordinates will only be used if the local x-axis direction is not specified in the GEOMETRY block.

Degrees of Freedom

1 = u_x = global Cartesian x-direction displacement
2 = u_y = global Cartesian y-direction displacement
3 = u_z = global Cartesian z-direction displacement
4 = θ_x = rotation about global x-direction
5 = θ_y = rotation about global y-direction
6 = θ_z = rotation about global z-direction

Tractions

The four types of distributed loading are as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via user subroutine FORCEM.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads and moments may be applied at the nodes.

Output Of Strains

Generalized strain components are as follows:

Axial stretch ε
Local γ_{xz} shear
Local γ_{yz} shear
Curvature about local x-axis of cross section κ_{xx}
Curvature about local y-axis of cross section κ_{yy}
Twist about local z-axis of cross section κ_{zz}

Output of Section Forces

Section forces are output as:

Axial force
Local T_x shear force
Local T_y shear force
Bending moment about x-axis of cross section
Bending moment about y-axis of cross section
Torque about beam axis

Transformation

Displacements and rotations may be transformed to local directions.

Tying

For interacting beam, use tying type 100 for fully moment carrying joint. Use tying type 52 for pinned joint.

Output Points

Centroidal section of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available for this element. The finite strain capability does not apply to this element.

Note: Nonlinear elasticity can be implemented with the HYPOELASTIC option and user subroutine UBEAM.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data.

Design Variables

The cross-sectional area (A) and moments of inertia (I_{xx}, I_{yy}) can be considered as design variables.

Element 99

Heat Transfer Link Element Compatible with Beam Elements

Not available at this time.

■ Element 100

Heat Transfer Link Element Compatible with Beam Elements

Not available at this time.

Element 101

Six-Node Plane Semi-infinite Heat Transfer Element

This is a six-node planar semi-infinite heat transfer element that may be used to model an unbounded domain in one direction. Element type 101 is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-5-3 direction. Mappings are such that the element will expand to infinity. Both the conductivity and the capacity matrices are numerically integrated using six (2 x 3) integration points.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Volume A.

Quick Reference

Type 101

Six-node planar semi-infinite heat transfer element.

Connectivity

Six nodes per element (see Figure 3-155).

Counterclockwise numbering. The 1-2 face should be connected to a standard four-node plane heat transfer element and the 2-3 and 4-1 faces should be either connected to another six-node plane semi-infinite heat transfer element or be a free surface. The 3-4 face should not be connected to any other elements.

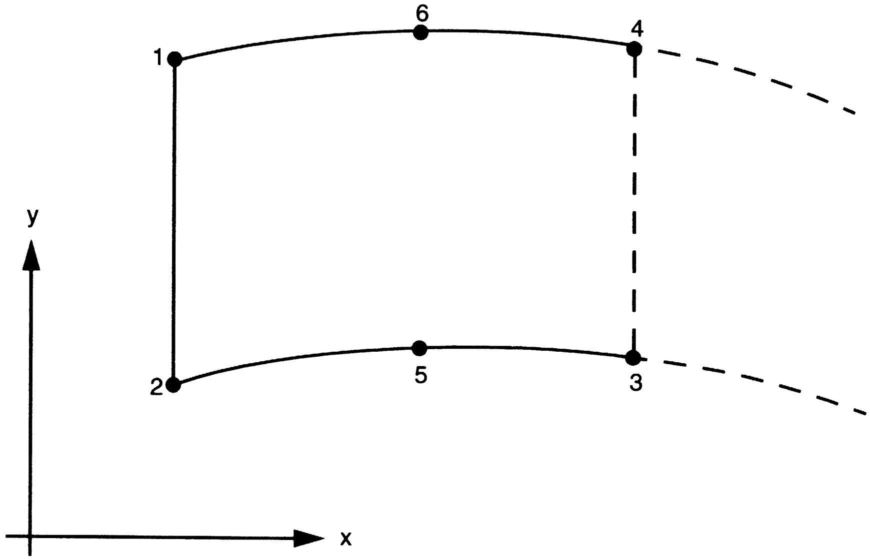


Figure 3-155 Six-Node Planar Semi-infinite Heat Transfer Element

Coordinates

Two coordinates in the global x and y directions.

Geometry

Thickness of the element is given in the first field EGEOM1.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2 face of the element.
1	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in subroutine FLUX.
2	Uniform flux per unit volume on whole element.

Flux Type	Description
3	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 2-5-3 face of the element.
9	Nonuniform flux per unit area on 2-5-3 face of the element; magnitude given in subroutine FLUX.
12	Uniform flux per unit area on 4-6-1 face of the element.
13	Nonuniform flux per unit area on 4-6-1 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Center or six Gaussian integration points (see Figure 3-156).

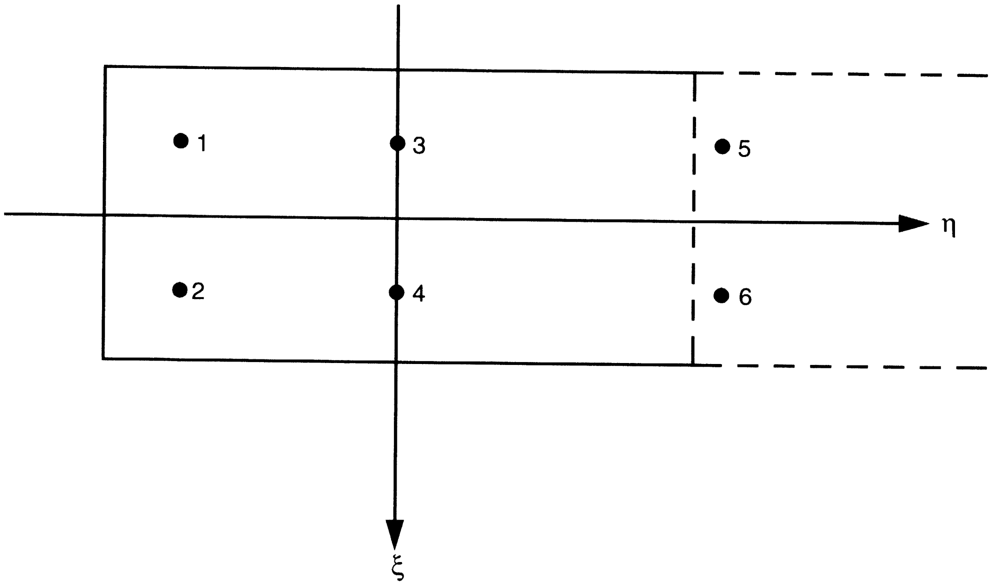


Figure 3-156 Integration Point Locations for Element 101

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 102

Six-Node Axisymmetric Semi-infinite Heat Transfer Element

This is a six-node axisymmetric semi-infinite heat transfer element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-5-3 direction. Mappings are such that the element will expand to infinity. Both the conductivity and the capacity matrices are numerically integrated using six (2 x 3) integration points.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Volume A.

Quick Reference

Type 102

Six-node axisymmetric semi-infinite heat transfer element.

Connectivity

Six nodes per element (see Figure 3-157).

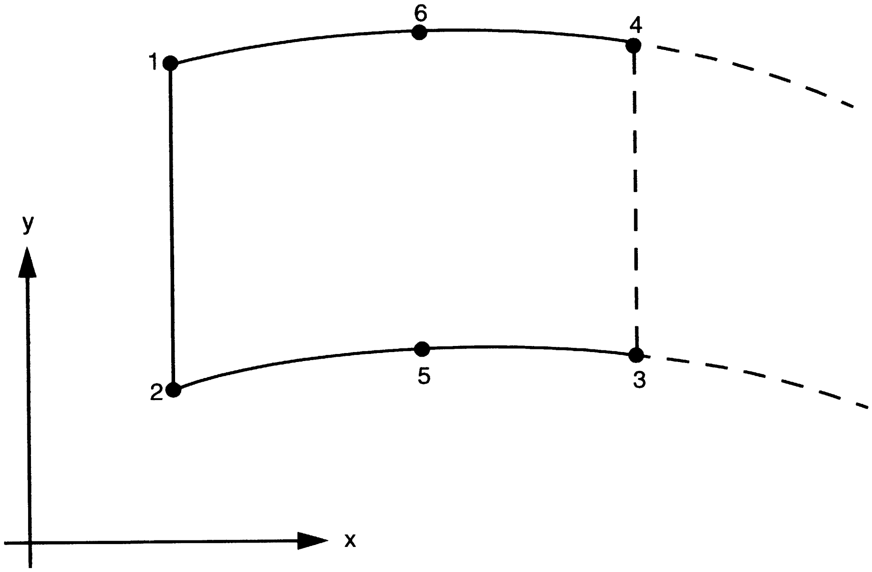


Figure 3-157 Six-Node Axisymmetric Semi-infinite Heat Transfer Element

Counterclockwise numbering. The 1-2 face should be connected to a standard four-node axisymmetric heat transfer element and the 2-3 and 4-1 faces should be either connected to another six-node axisymmetric semi-infinite heat transfer element or be a free surface. The 3-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global z and r directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2 face of the element.
1	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in subroutine FLUX.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 2-5-3 face of the element.
9	Nonuniform flux per unit area on 2-5-3 face of the element; magnitude given in subroutine FLUX.
12	Uniform flux per unit area on 4-6-1 face of the element.
13	Nonuniform flux per unit area on 4-6-1 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Charge

Same specifications as **Fluxes**.

Current

Same specifications as **Fluxes**.

Output Points

Center or six Gaussian integration points (see Figure 3-158).

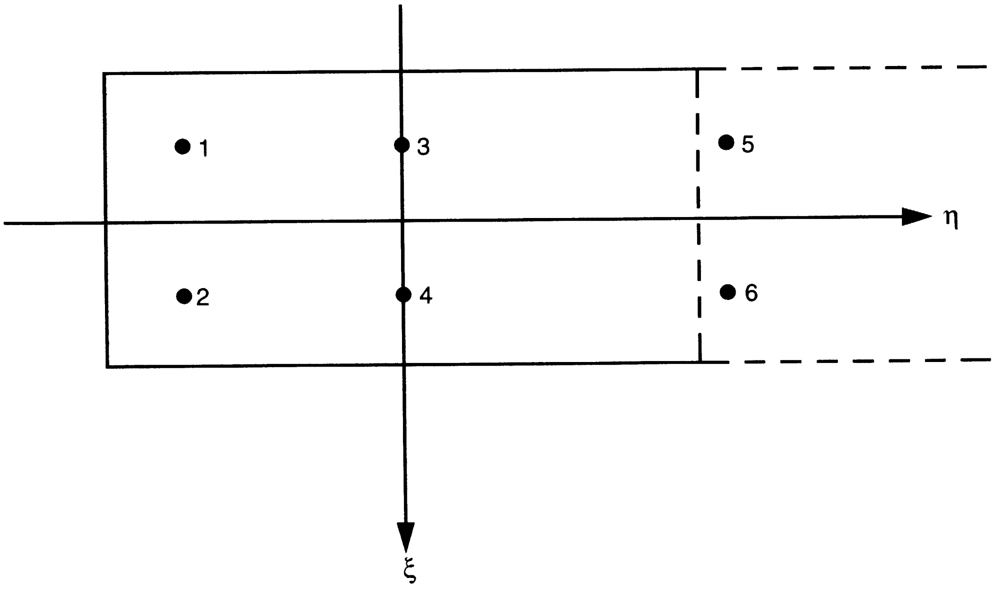


Figure 3-158 Integration Point Locations for Element 102

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 103

Nine-Node Planar Semi-infinite Heat Transfer Element

This is a nine-node planar semi-infinite heat transfer element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are parabolic in the 1-5-2 direction, and cubic in the 2-6-3 direction. Mappings are such that the element will expand to infinity. Both the conductivity and the capacity matrices are numerically integrated using nine (3 x 3) integration points.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Volume A.

Quick Reference

Type 103

Nine-node planar semi-infinite heat transfer element.

Connectivity

Nine nodes per element (see Figure 3-159).

Counterclockwise numbering. The 1-5-2 face should be connected to a standard eight-node planar heat transfer element and the 2-6-3 and 4-8-1 faces should be either connected to another nine-node plane semi-infinite heat transfer element or be a free surface. The 3-7-4 face should not be connected to any other elements.

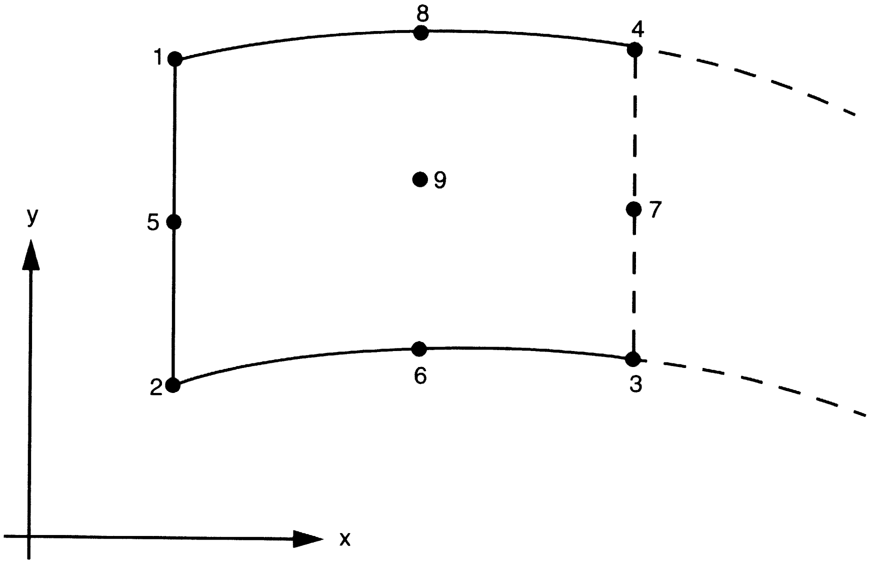


Figure 3-159 Nine-Node Planar Semi-infinite Heat Transfer Element

Coordinates

Two coordinates in the global x and y directions.

Geometry

Thickness of the element is given in the first field EGEOM1.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-5-2 face of the element.
1	Nonuniform flux per unit area on 1-5-2 face of the element; magnitude given in subroutine FLUX.
2	Uniform flux per unit volume on whole element.

Flux Type	Description
3	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 2-6-3 face of the element.
9	Nonuniform flux per unit area on 2-6-3 face of the element; magnitude given in subroutine FLUX.
12	Uniform flux per unit area on 4-8-1 face of the element.
13	Nonuniform flux per unit area on 4-8-1 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes.**

Charge

Same specifications as **Fluxes.**

Current

Same specifications as **Fluxes.**

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Output Points

Center or nine Gaussian integration points (see Figure 3-160).

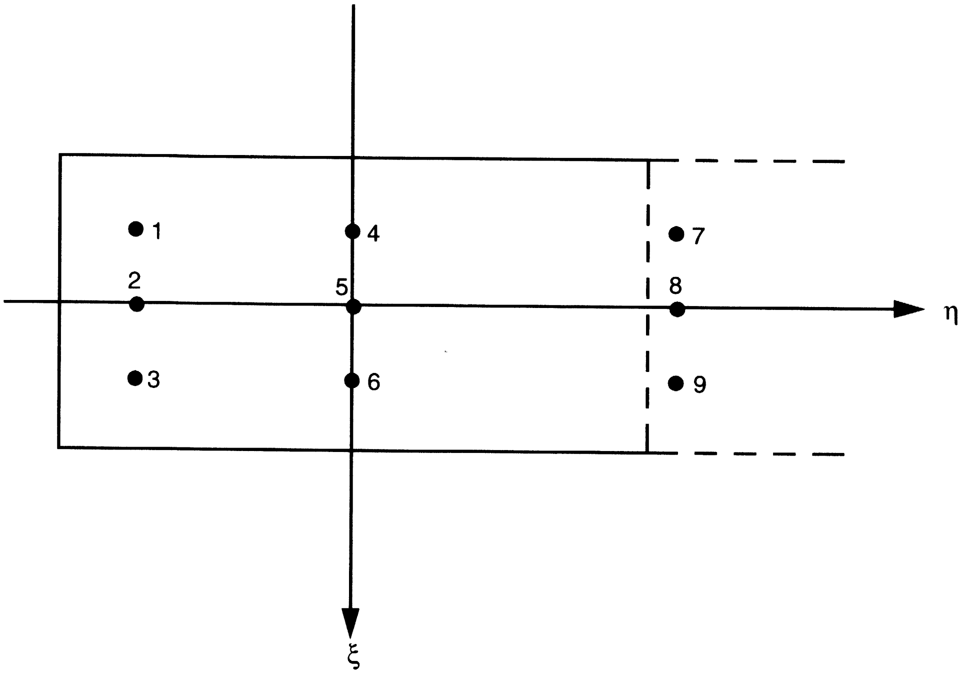


Figure 3-160 Integration Point Locations for Element 103

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 104

Nine-Node Axisymmetric Semi-infinite Heat Transfer Element

This is a nine-node axisymmetric semi-infinite heat transfer element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are parabolic in the 1-5-2 direction, and cubic in the 2-6-3 direction. Mappings are such that the element will expand to infinity. Both the conductivity and the capacity matrices are numerically integrated using nine (3 x 3) integration points.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Volume A.

Quick Reference

Type 104

Nine-node axisymmetric semi-infinite heat transfer element.

Connectivity

Nine nodes per element (see Figure 3-161).

Counterclockwise numbering. The 1-5-2 face should be connected to a standard eight-node axisymmetric heat transfer element and the 2-6-3 and 4-8-1 faces should be either connected to another nine-node axisymmetric semi-infinite heat transfer element or be a free surface. The 3-7-4 face should not be connected to any other elements.

