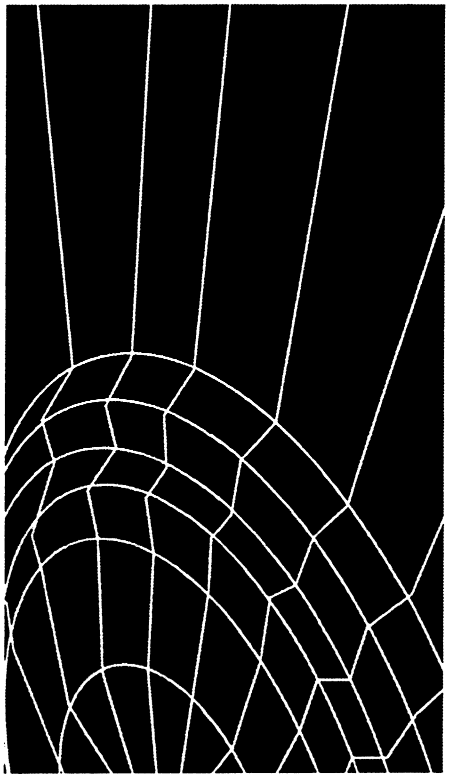
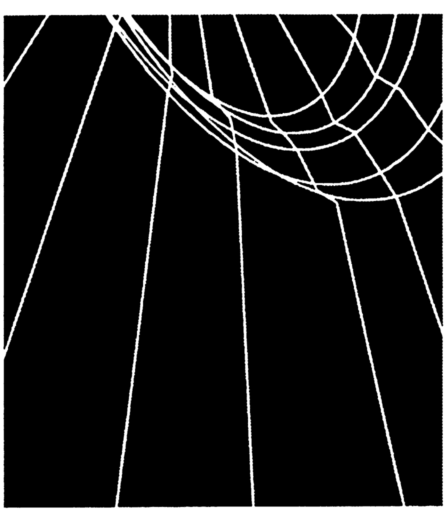
MARC
Analysis
Research
Corporation



MARC and Mentat

Release Notes



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

Phone: (415) 329-6800
FAX: (415) 323-5892

Document Title: **MARC and Mentat Release Notes**
Part Number: RL-3871-31
Revision Date: September, 1997

PROPRIETARY NOTICE

RESTRICTED RIGHTS NOTICE

TRADEMARKS

MARC
Analysis
Research
Corporation

260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel.: 415 329-6800 Fax: 415 323-5892

September 29, 1997

Dear User:

MARC Analysis Research Corporation is pleased to provide you with the latest releases of the MARC software systems. MARC K7.1 and Mentat 3.1. These new releases have many exciting new features, providing increased functionality and improved reliability over previous releases. The new functionality is summarized in the attached pages. This release is also packaged differently from previous releases, and this book highlights the implications of these changes. They can be summarized as follows:

1. The releases are now being delivered on CD. Each CD contains MARC's available products on a particular hardware platform, specifically MARC K7.1, Mentat 3.1, MARC/Link-S, and MARC/Link-P products. As a result, if you upgrade your operating system, it will be less likely that you will require new electronic media from MARC.

2. MARC has adopted the industry standard FLEXlm license management system from Globetrotter Software, Inc. This will allow you to acquire floating licenses of the MARC software. For most users, who are only executing MARC products on one platform, the behavior will be the same as previous releases. For information regarding acquiring additional licenses, please contact your local MARC office. This manual contains the installation instructions for our products and the End-user FLEXlm document.

3. This release introduces major changes to the documentation. The *MARC Volumes A, B, C,* and *D* are provided in both printed form and electronic form to our commercial users. The electronic form may be used with either the Acrobat Reader or FrameViewer from Adobe Systems. Acrobat Reader is provided at no charge on the CD. FrameViewer must be purchased from Adobe Systems. In the current release of Acrobat Reader, the user can hypertext within one manual, while FrameViewer additionally allows you to hypertext between the manuals.