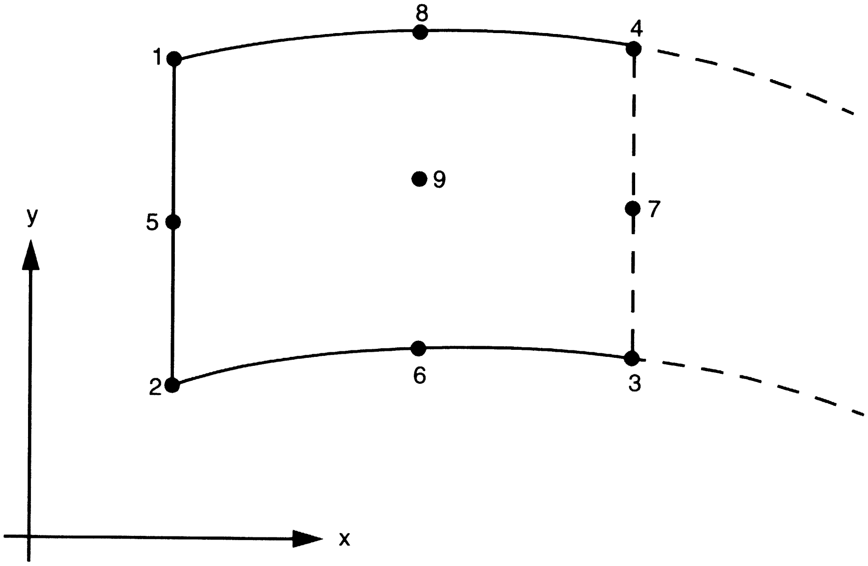


Figure 3-161 Nine-Node Axisymmetric Semi-infinite Heat Transfer Element

Coordinates

Two coordinates in the global z and r directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-5-2 face of the element.
1	Nonuniform flux per unit area on 1-5-2 face of the element; magnitude given in subroutine FLUX.
2	Uniform flux per unit volume on whole element.

Flux Type	Description
3	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 2-6-3 face of the element.
9	Nonuniform flux per unit area on 2-6-3 face of the element; magnitude given in subroutine FLUX.
12	Uniform flux per unit area on 4-8-1 face of the element.
13	Nonuniform flux per unit area on 4-8-1 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Charge

Same specifications as **Fluxes**.

Current

Same specifications as **Fluxes**.

Output Points

Center or nine Gaussian integration points (see Figure 3-162).

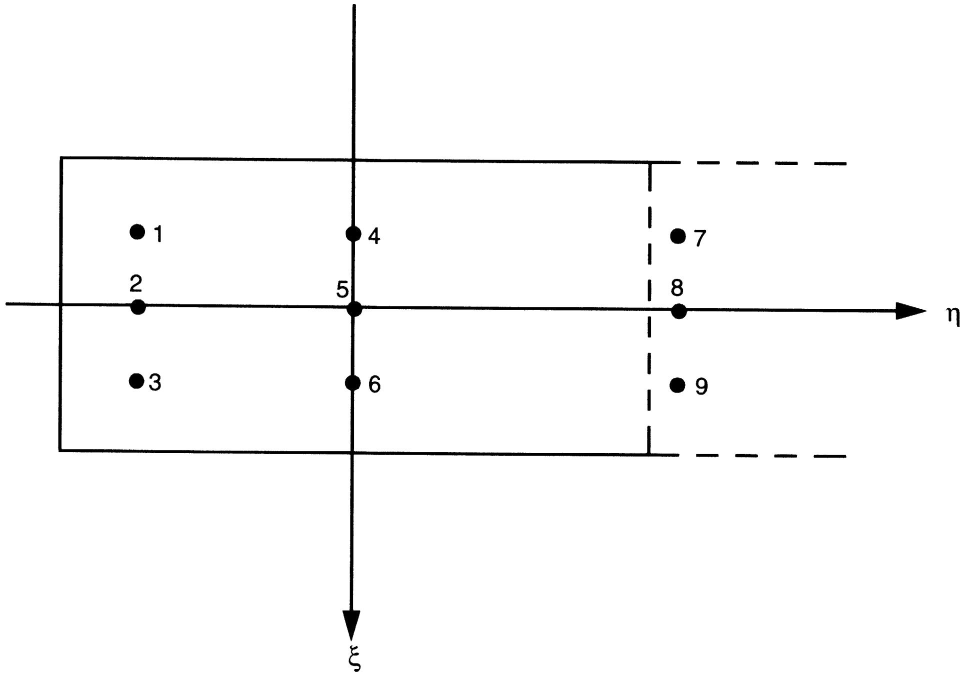


Figure 3-162 Integration Point Locations for Element 104

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

■ Element 105

Twelve-Node 3D Semi-infinite Heat Transfer Element

This is a 12-node 3D semi-infinite heat transfer element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element will expand to infinity. Both the conductivity and the capacity matrices are numerically integrated using 12 (2 x 3 x 2) integration points.

In addition, this element can be used for an electrostatic problem. A description of these two options can be found in Volume A.

Quick Reference

Type 105

Twelve-node 3D semi-infinite heat transfer element.

Connectivity

Twelve nodes per element.

See Figure 3-163 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3D heat transfer element and the 2-3-7-6, 5-6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 12-node 3D semi-infinite heat transfer element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)

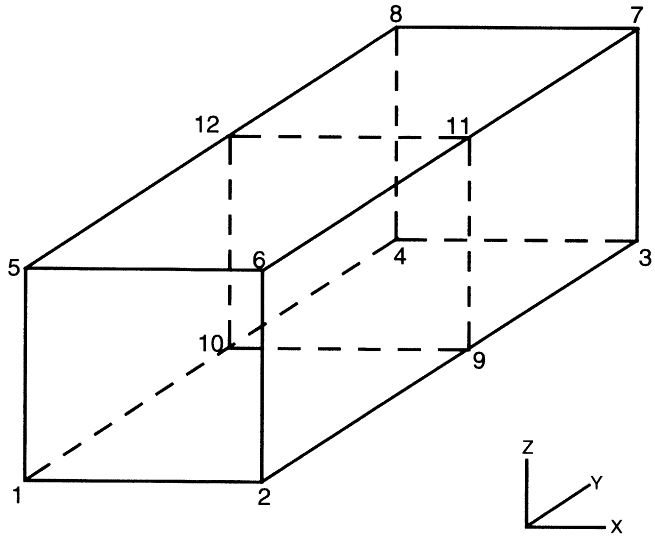


Figure 3-163 Twelve-Node 3D Semi-infinite Heat Transfer Element

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2-3-4 face of the element.
1	Nonuniform flux per unit area on 1-2-3-4 face of the element; magnitude given in subroutine FLUX.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
4	Uniform flux per unit area on 5-6-7-8 face of the element.
5	Nonuniform flux per unit area on 5-6-7-8 face of the element; magnitude given in subroutine FLUX.
6	Uniform flux per unit area on 1-2-6-5 face of the element.
7	Nonuniform flux per unit area on 1-2-6-5 face of the element; magnitude given in subroutine FLUX.
8	Uniform flux per unit area on 2-3-7-6 face of the element.

Flux Type	Description
9	Nonuniform flux per unit area on 2-3-7-6 face of the element; magnitude given in subroutine FLUX.
12	Uniform flux per unit area on 1-4-8-5 face of the element.
13	Nonuniform flux per unit area on 1-4-8-5 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as **Fluxes**.

Output Points

Center or 12 Gaussian integration points (see Figure 3-164).

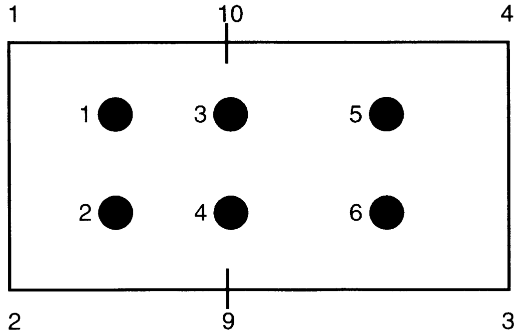


Figure 3-164 Integration Point Locations for Element 105

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 106

Twenty-seven-Node 3D Semi-infinite Heat Transfer Element

This is a 27-node 3D semi-infinite heat transfer element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element will expand to infinity. Both the conductivity and the capacity matrices are numerically integrated using 27 (3 x 3 x 3) integration points.

In addition, this element can be used for an electrostatic problem. A description of these two options can be found in Volume A.

Quick Reference

Type 106

Twenty-seven-node 3D semi-infinite heat transfer element.

Connectivity

Twenty-seven nodes per element.

See Figure 3-165 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3D heat transfer element and the 2-3-7-6, 5-6-7-8, 1-4-8-5, and 1-2-3-4 faces should be either connected to another 27-node 3D semi-infinite heat transfer element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)

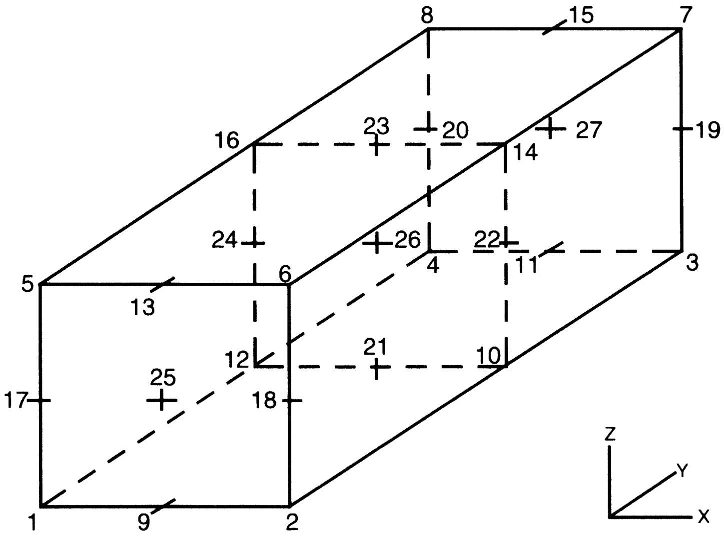


Figure 3-165 Twenty-seven-Node 3D Semi-infinite Heat Transfer Element

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2-3-4 face of the element.
1	Nonuniform flux per unit area on 1-2-3-4 face of the element; magnitude given in subroutine FLUX.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude given in subroutine FLUX.
4	Uniform flux per unit area on 5-6-7-8 face of the element.
5	Nonuniform flux per unit area on 5-6-7-8 face of the element; magnitude given in subroutine FLUX.
6	Uniform flux per unit area on 1-2-6-5 face of the element.
7	Nonuniform flux per unit area on 1-2-6-5 face of the element; magnitude given in subroutine FLUX.

Flux Type	Description
8	Uniform flux per unit area on 2-3-7-6 face of the element.
9	Nonuniform flux per unit area on 2-3-7-6 face of the element; magnitude given in subroutine FLUX.
12	Uniform flux per unit area on 1-4-8-5 face of the element.
13	Nonuniform flux per unit area on 1-4-8-5 face of the element; magnitude given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes.

Output Points

Center or 27 Gaussian integration points (see Figure 3-166).

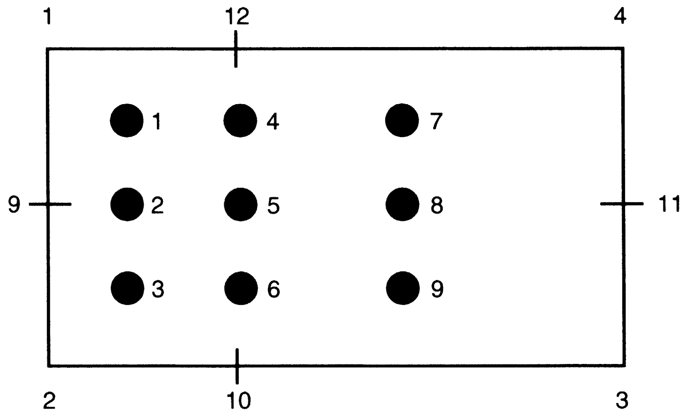


Figure 3-166 Integration Point Locations for Element 106

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 107

Twelve-Node 3D Semi-infinite Stress Element

This is a 12-node 3D semi-infinite stress element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element will expand to infinity. The stiffness matrix is numerically integrated using 12 (2 x 3 x 2) integration points. This element only has linear capability. This element may not be used with CONTACT.

Quick Reference

Type 107

Twelve-node 3D semi-infinite stress element.

Connectivity

Twelve nodes per element.

See Figure 3-167 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3D stress element and the 2-3-7-6, 5-6-7-8, 1-4-8-5, and 1-2-3-4 faces should be either connected to another 12-node 3D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = w displacement

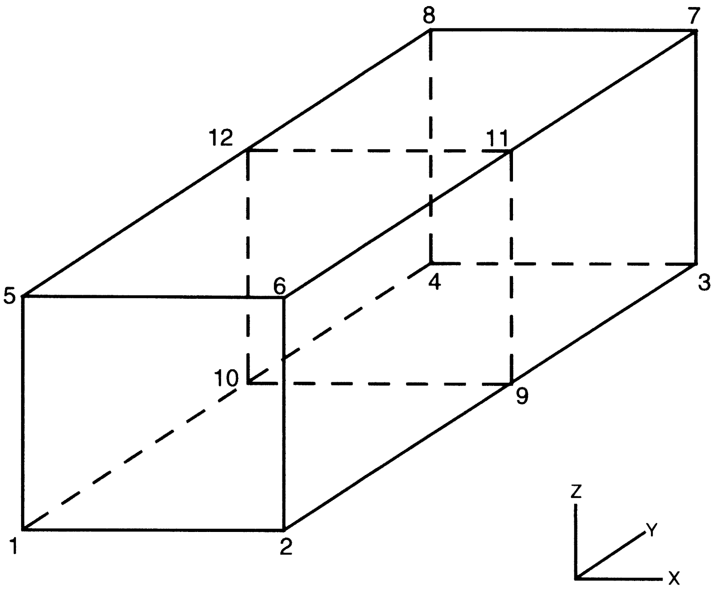


Figure 3-167 Twelve-Node 3D Semi-infinite Stress Element

Distributed Loads

Distributed loads are listed below:

Load Type	Description
0	Uniform load per unit area on 1-2-3-4 face of the element.
1	Nonuniform load per unit area on 1-2-3-4 face of the element; magnitude given in subroutine FORCEM.
2	Uniform load per unit volume on whole element.
3	Nonuniform load per unit volume on whole element; magnitude and direction given in subroutine FORCEM.
4	Uniform load per unit area on 5-6-7-8 face of the element.
5	Nonuniform load per unit area on 5-6-7-8 face of the element; magnitude given in subroutine FORCEM.
6	Uniform load per unit area on 1-2-6-5 face of the element.

Load Type	Description
7	Nonuniform load per unit area on 1-2-6-5 face of the element; magnitude given in subroutine FORCEM.
8	Uniform load per unit area on 2-3-7-6 face of the element.
9	Nonuniform load per unit area on 2-3-7-6 face of the element; magnitude given in subroutine FORCEM.
12	Uniform load per unit area on 1-4-8-5 face of the element.
13	Nonuniform load per unit area on 1-4-8-5 face of the element; magnitude given in subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads may be applied at nodes.

Output Points

Center or 12 Gaussian integration points (see Figure 3-168).

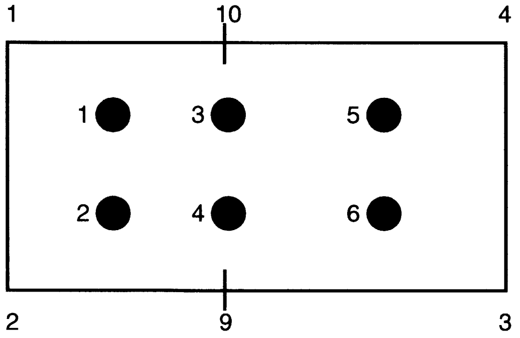


Figure 3-168 Integration Point Locations for Element 107

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 108

Twenty-seven-Node 3D Semi-infinite Stress Element

This is a 27-node 3D semi-infinite stress element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element will expand to infinity. The stiffness matrix is numerically integrated using 27 (3 x 3 x 3) integration points. This element only has linear capability. This element may not be used with CONTACT.

Quick Reference

Type 108

Twenty-seven-node 3D semi-infinite stress element.

Connectivity

Twenty-seven nodes per element.

See Figure 3-169 for numbering. The 1-2-6-5 face should be connected to a standard 20-node 3D stress element and the 2-3-7-6,6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 27-node 3D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

Geometry

Not required.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = w displacement

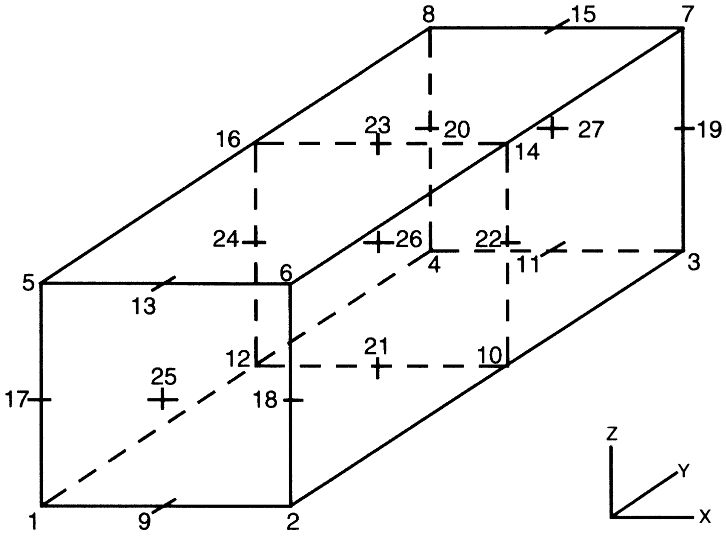


Figure 3-169 Twenty-seven-Node 3D Semi-infinite Stress Element

Distributed Loads

Distributed loads are listed below:

Load Type	Description
0	Uniform load per unit area on 1-2-3-4 face of the element.
1	Nonuniform load per unit area on 1-2-3-4 face of the element; magnitude given in subroutine FORCEM.
2	Uniform load per unit volume on whole element.
3	Nonuniform load per unit volume on whole element; magnitude and direction given in subroutine FORCEM.
4	Uniform load per unit area on 5-6-7-8 face of the element.
5	Nonuniform load per unit area on 5-6-7-8 face of the element; magnitude given in subroutine FORCEM.
6	Uniform load per unit area on 1-2-6-5 face of the element.
7	Nonuniform load per unit area on 1-2-6-5 face of the element; magnitude given in subroutine FORCEM.

Load Type	Description
8	Uniform load per unit area on 2-3-7-6 face of the element.
9	Nonuniform load per unit area on 2-3-7-6 face of the element; magnitude given in subroutine FORCEM.
12	Uniform load per unit area on 1-4-8-5 face of the element.
13	Nonuniform load per unit area on 1-4-8-5 face of the element; magnitude given in subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point loads may be applied at nodes.

Output Points

Center or 27 Gaussian integration points (see Figure 3-170).

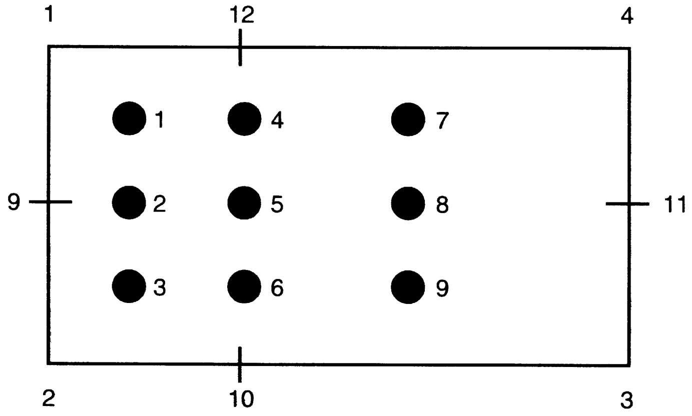


Figure 3-170 Integration Point Locations for Element 108

Tying

Use subroutine UFORMS.

Coupled Thermal-Stress Analysis

Capability is available.

Element 109

Eight-Node 3D Magnetostatic Element

This is an eight-node 3D magnetostatic element with linear interpolation functions. It is similar to element type 43. The coefficient matrix is numerically integrated using eight (2 x 2 x 2) integration points.

Quick Reference

Type 109

Eight-node 3D magnetostatic element.

Connectivity

Eight nodes per element.

See Figure 3-171 for numbering. Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on.

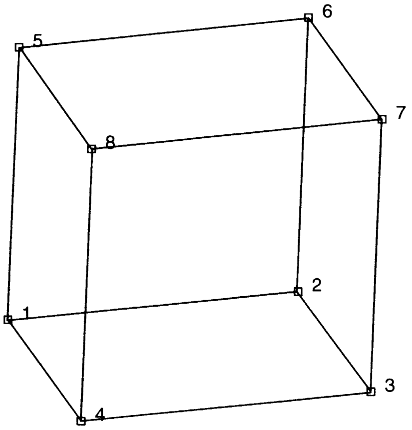


Figure 3-171 Eight-Node 3D Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current on 1-2-3-4 face.
1	Nonuniform current on 1-2-3-4 face; magnitude supplied through user subroutine FORCEM.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g. centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
4	Uniform current on 6-5-8-7 face.
5	Nonuniform current on 6-5-8-7 face.
6	Uniform current on 2-1-5-6 face.
7	Nonuniform current on 2-1-5-6 face.
8	Uniform current on 3-2-6-7 face.
9	Nonuniform current on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform current on 4-3-7-8 face.
12	Uniform current on 1-4-8-5 face.
13	Nonuniform current on 1-4-8-5 face.
20	Uniform current on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z direction.

Current Type	Description
23	Nonuniform body force per unit volume (e.g. centrifugal force); magnitude and direction supplied through user subroutine FORCEM.
24	Uniform current on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform current on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face.
28	Uniform current on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform current on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear 1-2-3-4 face in 1⇒2 direction.
43	Nonuniform shear 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear 3-2-6-7 face in 2⇒6 direction.

Current Type	Description
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.

For all nonuniform currents, body forces per unit volume and loads, the magnitude and direction are supplied via user subroutine FORCEM.

Currents are positive into element face.

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output Points

Centroid or eight Gaussian integration points.

Tying

Use subroutine UFORMS.

Element 110

Twelve-Node 3D Semi-infinite Magnetostatic Element

This is a 12-node 3D semi-infinite magnetostatic element that may be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-7-3 direction. Mappings are such that the element will expand to infinity. The coefficient matrix is numerically integrated using 12 (2 x 3 x 2) integration points.

Quick Reference

Type 110

Twelve-node 3D semi-infinite magnetostatic element.

Connectivity

Twelve nodes per element. See Figure 3-172 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3D stress element and the 2-3-7-6, 5-6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 12-node 3D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

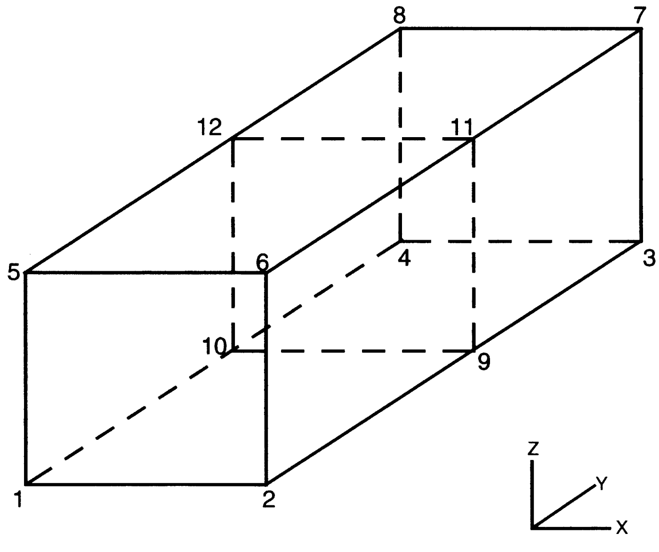


Figure 3-172 Twelve-Node 3D Semi-infinite Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current per unit area on 1-2-3-4 face of the element.
1	Nonuniform current per unit area on 1-2-3-4 face of the element; magnitude given in subroutine FORCEM.
2	Uniform current per unit volume on whole element.
3	Nonuniform current per unit volume on whole element; magnitude given in subroutine FORCEM.
4	Uniform current per unit area on 5-6-7-8 face of the element.
5	Nonuniform current per unit area on 5-6-7-8 face of the element; magnitude given in subroutine FORCEM.
6	Uniform current per unit area on 1-2-6-5 face of the element.
7	Nonuniform current per unit area on 1-2-6-5 face of the element; magnitude given in subroutine FORCEM.
8	Uniform current per unit area on 2-3-7-6 face of the element.
9	Nonuniform current per unit area on 2-3-7-6 face of the element; magnitude given in subroutine FORCEM.
12	Uniform current per unit area on 1-4-8-5 face of the element.
13	Nonuniform current per unit area on 1-4-8-5 face of the element; magnitude given in subroutine FORCEM.

Point currents may be applied at nodes.

Output Points

Center or 12 Gaussian integration points (see Figure 3-173).

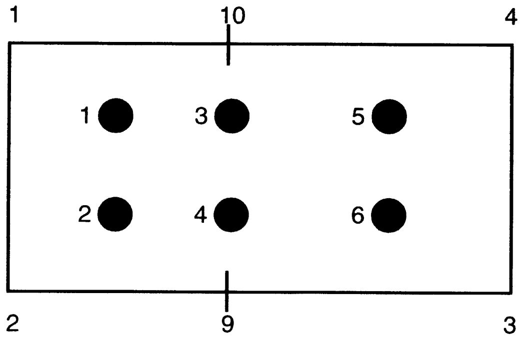


Figure 3-173 1-2 Integration Point Locations for Element 110

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Tying

Use subroutine UFORMS.

Element 111

Arbitrary Quadrilateral Planar Electromagnetic

Element type 111 is a four-node arbitrary quadrilateral written for planar electromagnetic applications. This element may be used for either transient or harmonic problems.

Quick Reference

Type 111

Planar quadrilateral.

Connectivity

Four nodes per element.

Node numbering follows right-handed convention (counterclockwise).

Geometry

Not applicable, the thickness is always equal to one.

Coordinates

Two coordinates in the global x- and y-directions.

Degrees Of Freedom

Global displacement degrees of freedom.

Magnetic Potential

1 = Ax
2 = Ay
3 = Az

Electric Potential

4 = V

Distributed Current

Current types for distributed currents as follows:

0 - 11	Currents normal to element edge.
20 - 27	Currents in plane, tangential to element edge.
30 - 41	Currents out of plane.

Current Type	Description
0	Uniform normal current everywhere distributed on 1-2 face of the element.
1	Uniform body current in the x-direction.
2	Uniform body current in the y-direction.
3	Nonuniform normal current everywhere on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body current in the x-direction.
5	Nonuniform body current in the y-direction
6	Uniform current on 2-3 face of the element.
7	Nonuniform current on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform current on 3-4 face of the element.
9	Nonuniform current on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
10	Uniform current on 4-1 face of the element.
11	Nonuniform current on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
20	Uniform shear current on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear current on side 1 - 2; magnitude supplied through user subroutine FORCEM.
22	Uniform shear current on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear current on side 2 - 3; magnitude supplied through user subroutine FORCEM.
24	Uniform shear current on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear current on side 3 - 4; magnitude supplied through user subroutine FORCEM.
26	Uniform shear current on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear current on side 4 - 1, magnitude supplied through user subroutine FORCEM.

Current Type	Description
30	Uniform current per unit area of 1-2 face of the element, normal to the plane.
31	Nonuniform current per unit area on 1-2 face of the element, normal to the plane; magnitude given in subroutine FORCEM.
36	Uniform current per unit area on 2-3 face of the element, normal to the plane.
37	Nonuniform current per unit area on 2-3 face of the element, normal to the plane; magnitude given in subroutine FORCEM.
38	Uniform current per unit area on 3-4 face of the element normal to the plane.
39	Nonuniform current per unit area on 3-4 face of the element, normal to the plane; magnitude given in subroutine FORCEM.
40	Uniform current per unit area on 4-1 face of the element, normal to the plane.
41	Nonuniform current per unit area on 4-1 face of the element, normal to the plane; magnitude given in subroutine FORCEM.

All currents are positive when directed into the element. In addition, point currents and charges may be applied at the nodes.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 face of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 face of the element.
54	Nonuniform charge per unit volume on whole element.
55	Nonuniform charge per unit volume on whole element.
56	Uniform charge per unit area on 2-3 face of the element.
57	Nonuniform charge per unit area on 2-3 face of the element.
58	Uniform charge per unit area on 3-4 face of the element.

Charge Type	Description
59	Nonuniform charge per unit area on 3-4 face of the element.
60	Uniform charge per unit area on 4-1 face of the element.
61	Nonuniform charge per unit area on 4-1 face of the element.

For all nonuniform charges, the magnitude is supplied through subroutine FORCEM.

All charges are positive when adding charge to the element. In addition, point charges may be applied at the nodes.

Output

Three components of:

Electric field intensity	E
Electric charge density	D
Magnetic field intensity	H
Magnetic charge density	B
Current density	J

Transformation

Two global degrees of freedom (Ax, Ay) may be transformed into local coordinates.

Output Points

Output is available at the four Gaussian points shown in Figure 3-174.

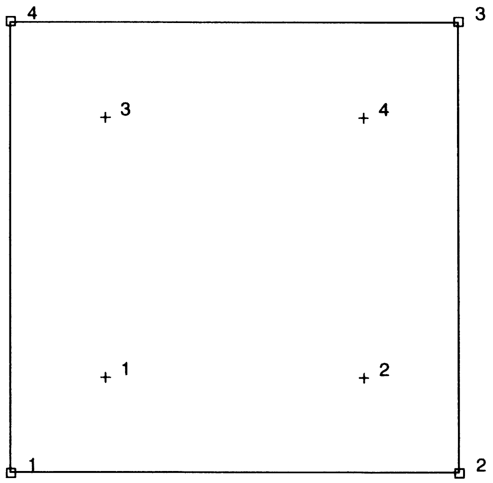


Figure 3-174 Gaussian Integration Points for Element Type 111

Element 112

Arbitrary Quadrilateral Axisymmetric Electromagnetic Ring

Element type 112 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric electromagnetic applications. This element may be used for either transient or harmonic analyses.

Quick Reference

Type 112

Axisymmetric, arbitrary ring with a quadrilateral cross-section.

Connectivity

Four nodes per element. Node numbering follows right-handed convention (counterclockwise).

Geometry

Not applicable for this element.

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Vector Potential

1 = Az
2 = Ar
3 = Aθ

Scalar Potential

4 = V

Distributed Currents

Current types for distributed currents are listed below:

Current Type	Description
0	Uniform normal current distributed on 1-2 face of the element.
1	Uniform body current in the z-direction.
2	Uniform body current in the r-direction.

Current Type	Description
3	Nonuniform normal current on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body current in the z-direction.
5	Nonuniform body current in the r-direction.
6	Uniform normal current on 2-3 face of the element.
7	Nonuniform normal current on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
8	Uniform normal current on 3-4 face of the element.
9	Nonuniform normal current on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
10	Uniform normal current on 4-1 face of the element.
11	Nonuniform normal current on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
20	Uniform shear current on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear current on side 1 - 2; magnitude supplied through user subroutine FORCEM.
22	Uniform shear current on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear current on side 2 - 3; magnitude supplied through user subroutine FORCEM.
24	Uniform shear current on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear current on side 3 - 4; magnitude supplied through user subroutine FORCEM.
26	Uniform shear current on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear current on side 4 - 1; magnitude supplied through user subroutine FORCEM.
30	Uniform current per unit area 1-2 face of the element in the theta direction.
33	Nonuniform current per unit area on 1-2 face of the element in the theta direction; magnitude given in subroutine FORCEM.

Current Type	Description
36	Uniform current per unit area on 2-3 face of the element in the theta direction.
37	Nonuniform current per unit area on 2-3 face of the element in the theta direction; magnitude given in subroutine FORCEM.
38	Uniform current per unit area on 3-4 face of the element in the theta direction.
39	Nonuniform current per unit area on 3-4 face of the element in the theta direction; magnitude given in subroutine FORCEM.
40	Uniform current per unit area on 4-1 face of the element in the theta direction.
41	Nonuniform current per unit area on 4-1 face of the element in the theta direction; magnitude given in subroutine FORCEM.

All currents are positive when directed into the element. In addition, point currents may be applied at the nodes. The magnitude of point current must correspond to the current integrated around the circumference.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 face of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 face of the element; magnitude given in subroutine FORCEM.
54	Nonuniform charge per unit volume on whole element; magnitude given in subroutine FORCEM.
55	Nonuniform charge per unit volume on whole element; magnitude given in subroutine FORCEM.
56	Uniform charge per unit area on 2-3 face of the element.
57	Nonuniform charge per unit area on 2-3 face of the element; magnitude given in subroutine FORCEM.

Charge Type	Description
58	Uniform charge per unit area on 3-4 face of the element.
59	Nonuniform charge per unit area on 3-4 face of the element; magnitude given in subroutine FORCEM.
60	Uniform charge per unit area on 4-1 face of the element.
61	Nonuniform charge per unit area on 4-1 face of the element; magnitude given in subroutine FORCEM.

All charges are positive when adding charge to the element. In addition, point charges may be applied at the nodes. The magnitude of the point charge must correspond to the charge integrated around the circumference.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Two global degrees (Az, Ar) of freedom may be transformed into local coordinates.

Output Points

Output is available at the four Gaussian points shown in Figure 3-175.

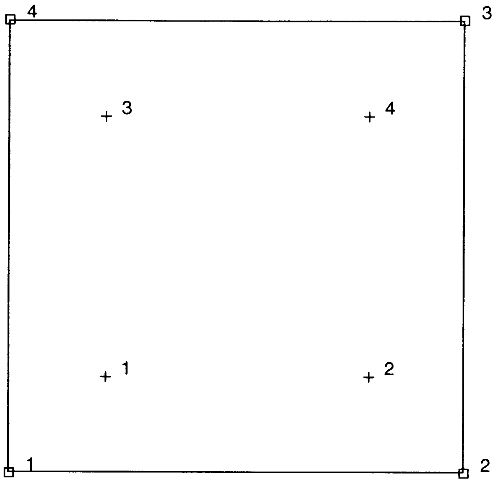


Figure 3-175 Integration Points for Element 112

Element 113

Three-Dimensional Electromagnetic Arbitrarily Distorted Brick

Element 113 is an eight-node isoparametric brick element and may be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using eight-point Gaussian integration.

Note: As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 113

Eight-node 3D first-order isoparametric element (arbitrarily distorted brick).

Connectivity

Eight nodes per element (see Figure 3-176). Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

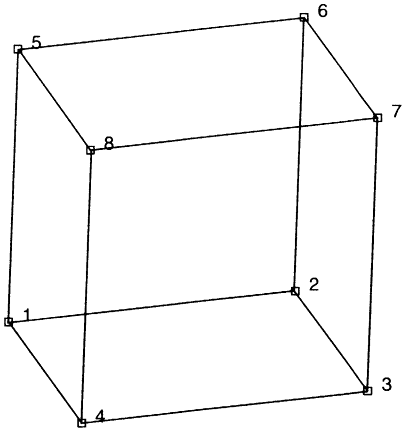


Figure 3-176 Nodes and Integration Point for Element 113

Geometry

Not applicable for this element.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Four global degrees of freedom Ax, Ay, Az, V per node.

Distributed Currents

Distributed currents chosen by value of IBODY as follows:

Current Type	Description
0	Uniform normal current on 1-2-3-4 face.
1	Nonuniform normal current on 1-2-3-4 face.
4	Uniform normal current on 6-5-8-7 face.
5	Nonuniform normal current on 6-5-8-7 face.
6	Uniform normal current on 2-1-5-6 face.
7	Nonuniform normal current on 2-1-5-6 face.
8	Uniform normal current on 3-2-6-7 face.
9	Nonuniform normal current on 3-2-6-7 face.
10	Uniform normal current on 4-3-7-8 face.
11	Nonuniform normal current on 4-3-7-8 face.
12	Uniform normal current on 1-4-8-5 face.
13	Nonuniform normal current on 1-4-8-5 face.
20	Uniform normal current on 1-2-3-4 face.
21	Nonuniform current on 1-2-3-4 face.
22	Same as 2.
23	Same as 3.
24	Uniform normal current on 6-5-8-7 face.
25	Nonuniform current on 6-5-8-7 face.
26	Uniform normal current on 2-1-5-6 face.