There is a new manual, *Mentat 3.1 - MARC K7.1 New Features* provided both in hard copy and electronic form. This manual covers the new capabilities of both products and provides procedures and cross-references to the new options.

MARC Volume A has been extensively rewritten to provide greater theoretical detail to the product.

A new version of *MARC Volume E: Demonstration Problems*, and *Mentat User Guide* will be available shortly.

We sincerely hope that these releases facilitate your engineering design and analysis. We look forward to your comments and working with you to further enhance our products to best serve your needs.

Sincerely yours,

Ted B. Wertheimer, Ph.D.
Vice President of Engineering

Jon Bishop
Mentat Product Manager

Sanjay Choudhry, Ph.D.
MARC Product Manager

MARC K7.1 and Mentat 3.1 Releases

MARC is very pleased to announce the release of the MARC product release K7.1 and the Mentat product release 3.1. The MARC vision embodies the idea to provide our users with the tools to simulate complex physical phenomena. We are committed to enrich our products so they provide an Integrated, Adaptive, Optimized Solution to Coupled Nonlinear Analysis. The new releases of our products continue the progression to this goal.These versions contain significant enhancements that broaden the analysis capabilities and improve engineering productivity. These versions have been extensively tested to insure the reliability of the software. There have been two major changes to the documentation as well. In addition to being on paper, the MARC manuals (Volumes A, B, C, and D) are now on-line and delivered in electronic form. There is also a new manual providing details of the new capabilities and their usage. A summary of the new capabilities is provided below.

MARC K7.1

Material Modeling

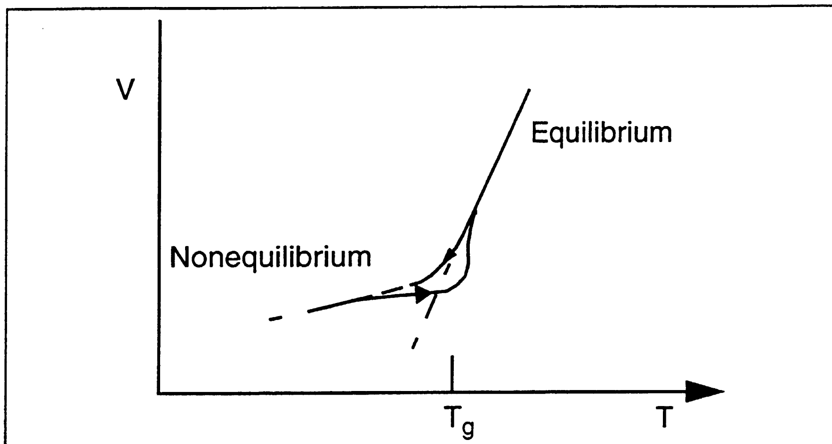
A new finite strain plasticity model has been developed for isotropic, kinematic, and combined hardening including temperature dependent behavior and thermal strains. The model will be available for plane stress, plane strain, axisymmetric, three-dimensional solid elements and membranes. This new procedure allows the user to take large increments of strain with greater accuracy and better convergence. It is of particular importance for problems where both the elastic and plastic strains are large. This model is based upon using the multiplicative decomposition of the deformation gradient, or F^eF^p model. The model assumes isotropic elasticity and the von Mises yield surface.

This new procedure also incorporates new technology to satisfy the incompressibility condition for plasticity. This allows the method to be used robustly for virtually all elements.

The Mooney and Ogden material models can now be used in an updated Lagrange framework for plane strain, axisymmetric, and three-dimensional solids. This new procedure incorporates new technology to satisfy the incompressibility condition, allowing these rubber models to be used with conventional displacement elements. In the K7.1 release, these rubber models in the updated Lagrange framework are rate independent. These models include temperature dependent properties and material damage. These models are advantageous for rubber analysis when used with rezoning or explicit dynamics. The new technology is also more computationally efficient. The user can implement new elasticity or plasticity models using new user subroutines upstretch or ufinite.