Current Type	Description
27	Nonuniform current on 2-1-5-6 face.
28	Uniform normal current on 3-2-6-7 face.
29	Nonuniform current on 3-2-6-7 face.
30	Uniform normal current on 4-3-7-8 face.
31	Nonuniform current on 4-3-7-8 face.
32	Uniform normal current on 1-4-8-5 face.
33	Nonuniform current on 1-4-8-5 face.
40	Uniform shear current 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear current 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear current 1-2-3-4 face in 2⇒3 direction.
43	Nonuniform shear current 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear current 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear current 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear current 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear current 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear current 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear current 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear current 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear current 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear current 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear current 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear current 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear current 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear current 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear current 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear current 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear current 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear current 1-4-8-5 face in 4⇒1 direction.

Current Type	Description
65	Nonuniform shear current 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear current 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear current 1-4-8-5 face in 1⇒5 direction.

For all nonuniform normal and shear currents, the magnitude is supplied through user subroutine FORCEM.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
70	Uniform current on 1-2-3-4 face.
71	Nonuniform charge on 1-2-3-4 face.
74	Uniform charge on 5-6-7-8 face.
75	Nonuniform charge on 5-6-7-8 face.
76	Uniform charge on 1-2-6-5 face.
77	Nonuniform charge on 1-2-6-5 face.
78	Uniform charge on 2-3-7-6 face.
79	Nonuniform charge on 2-3-7-6 face
80	Uniform charge on 3-4-8-7 face.
81	Nonuniform charge on 3-4-8-7 face.
82	Uniform charge on 1-4-8-5 face.
83	Nonuniform charge on 1-4-8-5 face.

For all nonuniform charges, the magnitude is supplied through user subroutine FORCEM.

Charges are positive into the element face.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Eight integration points as shown in Figure 3-176.

Element 114

Plane Stress Quadrilateral, Reduced Integration

Element type 114 is a four-node isoparametric arbitrary quadrilateral written for plane stress applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

All constitutive models may be used with this element. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Quick Reference

Type 114

Plane stress quadrilateral, using reduced integration.

Connectivity

Node numbering must follow right-handed convention (counterclockwise). See Figure 3-177.

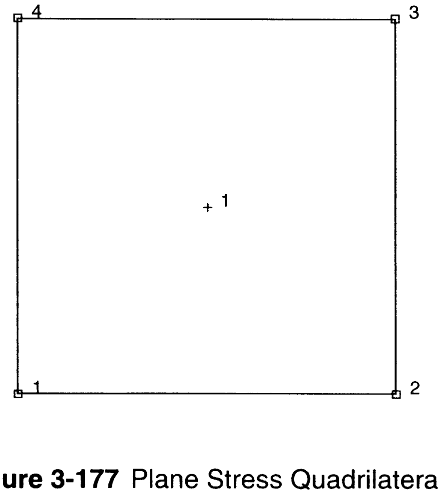


Figure 3-177 Plane Stress Quadrilateral

Geometry

The thickness is stored in the first data field (EGEOM1).

Default thickness is 1.

The second field is not used.

Coordinates

Two global coordinates, x and y directions.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through subroutine FORCEM.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through FORCEM.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through FORCEM.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through subroutine FORCEM.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through subroutine FORCEM.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through subroutine FORCEM.
* 20	Uniform shear force on side 1 - 2 (positive from 1 to 2).

Load Type	Description
21	Nonuniform shear force on side 1 - 2; magnitude supplied through subroutine FORCEM.
* 22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2 - 3; magnitude supplied through subroutine FORCEM.
* 24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3 - 4; magnitude supplied through subroutine FORCEM.
* 26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4 - 1; magnitude supplied through subroutine FORCEM.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x- and y-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element are:

ε_{xx}
ε_{yy}
γ_{xy}

Output of Stresses

Same as for **Output of Strains**.

Output Points

Output is available at the centroid.

Transformation

The two global degrees of freedom at the corner nodes may be transformed into local coordinates.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of true stress and logarithmic strain in global coordinate directions. Thickness will be updated if the FINITE parameter is specified.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Design Variables

The thickness can be considered a design variable.

Element 115

Arbitrary Quadrilateral Plane Strain, Reduced Integration

Element type 115 is a four-node isoparametric arbitrary quadrilateral written for plane strain applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 118 instead. Element type 118 is also preferable for small strain incompressible elasticity. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Quick Reference

Type 115

Plane stress quadrilateral, using reduced integration.

Connectivity

Four nodes per element. Node numbering follows right-handed convention (counterclockwise). See Figure 3-178.

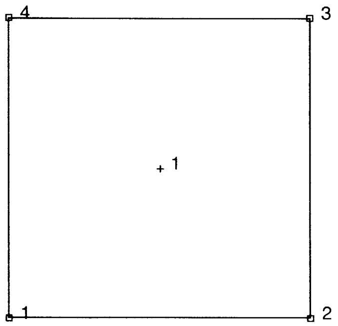


Figure 3-178 Nodes and Integration Point for Element 115

Geometry

The thickness is stored in the first data field (EGEOM1).

Default thickness is 1.

Coordinates

Two coordinates in the global -x and y-directions.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).

Load Type	Description
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4.
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform pressures, body and shear forces, magnitude is supplied through subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element are:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz} = 0$
$4 = \gamma_{xy}$

Output of Stresses

Same as for **Output of Strains.**

Transformation

The two global degrees of freedom at the corner nodes may be transformed into local coordinates.

Tying

Use subroutine UFORMS.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain in global coordinate directions. This element does not lock for nearly incompressible materials.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 116

Arbitrary Quadrilateral Axisymmetric Ring, Reduced Integration

Element type 116 is a four-node isoparametric arbitrary quadrilateral written for axisymmetric applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 80 instead. Element type 80 is also preferable for small strain incompressible elasticity. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Quick Reference

Type 116

Axisymmetric, arbitrary ring with a quadrilateral cross section, using reduced integration.

Connectivity

Four nodes per element (see Figure 3-179). Node numbering for the corner nodes follows right-handed convention (counterclockwise).

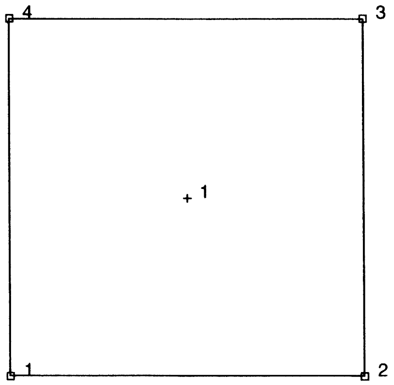


Figure 3-179 Integration Point for Element 116

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = u (displacement in the global z direction)
2 = v (displacement in the global r direction)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.

Load Type	Description
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1⇒2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform pressures and shear forces, the magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{rz}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

The global degrees of freedom at the corner nodes may be transformed into local coordinates.

Tying

May be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain in global coordinate directions. This element does not lock for nearly incompressible materials.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 122. See Element 122 for a description of the conventions used for entering the flux and film data for this element.

Element 117

Three-Dimensional Arbitrarily Distorted Brick, Reduced Integration

Element type 117 is an eight-node isoparametric arbitrary hexahedral for general three-dimensional applications using reduced integration.This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 120 instead. Element type 120 is also preferable for small strain incompressible elasticity. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Note: As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 117

Eight-node 3D first-order isoparametric element (arbitrarily distorted cube) using reduced integration.

Connectivity

Eight nodes per element (see Figure 3-180). Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

Geometry

If the automatic brick to shell constraints are to be used, the first field must contain the transition thickness.

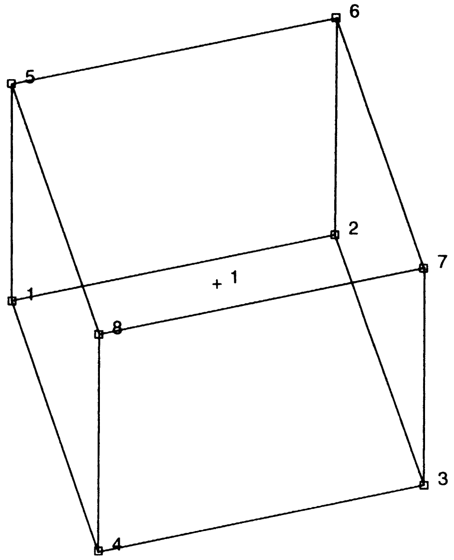


Figure 3-180 Nodes and Integration Point for Element 117

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction given in subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.

Load Type	Description
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force)
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear 1-2-3-4 face in 2⇒3 direction.
43	Nonuniform shear 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.

Load Type	Description
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Pressure forces are positive into element face.

Output of Strains

$1 = \varepsilon_{xx}$ = global xx strain
$2 = \varepsilon_{yy}$ = global yy strain
$3 = \varepsilon_{zz}$ = global zz strain
$4 = \gamma_{xy}$ = global xy strain
$5 = \gamma_{yz}$ = global yz strain
$6 = \gamma_{zx}$ = global xz strain

Output of Stresses

Output of stress is the same as for **Output of Strains**.

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom at the corner nodes.

Tying

No special tying available. An automatic constraint is available for brick to shell transition meshes (see **Geometry**).

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 123. See Element 123 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Notes: The element can be collapsed to a tetrahedron.

By collapsing one plane of the element to a line (see Figure 3-181) a transition element for connecting bricks with a four-node shell element type 75 is generated. Thickness of the shell is specified in the geometry field.

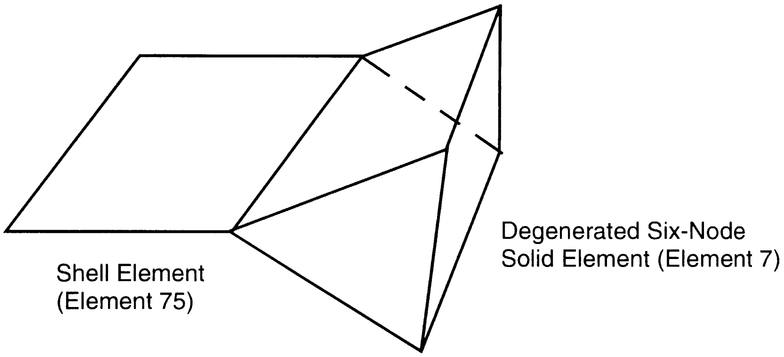


Figure 3-181 Shell-to-Solid Automatic Constraint

Element 118

Arbitrary Quadrilateral Plane Strain, Incompressible Formulation with Reduced Integration

Element type 118 is a four-node isoparametric arbitrary quadrilateral written for incompressible plane strain applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element. The displacement formulation has been modified using the Herrmann variational principle, the pressure field is constant in this element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 115 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 118

Plane strain quadrilateral, Herrmann formulation, using reduced integration.

Connectivity

Five nodes per element (see Figure 3-182). Node numbering for the corners follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note: Avoid reducing this element into a triangle.

Geometry

The thickness is entered in the first data field (EGEOM1). Defaults to unit thickness.

Coordinates

Two coordinates in the global x- and y-directions for the corner nodes. No coordinates are necessary for the fifth hydrostatic pressure node.

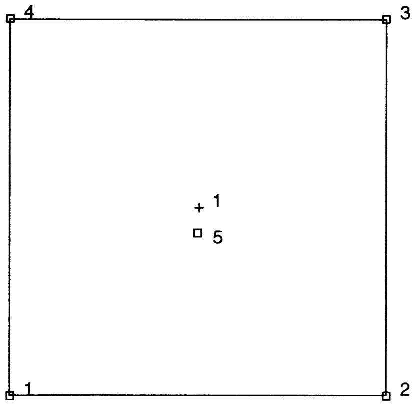


Figure 3-182 Nodes and Integration Point for Element Type 118

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = u displacement (x-direction)
2 = v displacement (y-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node.

1 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.

Load Type	Description
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1⇒2 direction.
21	Nonuniform shear force on side 1-2.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates are:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz} = 0$
$4 = \gamma_{xy}$
$5 = \sigma_{kk}/E$ = mean pressure variable (for Herrmann)
$= -p$ = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Same as for **Output of Strains**.

Transformation

The two global degrees of freedom at the corner nodes may be transformed into local coordinates.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element.

Element 119

Arbitrary Quadrilateral Axisymmetric Ring, Incompressible Formulation with Reduced Integration

Element type 119 is a four-node isoparametric arbitrary quadrilateral written for incompressible axisymmetric applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 116 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 119

Axisymmetric, arbitrary ring with a quadrilateral cross-section, Herrmann formulation, using reduced integration.

Connectivity

Five nodes per element (see Figure 3-183). Node numbering for the corner nodes follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note: Avoid reducing the element into a triangle.

Coordinates

Two coordinates in the global z- and r-directions for the corner nodes. No coordinates are necessary for the hydrostatic pressure node.

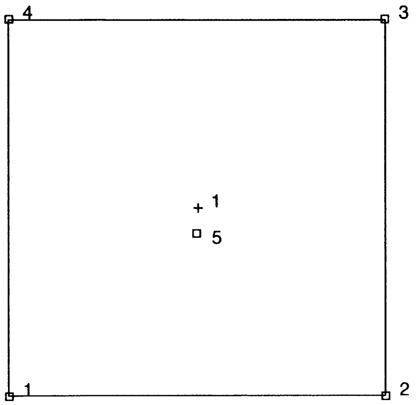


Figure 3-183 Integration Point for Element 119

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = axial displacement (in z-direction)
2 = radial displacement (in r-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node:

1 = σ_{kk}/E = mean pressure variable (for Herrmann)
= $-p$ = negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.

Load Type	Description
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1⇒2 direction.
21	Nonuniform shear force on side 1-2.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

All pressures are positive when directed into the element. In addition, point loads may be applied at the nodes.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{rz}$
$5 = \sigma_{kk}/E$ = mean pressure variable (for Herrmann)
$= -p$ = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Same as for **Output of Strains**.

Transformation

The global degrees of freedom at the corner nodes may be transformed into local coordinates. No transformation for the pressure node.

Tying

May be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element.

■ Element 120

Three-Dimensional Arbitrarily Distorted Brick, Incompressible Reduced Integration

Element type 120 is an eight-node isoparametric arbitrary hexahedral written for general three dimensional incompressible applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element may be used in conjunction with other elements such as type 117 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 120

Nine-node 3D first-order isoparametric element (arbitrarily distorted cube) with mixed formulation, using reduced integration.

Connectivity

Nine nodes per element (see Figure 3-184).

Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

The node with the pressure degree of freedom is the last node in the connectivity list, and should not be shared with other elements.

Note: Avoid reducing this element into a tetrahedron or a wedge.

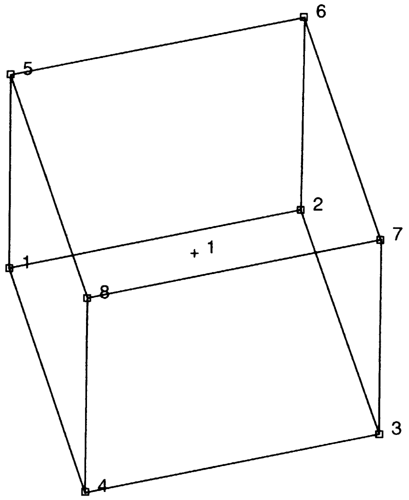


Figure 3-184 Nodes and Integration Point for Element 120

Coordinates

Three coordinates in the global x-, y-, and z-directions for the first eight nodes. No coordinates are necessary for the pressure node.

Degrees of Freedom

Three global degrees of freedom u, v, and w at the first eight nodes. One degree of freedom (negative hydrostatic pressure) at the last node.

1 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction given in subroutine FORCEM.
4	Uniform pressure on 6-5-8-7 face.

Load Type	Description
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear 1-2-3-4 face in 2⇒3 direction.
43	Nonuniform shear 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear 6-5-8-7 face in 5⇒6 direction.

Load Type	Description
50	Uniform shear 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒
59	Nonuniform shear 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform pressures, loads, and shear forces, magnitude is supplied via user subroutine FORCEM.

Pressure forces are positive into element face.

Output of Strains

1 = ε_{xx} = global xx strain
2 = ε_{yy} = global yy strain
3 = ε_{zz} = global zz strain
5 = γ_{yz} = global yz strain
6 = γ_{zx} = global zx strain
7 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney or Ogden)

Output of Stresses

Output of stress is the same as for **Output of Strains**.

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom at the corner nodes. No transformation for the pressure node.

Tying

No special tying available.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 123. See Element 123 for a description of the conventions used for entering the flux and film data for this element.

■ Element 121

Planar Bilinear Quadrilateral, Reduced Integration (Heat Transfer Element)

Element type 121 is a four-node isoparametric arbitrary quadrilateral written for planar heat transfer applications using reduced integration. This element may also be used for electrostatic or magnetostatic applications.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the **Quick Reference**.

In general, one needs more of these lower-order elements than the higher-order elements such as types 41 or 69. Hence, use a fine mesh.

The conductivity of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent conductivity term is included to eliminate the hourglass modes that are normally associated with reduced integration.

Quick Reference

Type 121

Arbitrary, planar, heat transfer quadrilateral, using reduced integration.

Connectivity

Node numbering must follow right-hand convention (see Figure 3-185).

Geometry

Thickness is input in the first data field (EGEOM1). The other two data fields are not used. If no thickness is input, unit thickness will be assumed.

Coordinates

Two global coordinates, x and y.

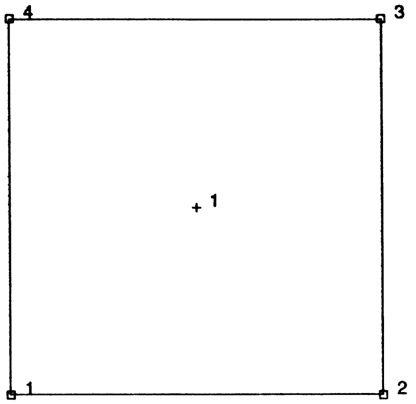


Figure 3-185 Nodes and Integration Point for Element 121

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element.
4	Nonuniform flux per unit volume on whole element.
5	Nonuniform flux per unit volume on whole element.
6	Uniform flux per unit area on 2-3 face of the element.

Flux Type	Description
7	Nonuniform flux per unit area on 2-3 face of the element.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element.

For all nonuniform fluxes, the magnitude is given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes.**

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes.**

Charge

Same specifications as **Fluxes.**

Output Points

Centroid.

Tying

Use subroutine UFORMS.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

■ Element 122

Axisymmetric Bilinear Quadrilateral, Reduced Integration (Heat Transfer Element)

Element type 122 is a four-node isoparametric arbitrary quadrilateral written for axisymmetric heat transfer applications using reduced integration. This element may also be used for electrostatic or magnetostatic applications.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the **Quick Reference**. The view factors calculation for radiation boundary conditions is available for this axisymmetric heat transfer element.

In general, one needs more of these lower-order elements than the higher-order elements such as types 42 or 70. Hence, use a fine mesh.

The conductivity of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent conductivity term is included to eliminate the hourglass modes that are normally associated with reduced integration.

Quick Reference

Type 122

Arbitrarily distorted axisymmetric heat transfer quadrilateral, using reduced integration.

Connectivity

Node numbering must follow right-hand convention (see Figure 3-186).

Geometry

Not applicable.

Coordinates

Two global coordinates, z and r.

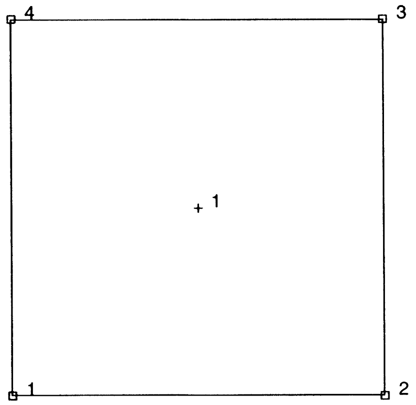


Figure 3-186 Nodes and Integration Point for Element 122

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element.
4	Nonuniform flux per unit volume on whole element.
5	Nonuniform flux per unit volume on whole element.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element.

Flux Type	Description
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element.

For all nonuniform fluxes, the magnitude is given in subroutine FLUX.

All fluxes are positive when adding heat to the element. In addition, point fluxes may be applied at the nodes.

Films

Same specification as **Fluxes**.

Tying

Use subroutine UFORMS.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

View Factors Calculation for Radiation

Capability is available.

Element 123

Three-Dimensional Eight-Node Brick, Reduced Integration (Heat Transfer Element)

Element type 123 is an eight-node isoparametric arbitrary hexahedral written for three dimensional heat transfer applications using reduced integration. This element may also be used for electrostatic applications.

Element type 123 can also be used as a thermal contact or a fluid channel element. Model definition blocks CONRAD GAP and CHANNEL must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Volume A. Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each GAP/CHANNEL. Face identifications for this element are given in the **Quick Reference**.

As this element uses trilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as types 44 or 71. Hence, use a fine mesh.

The conductivity of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

Quick Reference

Type 123

Eight-node 3D first-order, isoparametric heat transfer element using reduced integration.

Connectivity

Eight nodes per element, labelled as follows:

> Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on (see Figure 3-187).

Geometry

Not applicable for this element.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

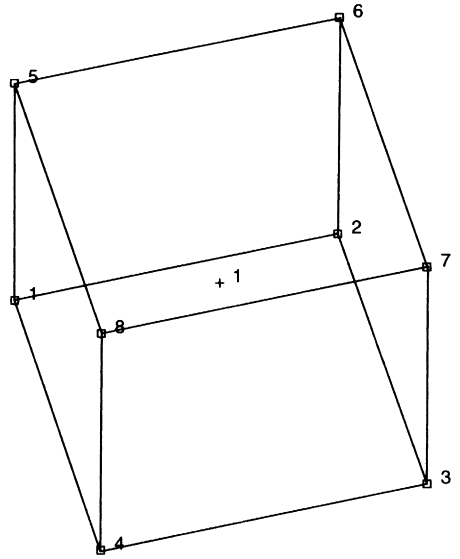


Figure 3-187 Nodes and Integration Point for Element 123

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are assumed positive when directed into the element.

Load Type (IBODY)	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform surface flux on 1-2-3-4 face.
2	Uniform volumetric flux.

Load Type (IBODY)	Description
3	Nonuniform volumetric flux.
4	Uniform flux on 5-6-7-8 face.
5	Nonuniform surface flux on 5-6-7-8 face.
6	Uniform flux on 1-2-6-5 face.
7	Nonuniform flux on 1-2-6-5 face.
8	Uniform flux on 2-3-7-6 face.
9	Nonuniform flux on 2-3-7-6 face.
10	Uniform flux on 3-4-8-7 face.
11	Nonuniform flux on 3-4-8-7 face.
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face.

For all nonuniform fluxes, the magnitude is supplied via subroutine FLUX.

For IBODY= 3, P is the magnitude of volumetric flux at volumetric integration point NN of element N. For IBODY odd but not equal to 3, P is the magnitude of surface flux for surface integration point NN of element N.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

Centroid.

Note: As in all three-dimensional analysis a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 124

Plane Stress, Six-Node Distorted Triangle

This is a second-order isoparametric two-dimensional plane stress triangular element. Displacement and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-188). The interpolation function is such that each edge has parabolic variation along itself. This allows for an accurate representation of the strain field in elastic analyses.

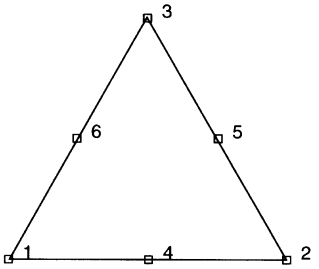


Figure 3-188 Nodes of Six-Node, 2D Element

The stiffness of this element is formed using three-point integration.

All constitutive relations may be used with this element.

The connectivity ordering is shown in Figure 3-188. Note that the basic numbering is counterclockwise in the (x-y) plane.

The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems, especially when bending behavior in the plane of the elements is expected, since the basic strain variation in the second-order elements is linear in any direction.

Quick Reference

Type 124

Second order, isoparametric, distorted triangle. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth between second and third, etc. See Figure 3-188.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-4-2 face.
* 1	Nonuniform pressure on 1-4-2 face.
* 2	Uniform shear on 1-4-2 face.
* 3	Nonuniform shear on 1-4-2 face.
* 4	Uniform pressure on 2-5-3 face.
* 5	Nonuniform pressure on 2-5-3 face.
* 6	Uniform shear on 2-5-3 face.
* 7	Nonuniform shear on 2-5-3 face.
* 8	Uniform pressure on 3-6-1 face.
* 9	Nonuniform pressure on 3-6-1 face.
* 10	Uniform shear on 3-6-1 face.
* 11	Nonuniform shear on 3-6-1 face.
* 12	Uniform body force in x-direction.
* 13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y direction.

Load Type (IBODY)	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, y direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-189 and **Output Points**) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = γ_{xy}, shear

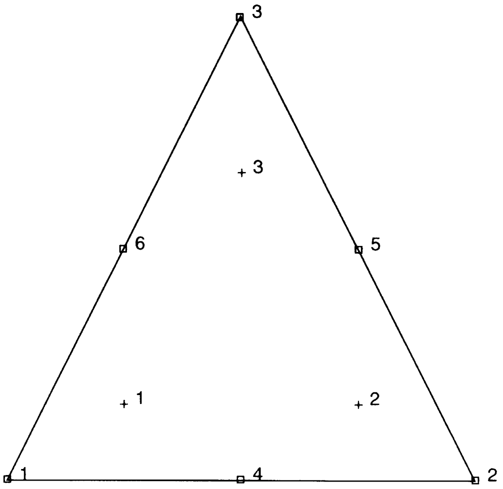


Figure 3-189 Integration Points of Six-Node, 2D Element

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-189. This is the usual option for a second-order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness will be updated.

Note: Distortion of element during solution may cause poor results. Element type 3 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 131. See Element 131 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 125

Plane Strain, Six-Node Distorted Triangle

This is a second-order isoparametric two-dimensional plane strain triangular element. Displacement and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-190). The interpolation function is such that each edge has parabolic variation along itself. This allows for an accurate representation of the strain field in elastic analyses.

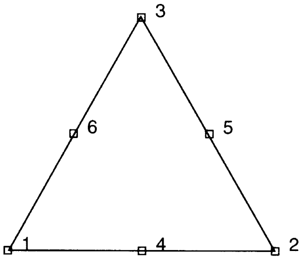


Figure 3-190 Nodes of Six-Node, 2D Element

The stiffness of this element is formed using three-point integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 128. Element type 128 is also preferable for small strain incompressible elasticity.

The connectivity ordering is shown in Figure 3-190. Note that the basic numbering is counterclockwise in the (x-y) plane.

The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems, especially when bending behavior in the plane of the elements is expected, since the basic strain variation in the second-order elements is linear in any direction.

Quick Reference

Type 125

Second order isoparametric distorted triangle. Plane strain.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-190.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in x-direction.
13	Nonuniform body force in x-direction.

Load Type (IBODY)	Description
14	Uniform body force in y-direction.
15	Nonuniform body force in y-direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, y direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-191 and **Output Points**) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, direct
4 = γ_{xy}, shear

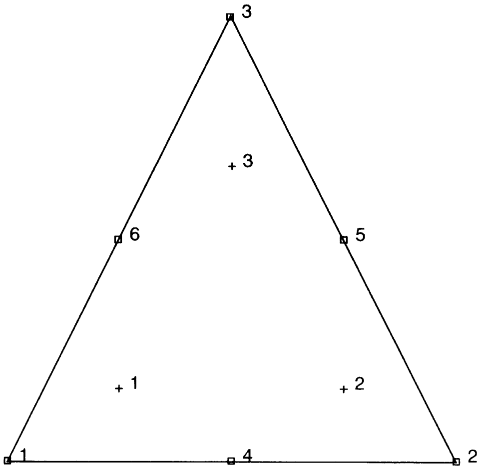


Figure 3-191 Integration Points of Six-Node, 2D Element

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-191. This is the usual option for a second order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness will be updated.

Note: Distortion of element during solution may cause poor results. Element type 11 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 131. See Element 131 for a description of the conventions used for entering the flux and film data for this element.

Element 126

Axisymmetric, Six-Node Distorted Triangle

This is a second-order isoparametric two-dimensional axisymmetric triangular element. Displacement and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-192). The interpolation function is such that each edge has parabolic variation along itself. This allows for an accurate representation of the strain field in elastic analyses.

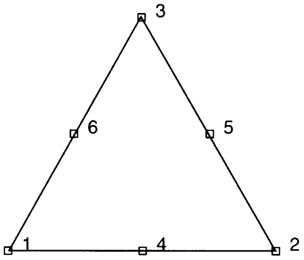


Figure 3-192 Nodes of Six-Node, Axisymmetric Element

The stiffness of this element is formed using three-point integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 129. Element type 129 is also preferable for small strain incompressible elasticity.

The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems, especially when bending behavior in the plane of the elements is expected, since the basic strain variation in the second-order elements is linear in any direction.

Quick Reference

Type 126

Second-order isoparametric distorted triangle. Axisymmetric.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-192.

Geometry

Geometry input is not necessary for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = global z-direction displacement
2 = v = global r-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in z-direction.
13	Nonuniform body force in z-direction.
14	Uniform body force in r-direction.
15	Nonuniform body force in r-direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.

Load Type (IBODY)	Description
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global z, r direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-193 and **Output Points**) in the following order:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direct
4 = γ_{xy}, shear in the section

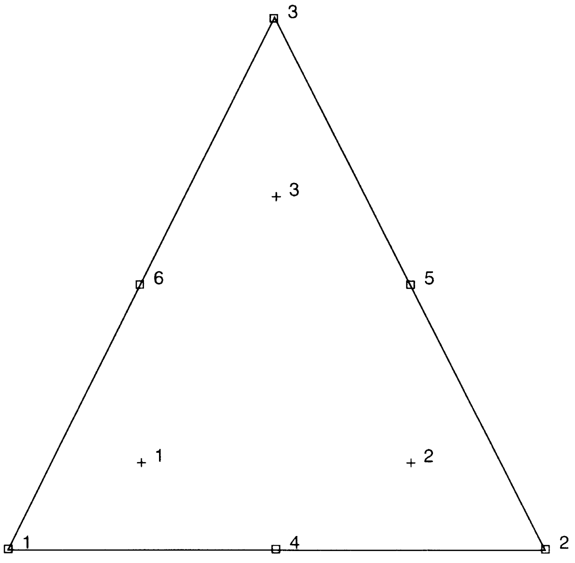


Figure 3-193 Integration Points of Six-Node, Axisymmetric Element

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-193. This is the usual option for a second-order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available - output of stress and strain in global coordinates. Thickness will be updated.

Note: Distortion of element during solution may cause poor results. Element type 10 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 132. See Element 132 for a description of the conventions used for entering the flux and film data for this element.

Element 127

Three-Dimensional Ten-Node Tetrahedron

This element is a second-order isoparametric three-dimensional tetrahedron. Each edge forms a parabola so that four nodes define the corners of the element and a further six nodes define the position of the "midpoint" of each edge (Figure 3-194). This allows for an accurate representation of the strain field in elastic analyses.

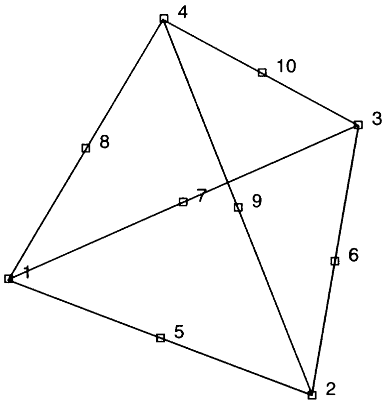


Figure 3-194 Form of Element 127

The stiffness of this element is formed using four-point integration.

This element may be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 130. Element type 130 is also preferable for small strain incompressible elasticity.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat) so that the user need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element with the first point closest to the first node of the element (see Figure 3-195).

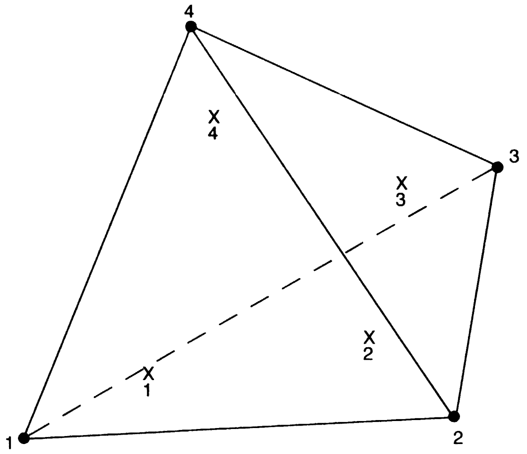


Figure 3-195 Element 127 Integration Plane

Quick Reference

Type 127

Ten nodes, isoparametric arbitrary distorted tetrahedron.

ConnectivitY

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-194.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x-direction.
9	Nonuniform body force per unit volume in x-direction.
10	Uniform body force per unit volume in y-direction.
11	Nonuniform body force per unit volume in y-direction.
12	Uniform body force per unit volume in z-direction.
13	Nonuniform body force per unit volume in z-direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

The subroutine FORCEM will be called once per integration point when flagged. The magnitude of load defined by DIST LOADS will be ignored and the FORCEM value will be used instead.

For nonuniform body force, force values must be provided for the four integration points.

For nonuniform surface pressure, force values need only be supplied for the three integration points on the face of application.

Output of Strains

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \varepsilon_{xy}$
$5 = \varepsilon_{yz}$
$6 = \varepsilon_{zx}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points (see Figure 3-195).

The user should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Note: A large bandwidth results in a lengthy central processing time.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note: Distortion of element during analysis may cause bad solutions. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used or entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 128

Plane Strain, Six-Node Distorted Triangle, Herrmann Formulation

This is a second-order isoparametric triangular element written for incompressible plane strain applications. This element uses biquadratic interpolation functions to represent coordinates and displacements. This allows for an accurate representation of the strain field. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using three integration points.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element may be used in conjunction with other elements such as type 125 when other material behavior, such as plasticity, must be represented.

The connectivity ordering is shown in Figure 3-196. Note that the basic numbering is counterclockwise in the (x-y) plane.

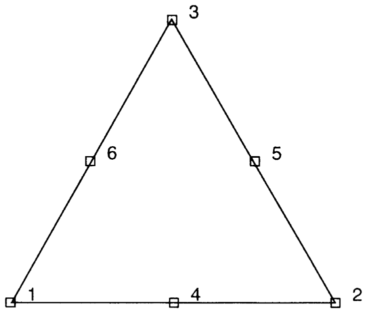


Figure 3-196 Nodes of Six-Node, 2D Element

Quick Reference

Type 128

Second-order isoparametric distorted triangle. Plane strain. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-196.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

At each corner node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

At corner nodes only,

3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.

Load Type (IBODY)	Description
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in x-direction.
13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, y direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-197 and **Output Points**) in the following order:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction, direct
4 = γ_{xy}, shear
5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative pressure (for Mooney)

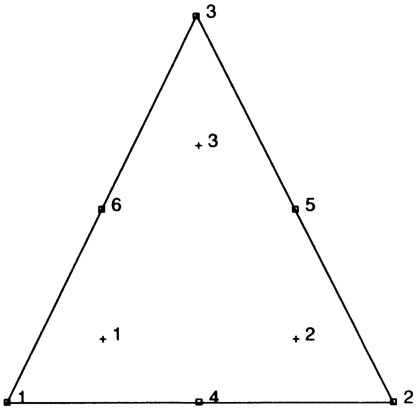


Figure 3-197 Integration Points of Six-Node, 2D Element

Output of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in x-y plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-197. This is the usual option for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 131. See Element 131 for a description of the conventions used for entering the flux and film data for this element.

Element 129

Axisymmetric, Six-Node Distorted Triangle, Herrmann Formulation

This is a second-order isoparametric triangular element written for incompressible axisymmetric applications. This element uses biquadratic interpolation functions to represent coordinates and displacements. This allows for an accurate representation of the strain field. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using three integration points.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element may be used in conjunction with other elements such as type 126 when other material behavior, such as plasticity, must be represented.

The connectivity ordering is shown in Figure 3-198. Note that the basic numbering is counterclockwise in the (z-r) plane.

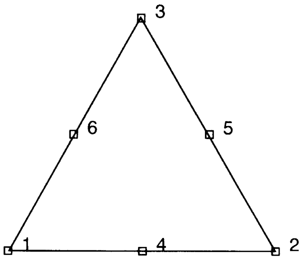


Figure 3-198 Nodes of Six-Node, 2D Element

Quick Reference

Type 129

Second order, isoparametric, distorted triangle. Axisymmetric. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-198.

Geometry

Not required.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

At each corner node:

1 = u = global z-direction displacement
2 = v = global r-direction displacement

At corner nodes only:

3 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.

Load Type (IBODY)	Description
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in x-direction.
13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global z, r direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

For all nonuniform loads, the load magnitude is supplied via user subroutine FORCEM.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-199 and **Output Points** on the following page) in the following order:

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, hoop direction, direct
4 = γ_{xy}, shear
5 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative pressure (for Mooney)

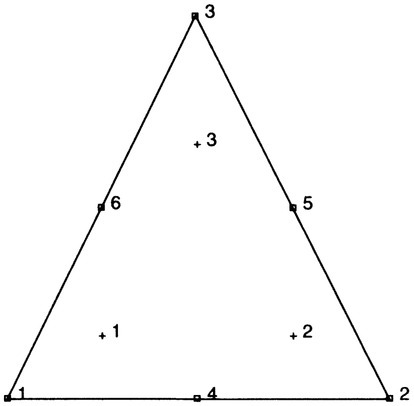


Figure 3-199 Integration Points of Six-Node, 2D Element

Output Of Stresses

Output of stresses is the same as **Output of Strains**.

Transformation

Only in z-r plane.

Tying

Use subroutine UFORMS.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-199. This is the usual option for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 132. See Element 132 for a description of the conventions used for entering the flux and film data for this element.

Element 130

Three-Dimensional Ten-Node Tetrahedron, Herrmann Formulation

This is a second-order isoparametric tetrahedron element written for incompressible three-dimensional applications. This element uses biquadratic interpolation functions to represent coordinates and displacements. This allows for an accurate representation of the strain field. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using three integration points.

This element is designed to be used for incompressible elasticity only. It may be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element may be used in conjunction with other elements such as type 127 when other material behavior, such as plasticity, must be represented.

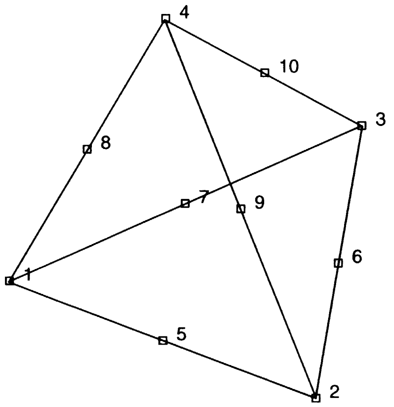


Figure 3-200 Form of Element 130

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counter clockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat) so that the user need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using 4 points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element, with the first point closest to the first node of the element (see Figure 3-201).

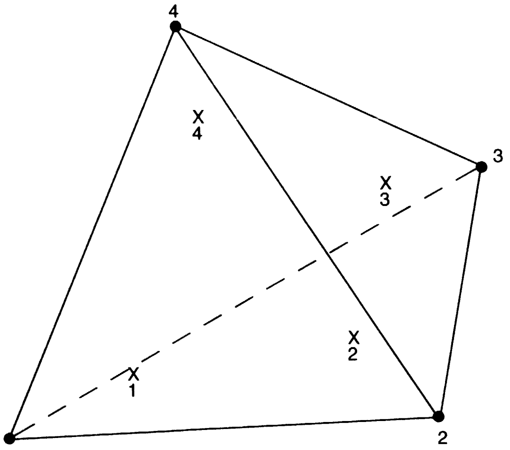


Figure 3-201 Element 130 Integration Plane

Quick Reference

Type 130

Ten-nodes, isoparametric arbitrary distorted tetrahedron.

Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-200.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

At each corner:

1 = u = global x-direction displacement
2 = v = global y-direction displacement
3 = w = global z-direction displacement

At corner nodes only:

4 = σ_{kk}/E = mean pressure variable (for Herrmann)
= -p = negative hydrostatic pressure (for Mooney)

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.

Load Type	Description
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

The subroutine FORCEM will be called once per integration point when flagged. The magnitude of load defined by DIST LOADS will be ignored and the FORCEM value will be used instead.

For nonuniform body force, force values must be provided for the four integration points.

For nonuniform surface pressure, force values need only be supplied for the three integration points on the face of application.

Output of Strains

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}
7 = σ_{kk}/E = mean pressure variable (for Herrmann)
= $-p$ = negative hydrostatic pressure (for Mooney)

Output of Stresses

Same as for **Output of Strains**.

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid or four Gaussian integration points (see Figure 3-201).

Notes: A large bandwidth results in a lengthy central processing time.

The user should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous cross changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of inelastic work specified with type 101.

Element 131

Planar, Six-Node Distorted Triangle (Heat Transfer Element)

This is a second-order isoparametric two-dimensional heat transfer triangular element. Temperatures and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-202). The interpolation function is such that each edge has parabolic variation along itself. The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems. The conductivity of this element is formed using three-point integration.

The connectivity ordering is shown in Figure 3-202. Note that the basic numbering is counterclockwise in the (x-y) plane.

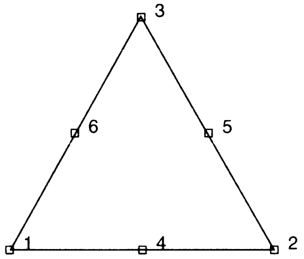


Figure 3-202 Nodes of Six-Node, 2D Element

Quick Reference

Type 131

Second-order isoparametric distorted heat transfer triangle.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-202.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatics)

Fluxes

Surface fluxes are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform flux on 1-4-2 face.
1	Nonuniform flux on 1-4-2 face.
4	Uniform flux on 2-5-3 face.
5	Nonuniform flux on 2-5-3 face.
8	Uniform flux on 3-6-1 face.
9	Nonuniform flux on 3-6-1 face.
12	Uniform body flux per unit volume.

For all nonuniform fluxes, the load magnitude is supplied via user subroutine FLUX.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as **Fluxes**.

Charge

Same specifications as **Fluxes**.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-203. This is the usual option for a second-order element.

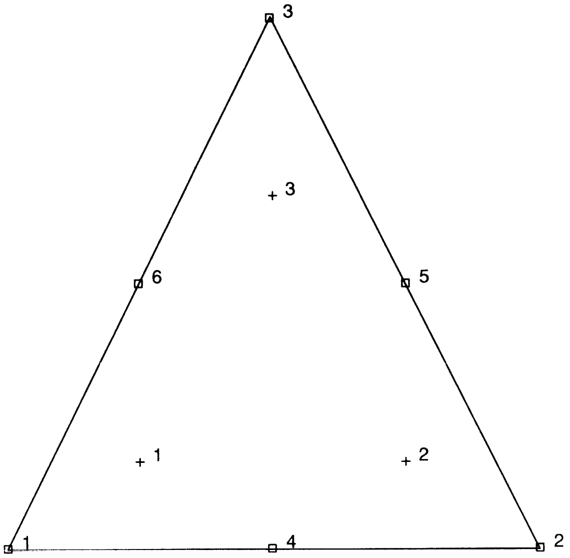


Figure 3-203 Integration Points of Six-Node, 2D Element

Element 132

Axisymmetric, Six-Node Distorted Triangle (Heat Transfer Element)

This is a second-order isoparametric two-dimensional axisymmetric triangular element. Temperatures and position (coordinates) within the heat transfer element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-204). The interpolation function is such that each edge has parabolic variation along itself. The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems. The conductivity of this element is formed using three-point integration.

The connectivity ordering is shown in Figure 3-204. Note that the basic numbering is counterclockwise in the (z-r) plane.

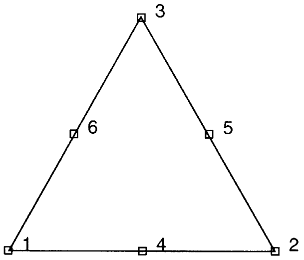


Figure 3-204 Nodes of Six-Node, Axisymmetric Element

Quick Reference

Type 132

Second-order isoparametric distorted axisymmetric heat transfer triangle.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-204.

Geometry

Not required.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatics)

Fluxes

Surface fluxes are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform flux on 1-4-2 face.
1	Nonuniform flux on 1-4-2 face.
4	Uniform flux on 2-5-3 face.
5	Nonuniform flux on 2-5-3 face.
8	Uniform flux on 3-6-1 face.
9	Nonuniform flux on 3-6-1 face.
12	Uniform body flux per unit volume.

For all nonuniform loads, the load magnitude is supplied via user subroutine FLUX.

Films

Same specification as **Fluxes**.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specification as **Fluxes**.

Charge

Same specification as **Fluxes**.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-205. This is the usual option for a second-order element.

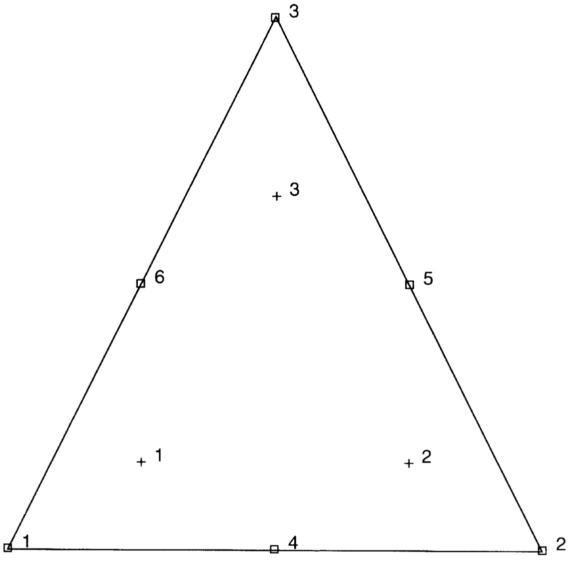


Figure 3-205 Integration Points of Six-Node, Axisymmetric Element

Element 133

Three-Dimensional Ten-Node Tetrahedron (Heat Transfer Element)

This element is a second-order isoparametric three-dimensional heat transfer tetrahedron. Each edge forms a parabola, so that four nodes define the corners of the element and a further six nodes define the position of the "midpoint" of each edge (Figure 3-206). The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems. The conductivity of this element is formed using four-point integration.

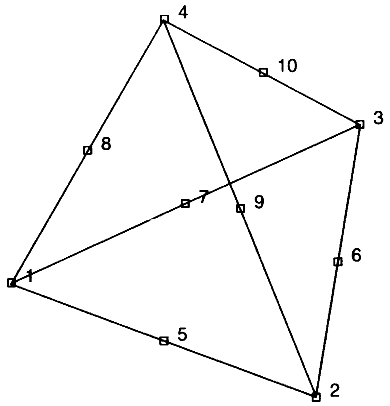


Figure 3-206 Form of Element 133

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat), so that the user need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element, with the first point closest to the first node of the element (see Figure 3-207).

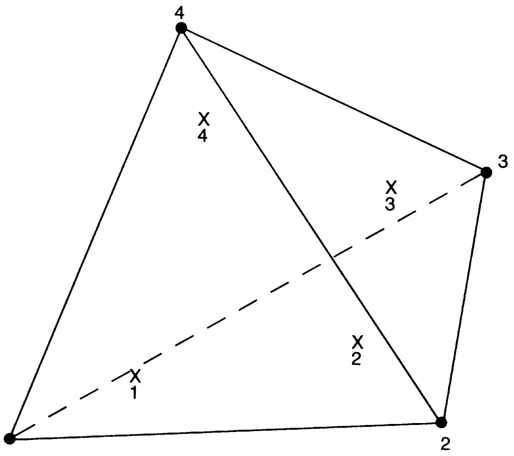


Figure 3-207 1-2 Element 133 Integration Plane

Quick Reference

Type 133

Ten-nodes, isoparametric arbitrary distorted heat transfer tetrahedron.

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-206.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential, (electrostatic)

Distributed Fluxes

Distributed fluxes chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform flux on 1-2-3 face.
1	Nonuniform flux on 1-2-3 face.
2	Uniform flux on 1-2-4 face.
3	Nonuniform flux on 1-2-4 face.
4	Uniform flux on 2-3-4 face.
5	Nonuniform flux on 2-3-4 face.
6	Uniform flux on 1-3-4 face.
7	Nonuniform flux on 1-3-4 face.
8	Uniform body flux per unit volume.
9	Nonuniform body flux per unit volume.

The subroutine FLUX will be called once per integration point when flagged. The magnitude of load defined by DIST FLUXES will be ignored and the FLUX value will be used instead.

For nonuniform body flux, flux values must be provided for the four integration points.

For nonuniform surface pressure, flux values need only be supplied for the three integration points on the face of application.

Films

Same specification as **Distributed Fluxes.**

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as **Distributed Fluxes**.

Charges

Same specifications as **Distributed Fluxes**.

Output Points

Centroid or four Gaussian integration points (see Figure 3-207).

Notes: A large bandwidth results in a lengthy central processing time.

The user should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Element 134

Three-Dimensional Four-Node Tetrahedron

This element is a linear isoparametric three-dimensional tetrahedron. (Figure 3-208). As this element uses linear interpolation functions, the strains are constant throughout the element. This results in a poor representation of shear behavior. A fine mesh is required to obtain an accurate solution. Note that this element is known to give poor results for plasticity or incompressible behavior. This element should only be used for linear elasticity. If tetrahedral are required, use element 127 for nonlinear analysis. The element is integrated numerically using one point at the centroid of the element.

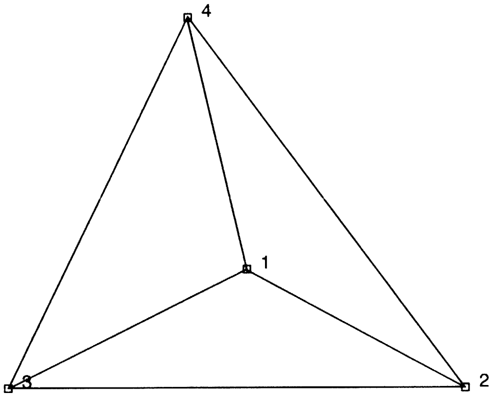


Figure 3-208 Form of Element 134

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of four nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat or a CAD program) so that the user need not be concerned with the node numbering scheme.

Quick Reference

Type 134

Four-nodes, isoparametric arbitrary distorted tetrahedron.

Connectivity

Four nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-208.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.

Load Type	Description
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

The subroutine FORCEM will be called once per integration point when flagged. The magnitude of load defined by DIST LOADS will be ignored and the FORCEM value will be used instead.

For nonuniform body force, force values must be provided for the one integration point.

For nonuniform surface pressure, force values need only be supplied for the one integration point on the face of application.

Output of Strains

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \gamma_{yz}$
$6 = \gamma_{zx}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Three global degrees of freedom may be transformed to local degrees of freedom.

Tying

Use subroutine UFORMS.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Notes: A large bandwidth results in a lengthy central processing time.

The user should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 135. See Element 135 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 135

Three-Dimensional Four-Node Tetrahedron (Heat Transfer Element)

This element is a linear isoparametric three-dimensional tetrahedron for heat transfer applications (see Figure 3-209). As this element uses linear interpolation functions, the thermal gradients are constant throughout the element. A fine mesh is required to obtain an accurate solution. The element is integrated numerically using one point at the centroid of the element.

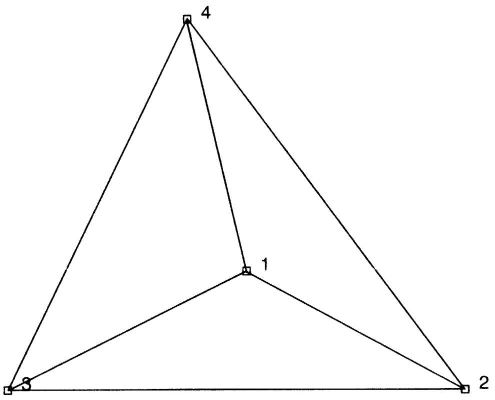


Figure 3-209 Form of Element 135

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of four nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat or a CAD program) so that the user need not be concerned with the node numbering scheme.

Quick Reference

Type 135

Four-nodes, isoparametric arbitrary heat transfer tetrahedron.

Connectivity

Four nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-209.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Distributed Fluxes

Distributed fluxes chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform flux on 1-2-3 face.
1	Nonuniform flux on 1-2-3 face.
2	Uniform flux on 1-2-4 face.
3	Nonuniform flux on 1-2-4 face.
4	Uniform flux on 2-3-4 face.
5	Nonuniform flux on 2-3-4 face.
6	Uniform flux on 1-3-4 face.
7	Nonuniform flux on 1-3-4 face.
8	Uniform body flux per unit volume.
9	Nonuniform body flux per unit volume.

The subroutine FLUX will be called once per integration point when flagged. The magnitude of load defined by DIST FLUXES will be ignored and the FLUX value will be used instead.

For nonuniform body flux, flux values must be provided for the one integration point.

For nonuniform surface pressure, flux values need only be supplied for the one integration point on the face of application.

Films

Same specification as **Distributed Fluxes**.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as **Distributed Fluxes**.

Charges

Same specification as **Distributed Fluxes**.

Output Points

Centroid.

Notes: A large bandwidth results in a lengthy central processing time.

The user should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

■ Element 136

Six-node Wedge

Not available at this time.

Element 137

Six-node Wedge Heat Transfer

Not available at this time.

Element 138

Bilinear Thin-Triangular Shell Element

This is a three-node, thin-triangular shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion. All constitutive relations may be used with this element.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system.

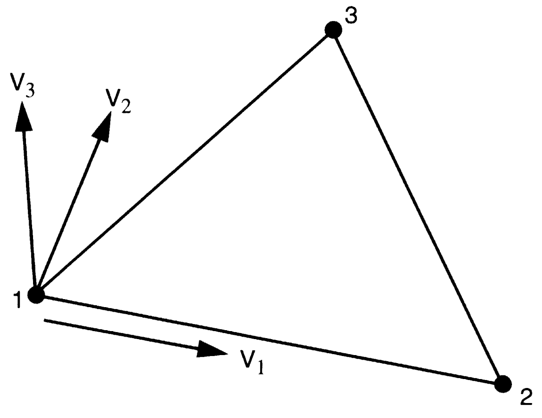


Figure 3-210 Form of Element 138

There are three integration points in the plane of the element.

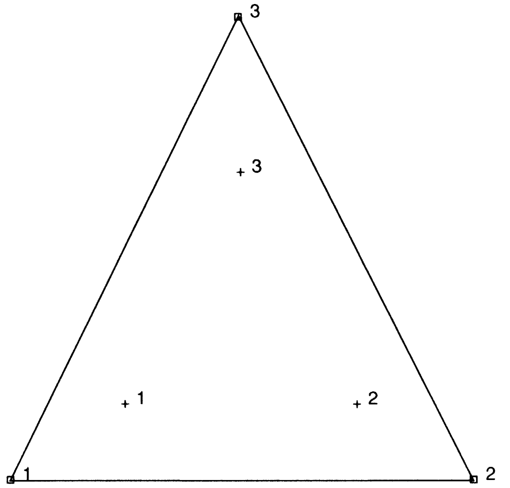


Figure 3-211 Integration Points of Three-Node Shell Element

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

Quick Reference

Type 138

Bilinear, three-node thin shell element.

Connectivity

Three nodes per element.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, and third nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), and third (EGEOM3), geometry data fields, respectively. If EGEOM2=EGEOM3=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_y = rotation about global x-axis
5 = ϕ_z = rotation about global z-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude given in user subroutine FORCEM.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in user subroutine FORCEM.
11	Uniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge.
12	Uniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge.
13	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge; magnitude given in user subroutine FORCEM.
14	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge; magnitude given in user subroutine FORCEM.
15	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge; magnitude and direction given in user subroutine FORCEM.
21	Uniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge.

Load Type	Description
22	Uniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge.
23	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge; magnitude given in user subroutine FORCEM.
24	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge; magnitude given in user subroutine FORCEM.
25	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge; magnitude and direction given in user subroutine FORCEM.
31	Uniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge.
32	Uniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge.
33	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge; magnitude given in user subroutine FORCEM.
34	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge; magnitude given in user subroutine FORCEM.
35	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge; magnitude and direction given in user subroutine FORCEM.
100	Centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity load in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Point Loads

Point loads and moments may also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches: ε_{11} ε_{22} ε_{12}
Middle surface curvatures: κ_{11} κ_{22} κ_{12}

in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12} in local ($\underline{V}_1$, $\underline{V}_2$, $\underline{V}_3$) system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at corner nodes may be transformed to local direction.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, the user has to select his load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.q., thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 50. See Element 50 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 139

Bilinear Thin-Shell Element

This is a four-node, thin-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion. All constitutive relations may be used with this element.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface will form a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, V_1, V_2, and V_3, which for the centroid are defined in the following way (see Figure 3-212).

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} / \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s| \quad \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}$$

$$V_2 = \bar{s} - \bar{d}$$

and

$$V_3 = V_1 \times V_2$$

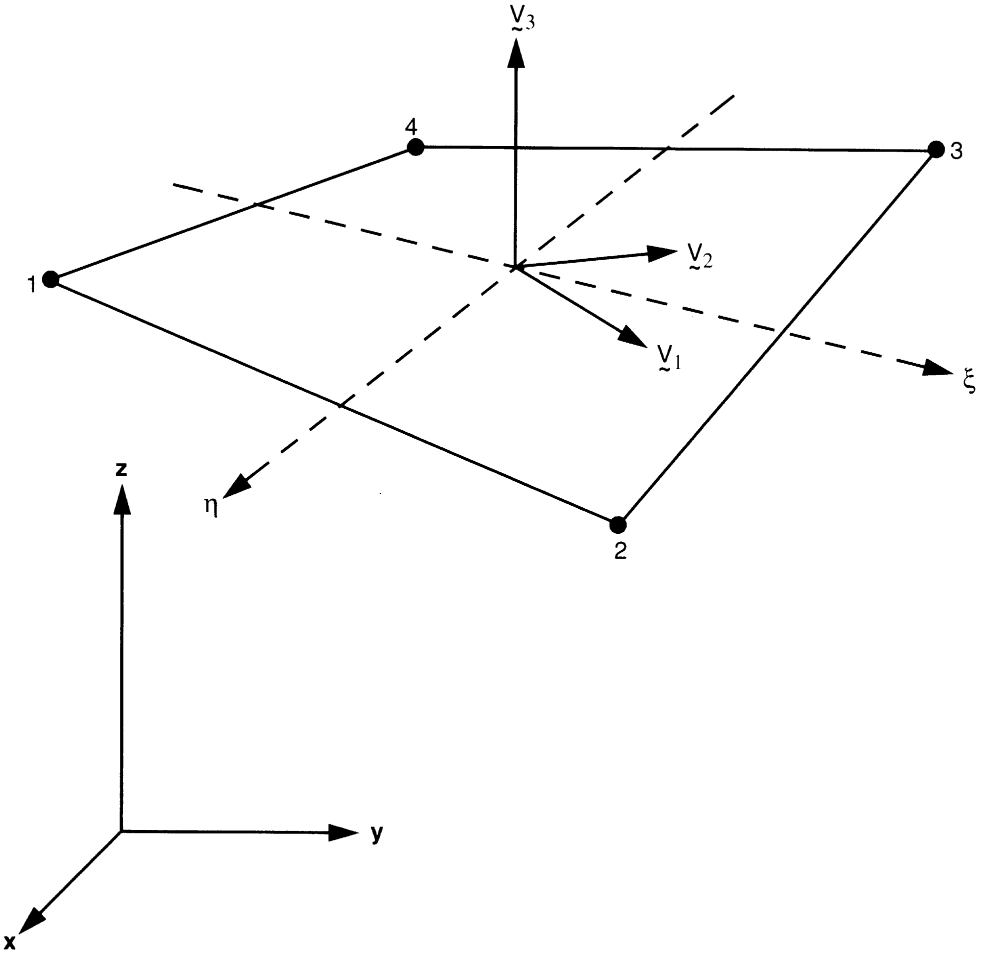


Figure 3-212 Form of Element 139

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

Quick Reference

Type 139

Bilinear, four-node thin shell element.

Connectivity

Four nodes per element. The element may be collapsed into a triangle in which case the element is identical to element type 138.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_y = rotation about global x-axis
5 = ϕ_z = rotation about global z-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude given in user subroutine FORCEM.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in user subroutine FORCEM.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in FORCEM on 1-2 edge.
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in FORCEM on 2-3 edge.
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction given in FORCEM on 3-4 edge.

Load Type	Description
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in FORCEM on 4-1 edge.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches: $\varepsilon_{11}\ \varepsilon_{22}\ \varepsilon_{12}$
Middle surface curvatures: $\kappa_{11}\ \kappa_{22}\ \kappa_{12}$

in local ($\underline{V}_1$, $\underline{V}_2$, $\underline{V}_3$) system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12} in local ($\underline{V}_1$, $\underline{V}_2$, $\underline{V}_3$) system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at corner nodes may be transformed to local direction.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, the user has to select his load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.q., thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 140

Bilinear Thick-Shell Element with Reduced Integration

This is a four-node, thick-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. In this way, a very efficient and simple element is obtained which exhibits correct behavior in the limiting case of thin shells. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is sensitive to distortion. All constitutive relations may be used with this element. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface will form a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, V_1, V_2, and V_3, which for the centroid are defined in the following way (see Figure 3-213).

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s/\sqrt{2}|s| \quad \bar{d} = d/\sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}$$

$$V_2 = \bar{s} - \bar{d}$$

and

$$V_3 = V_1 \times V_2$$

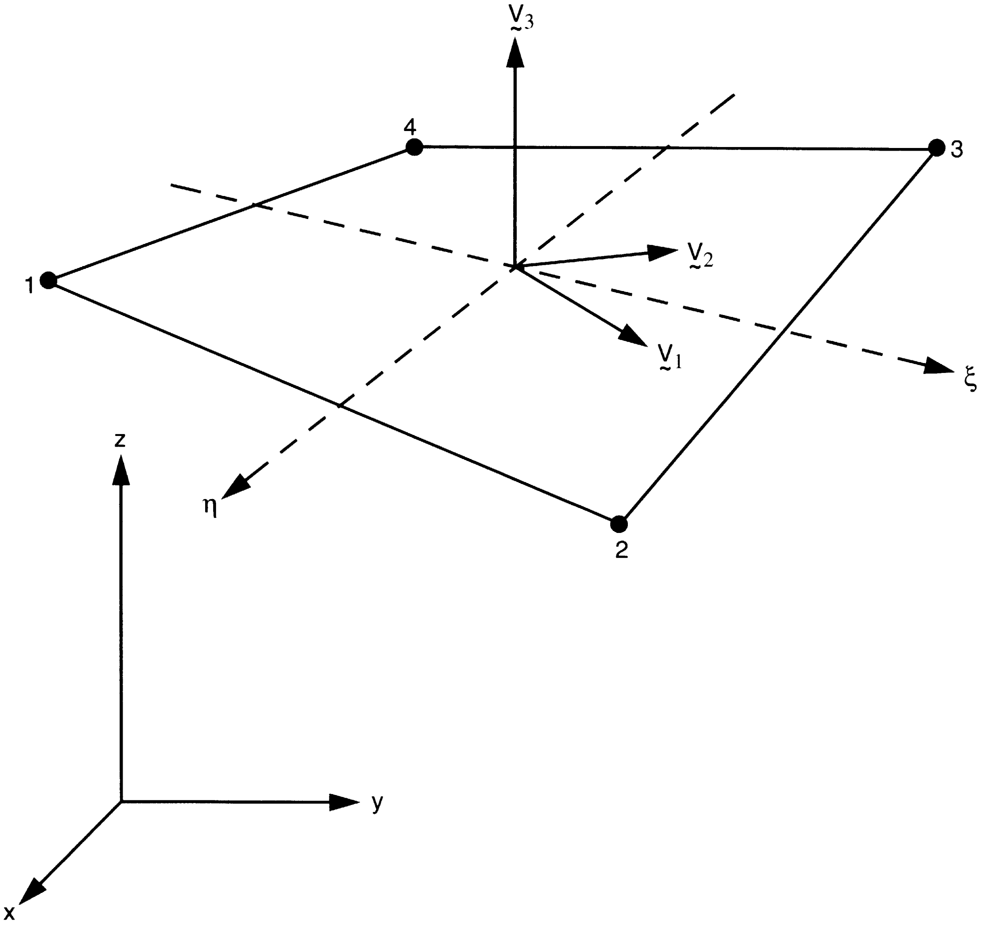


Figure 3-213 Form of Element 140

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

Quick Reference

Type 140

Bilinear, four-node shell element including transverse shear effects using reduced integration with hourglass correction.

Connectivity

Four nodes per element. The element should not be collapsed into a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2=EGEOM3=EGEOM4=0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_y = rotation about global x-axis
5 = ϕ_z = rotation about global z-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction, magnitude given in user subroutine FORCEM.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude given in user subroutine FORCEM.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in user subroutine FORCEM.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in FORCEM on 1-2 edge.
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in FORCEM on 2-3 edge.
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 3-4 edge.

Load Type	Description
33	Nonuniform edge load magnitude and direction given in FORCEM on 3-4 edge.
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude given in FORCEM in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in FORCEM on 4-1 edge.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in ROTATION A option.

Point Loads

Point loads and moments may also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches: ε_{11} ε_{22} ε_{12}
Middle surface curvatures: κ_{11} κ_{22} κ_{12}
Transverse shear strains: γ_{23} γ_{31}

in local ($\underline{V}_1$, $\underline{V}_2$, $\underline{V}_3$) system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local ($\underline{V}_1$, $\underline{V}_2$, $\underline{V}_3$) system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at corner nodes may be transformed to local direction.

Tying

Use subroutine UFORMS.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, the user has to select his load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.q., thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 141. This is not available in the K7 release.

Design Variables

The thickness can be considered as a design variable.

Element 141

Heat Transfer Shell

Not available at this time.

■ Element 142

Eight-Node Axisymmetric Rebar Element with Twist

This element is similar to element 48, but is written for axisymmetric applications with torsional strains. It is a hollow, isoparametric 8-node quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element with twist (e.g., element 66 or 67) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by the user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-214), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

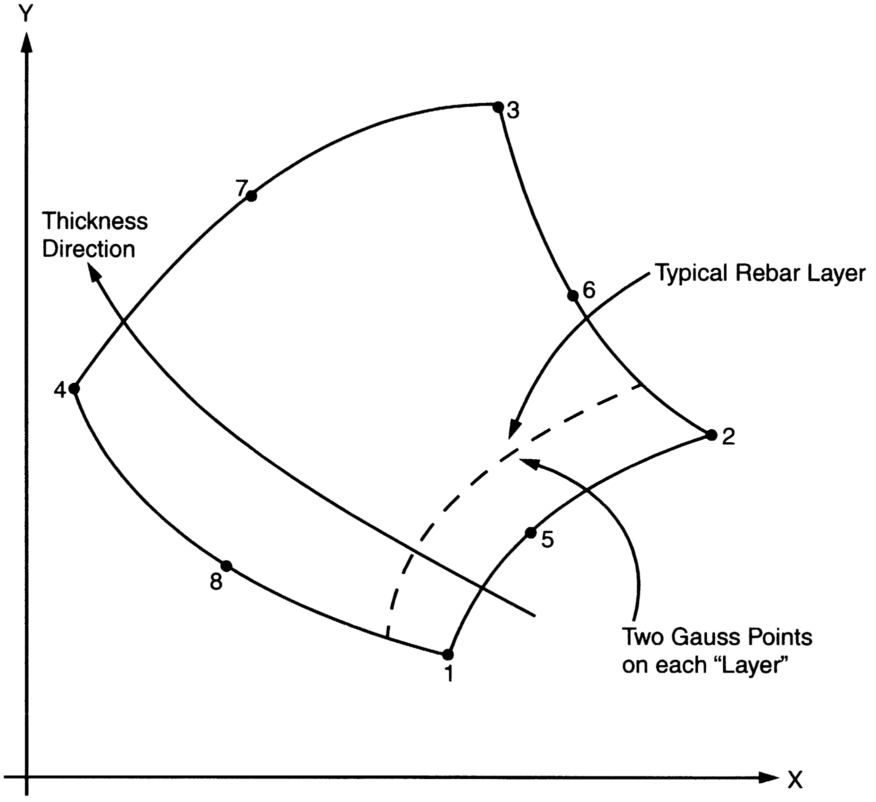


Figure 3-214 Eight-Node Rebar Element Conventions

Quick Reference

Type 142

Eight-node, isoparametric rebar element with torsional strains.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 66 or 67.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, enter either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-214).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates in z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r
3 - θ

Tractions

Point loads may be applied at the nodes but no distributed loads are available. Distributed loads are applied only to corresponding 8-node axisymmetric element with twist (e.g., element types 66 or 67).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformations

Any local set (u,v) may be used in the (z-r) plane at any node.

Special Considerations

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 143

Four-Node Plane Strain Rebar Element

This element is isoparametric, plane strain, 4-node hollow quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 4-node plane strain continuum element (e.g., element 11 or 80) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by the user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-215), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

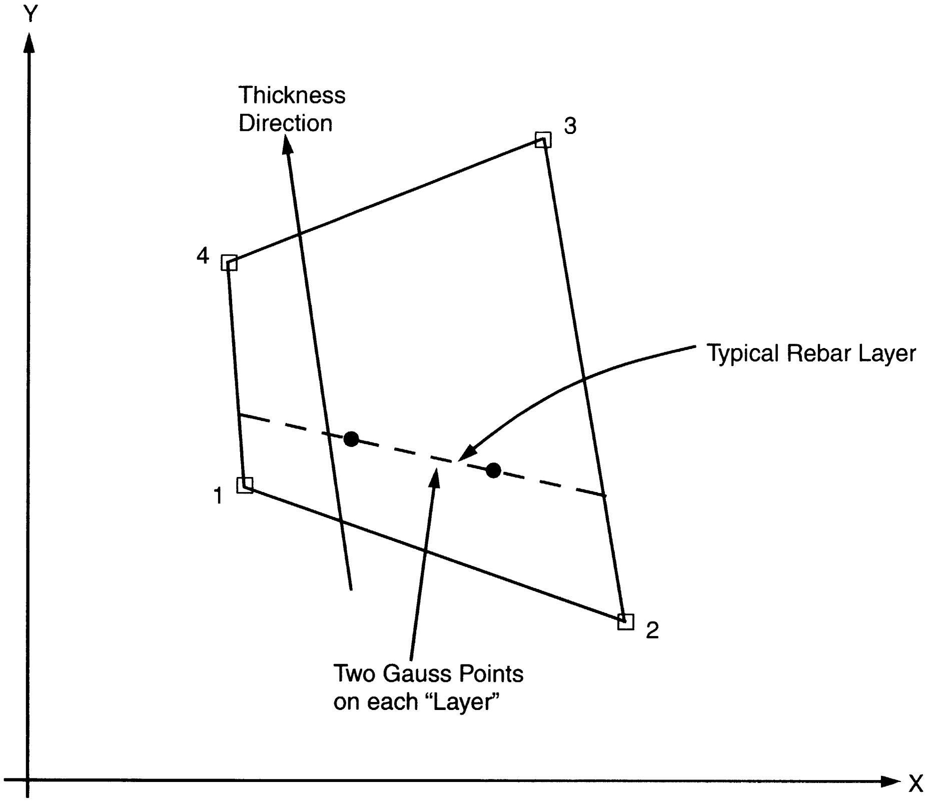


Figure 3-215 Four-Node Rebar Element Conventions

Quick Reference

Type 143

Four-node, isoparametric rebar element to be used with 4-node plane strain continuum element.

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 11 or 80.

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the "thickness" concept associated with rebar layers. If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, enter either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-215).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node plane strain elements (e.g., element types 11 or 80).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Any local set (u,v) may be used at any node.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

■ Element 144

Four-Node Axisymmetric Rebar Element

This element is similar to element 143, but is written for axisymmetric conditions. It is a hollow, isoparametric 4-node quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 4-node axisymmetric continuum element (e.g., element 10 or 82) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by the user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-216), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

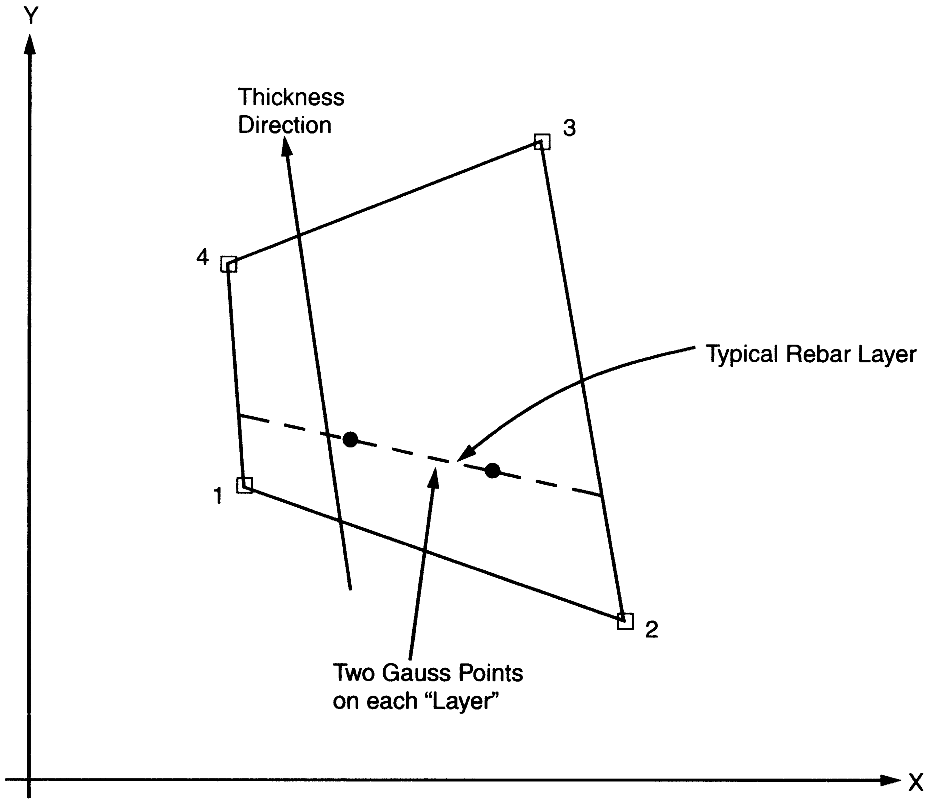


Figure 3-216 Four-Node Rebar Element Conventions

Quick Reference

Type 144

Four-node, isoparametric rebar element to be used with 4-node axisymmetric continuum element.

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 10 or 82.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, enter either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-216).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node axisymmetric elements (e.g., element types 10 or 82).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Any local set (u,v) may be used in the (z-r) plane at any node.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 145

Four-Node Axisymmetric Rebar Element with Twist

This element is similar to element 144, but is written for axisymmetric applications with torsional strains. It is a hollow, isoparametric 4-node quadrilateral in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 4-node axisymmetric continuum element with twist (e.g., element 20 or 83) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by the user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-217), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

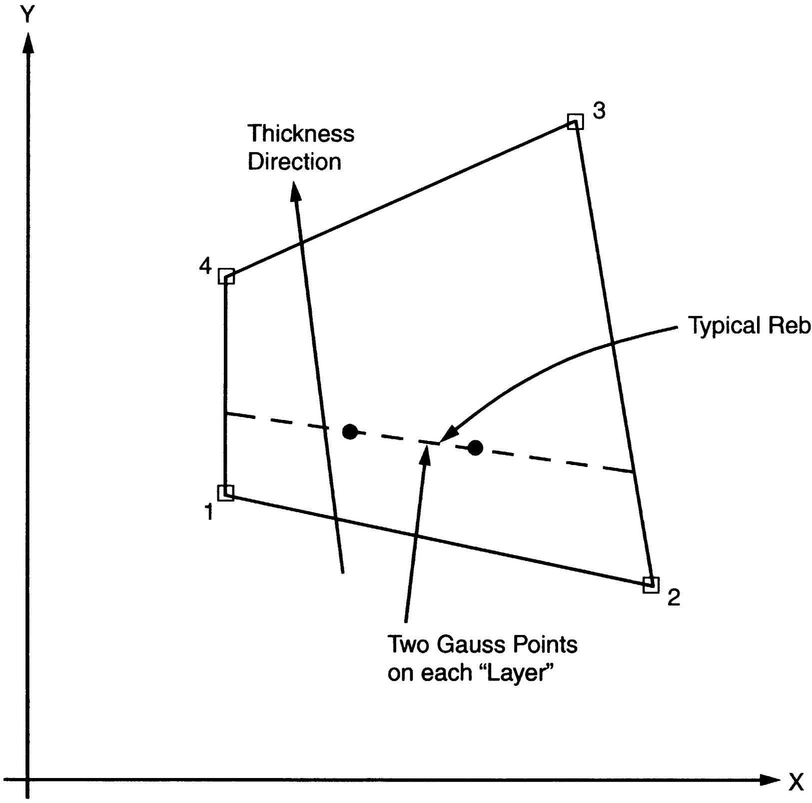


Figure 3-217 Four-Node Rebar Element Conventions

Quick Reference

Type 145

Four-node, isoparametric rebar element with torsional strains.

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 20 or 83.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, enter either 1 or 2:

1. Rebar layers are similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element (see Figure 3-217).
2. Rebar layers are similar to the 1, 4 and 2, 3 edges of the element, the "thickness" direction is from the 1, 4 edge to 2, 3 edge of the element.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r
3 - θ

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node axisymmetric elements with twist (e.g., element types 20 or 83).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Any local set (u,v) may be used in the (z-r) plane at any node.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 146

Three-Dimensional 8-Node Rebar Element

This element is an isoparametric, three-dimensional, 8-node empty brick in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node brick continuum element (e.g., element types 7 or 84) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by the user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element faces (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element faces to its opposite one. For instance (see Figure 3-218), if the layer is similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-218). At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

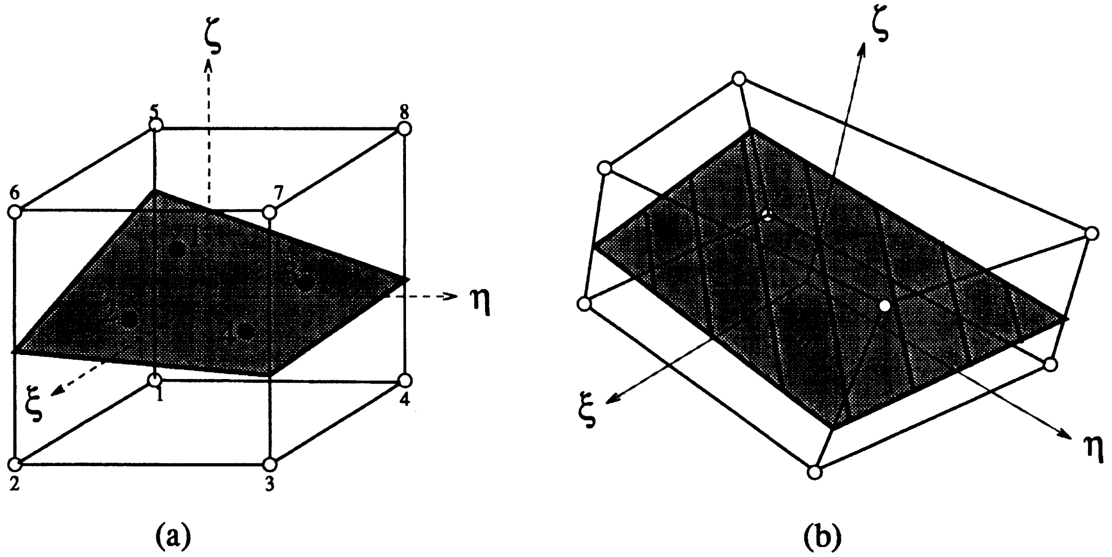


Figure 3-218 Typical Layer in 8-Node 3D Rebar Element:
(a) Parent Domain; (b) Mapping of Parent Domain

Quick Reference

Type 146

Eight-node, isoparametric rebar element to be used with 8-node brick continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element types 7 or 84.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

The isoparametric direction of rebar layers is defined in the third field, enter either 1, 2, or 3:

1. Rebar layers are similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element (see Figure 3-218).
2. Rebar layers are similar to the 1, 4, 8, 5 and 2, 3, 7, 6 faces of the element, the "thickness" direction is from the 1, 4, 8, 5 face to 2, 3, 7, 6 face of the element.

3. Rebar layers are similar to the 2, 1, 5, 6 and 3, 4, 8, 7 faces of the element, the "thickness" direction is from the 2, 1, 5, 6 face to 3, 4, 8, 7, face of the element.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node brick elements (e.g., element types 7 or 84).

Output Of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Any local set (u,v,w) may be used at any node.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

■ Element 147

Four-Node Rebar Membrane

This element is hollow, isoparametric 4-node membrane in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 4-node membrane (element 18) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The rebar layers are assumed to be placed on the same spatial position as that of the element (although the rebar direction is arbitrary and the "thickness" of the layers may be different). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points. At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

Quick Reference

Type 147

Four-node, isoparametric rebar membrane to be used with 4-node membrane (element 18).

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 18.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node membrane (element type 18).

Output Of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Nodal degrees of freedom may be transformed to local degrees of freedom.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

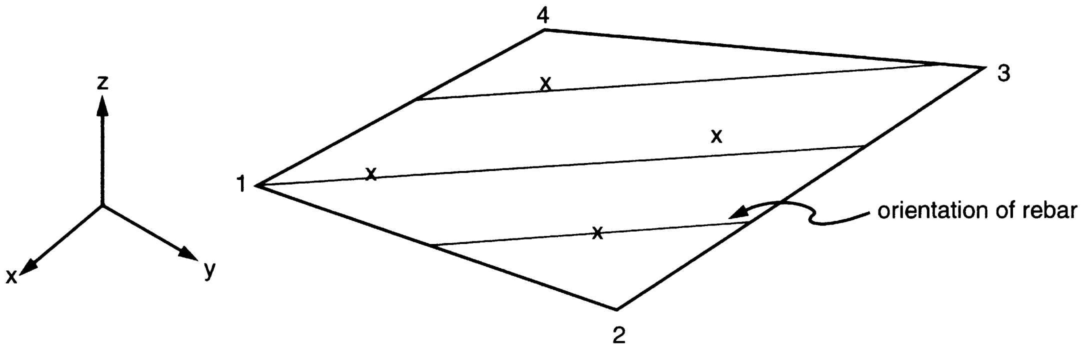


Figure 3-219 One Layer of Rebar Membrane Element

Element 148

Eight-Node Rebar Membrane

This element is hollow, isoparametric 8-node membrane in which the user may place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node membrane (element 30) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories may be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via user subroutine REBAR.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by user via REBAR option or, if user subroutine REBAR is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The rebar layers are assumed to be placed on the same spatial position as that of the element (although the rebar direction is arbitrary and the "thickness" of the layers may be different). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points. At each such integration point on each layer, the user must input via either the REBAR option or the user subroutine REBAR the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Volume C for REBAR option or Volume D for user subroutine REBAR.

Quick Reference

Type 148

Eight-node, isoparametric rebar membrane to be used with 8-node membrane (element 30).

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 30.

Geometry

If the position, equivalent thickness, and orientation of the rebars are input via the user subroutine REBAR, the number of rebar layers is input in the second field as a floating point number. Maximum is five.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads may be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node membrane (element type 30).

Output Of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

Transformation

Nodal degrees of freedom may be transformed to local degrees of freedom.

Special Consideration

Either REBAR option or user subroutine REBAR is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

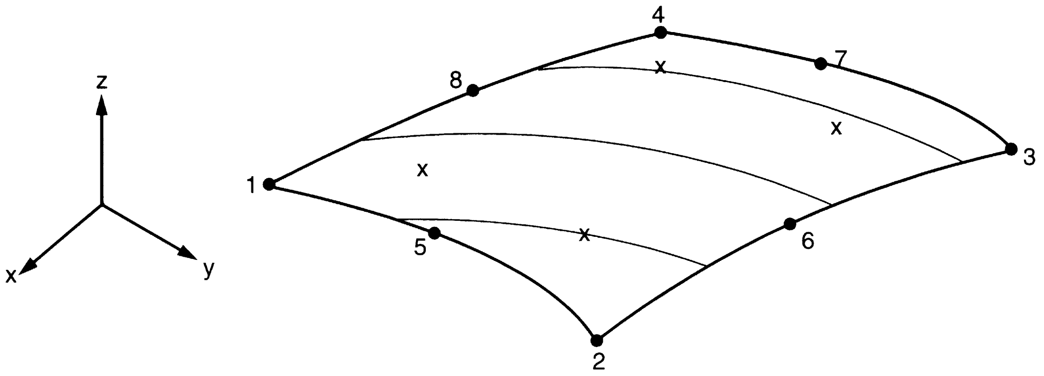


Figure 3-220 One Layer of Rebar Membrane Element

MARC
Analysis
Research
Corporation

Document Title: **MARC Volume B: Element Library K7**
Part Number: RF-3002-07
Revision Date: August, 1997

CALIFORNIA - Palo Alto
MARC Corporate Headquarters
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel: 1 650 329 6800
Fax: 1 650 323 5892
Email: support@marc.com

CALIFORNIA - San Diego
MARC Analysis Research Corp.
4330 LaJolla Village Drive, Suite 320
San Diego, CA 92122
Tel: 1 619 658 9588
Fax: 1 619 587 8710

CONNECTICUT - E. Berlin
MARC Analysis Research Corp.
1224 Mill Street
E. Berlin, CT 06023
Tel: 1 860 828-2055
Fax: 1 860 828 2031

MICHIGAN - Ann Arbor
MARC Analysis Research Corp.
24 Frank Lloyd Wright Drive
Ann Arbor, MI 48106-0523
Tel: 1 313 998 0540
Fax: 1 313 998 0542

CZECH REPUBLIC
MARC Overseas, Inc.
Podolska 50
147 00 Praha 4
Czech Republic
Tel: 011 420 (2) 61 21 41 23
Mobil: 011 420 (602) 21 23 26
Fax: 011 420 (2) 61 21 41 23

FRANCE
MARC France S.A.
30, Rue de Rocroy
94100 Saint-Maur Des Fossés (near Paris)
France
Tel: 011 33 (1) 41 81 1 0 94
Fax: 011 33 (1) 41 81 1 0 95

GERMANY - Munich
MARC Software Deutschland GmbH
Ismaninger Str. 9
85609 Aschheim (bie Munich)
Germany
Tel: 011 49 (89) 904 50 33
Fax: 011 49 (89) 903 06 76
Email: system@marc.co.de

GERMANY - Hannover
MARC Software Deutschland Gmbh
Alte Dohrener Str. 66
30173 Hannover, Germany
Tel: 011 49 (511) 980 5182
Fax: 011 49 (511) 980 5187

ITALY
Espri-MARC s.r.l.
16129 Viale Brigata Bisagno 2/10
16129 Genova
Italy
Tel: 011 39 (10) 585 949
Fax: 011 39 (10) 585 949

THE NETHERLANDS
MARC Analysis Research Corporation
Dublinstraat 32
2713 HS Zoetermeer
The Netherlands
Tel: 011 31 (79) 3510 411
Fax: 011 31 (79) 3517 560
Email: support@marc.nl

UNITED KINGDOM
MARC UK Ltd.
35, Shenley Pavilions
Chalkdell Drive
Shenley Wood
Milton Keynes, MK56LB
Tel: 011 44 (1908) 506 505
Fax: 011 44 (1908) 506 522

CHINA
MARC Overseas, Inc.
Bright Chang An Building, Suite 321
7# Jian Guo Men Nei Street
Beijing, China 100 005
Tel: 011 86 (10) 6510 2056
011 86 (10) 6510 2057
011 86 (10) 6510 2058
Fax: 011 86 (10) 6510 2053

JAPAN - Nagoya
Nippon MARC Co., Ltd.
Nagoya Nishiki Daiichi Seimei Bldg.
2F, 6-5 Nishiki 1-chome
Naka-ku, Nagoya, Aichi 460
Tel: 011 81 (52) 202 5661
Fax: 011 81 (52) 202-5665

JAPAN - Osaka
Nippon MARC Co., Ltd.
4F 2nd Kimi Building
2-11 Toyotucho Suita-shi
Osaka 564
Tel: 011 81 (6) 385 1101
Fax: 011 81 (6) 385-4343

JAPAN - Tokyo
(Asian Operations Headquarters)
Nippon MARC Co., Ltd.
Shinjuku Daiichi Seimei Building
P.O. Box 5056
2-7-1 Nishi-Shinjuku, Shinjuku-ku
Tokyo 163
Tel: 011 81 (3) 3345 0181
Fax: 011 81 (3) 3345 1529
Email: system@marc.co.jp

KOREA
Dong Kyung Bldg. 7FL
824-19 Yuksam-Dong
Kangnam-Ku
Seoul, Korea
Tel: 011 82 (2) 561 7543
Fax: 011 82 (2) 561 7767