The hypoelastic material model has been expanded such that the user can obtain the deformation gradient (F), the rotation tensor (R), and the principal values of the stretch tensor (U). The user can implement advanced material models using the new user subroutine HYPELA2. User subroutine HYPELA is still available to insure compatibility with previous releases.

The study of glass through the materials glass transition temperature can be modeled using the new viscoelastic thermal expansion model based upon the Narayanaswamy model. The user can compute the structural relaxation or the volumetric viscoelastic behavior.



Hysteresis due to Thermal Recycling Captured by Narayanaswamy Mode

Rubber material degrade under both cyclic loading and with increased strain. This is particularly observed in filled rubber compounds under cyclic loading where the unloading curve is softer than the loading curve.

The rubber damage model has been enhanced such that both the Mullins effect (discontinuous damage) and the Miehe effect can be represented.

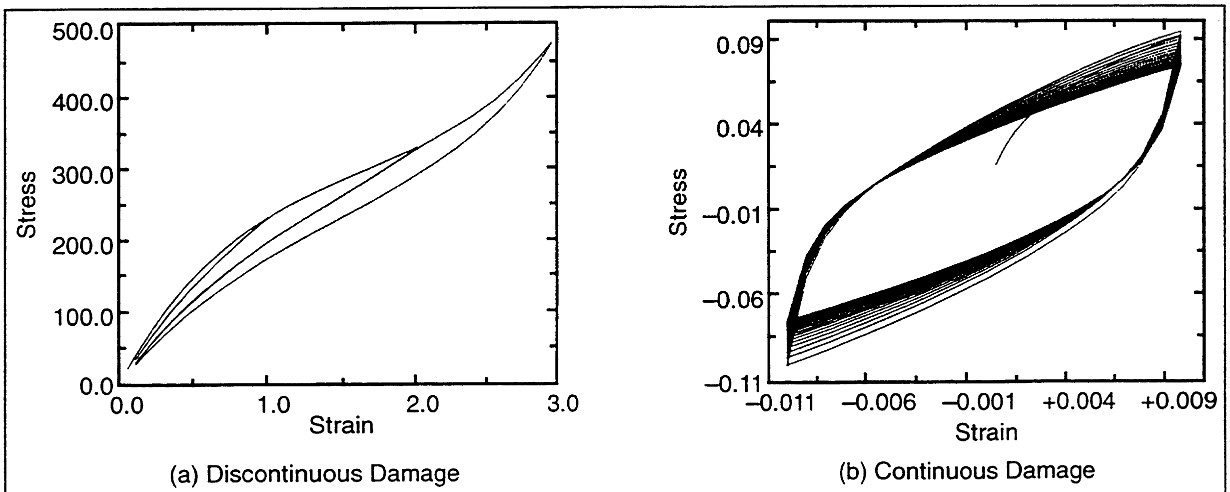


Figure 1

Contact

The contact capabilities in MARC have been extensively enhanced to increase the robustness of the procedures. A new friction model has been incorporated that represents perfect stick-slip. This model can be used in conjunction with either deformable-rigid contact or deformable-deformable contact. The model uses the nodal based approach which is applicable for continuum elements or shell elements. Extensive development has been done so that the model is robust and computationally efficient.

A new contact procedure has been developed, such that increment splitting is no longer necessary. In this procedure, the Newton-Raphson procedure is used to iterate to satisfy both the contact conditions and global equilibrium. This results in both a robust and computationally efficient procedure.

The deformable-deformable contact procedure has been enhanced such that the calculation of the normal to the surface is improved. For 2D, a spline curve is placed through the boundary nodes; while for 3D, a Coons surface is placed through the nodes. This allows a continuous normal to be calculated. This is especially useful for modeling the interface between circular shafts. A smooth behavior is obtained at a reduced computational cost.

The contact algorithm has been modified so that it is more robust for shell-to-shell contact problems.

Heat generated due to friction is now included in a coupled analysis when the nodal based friction model is activated.

The frictional forces are now written to the post file and can be viewed by Mentat.

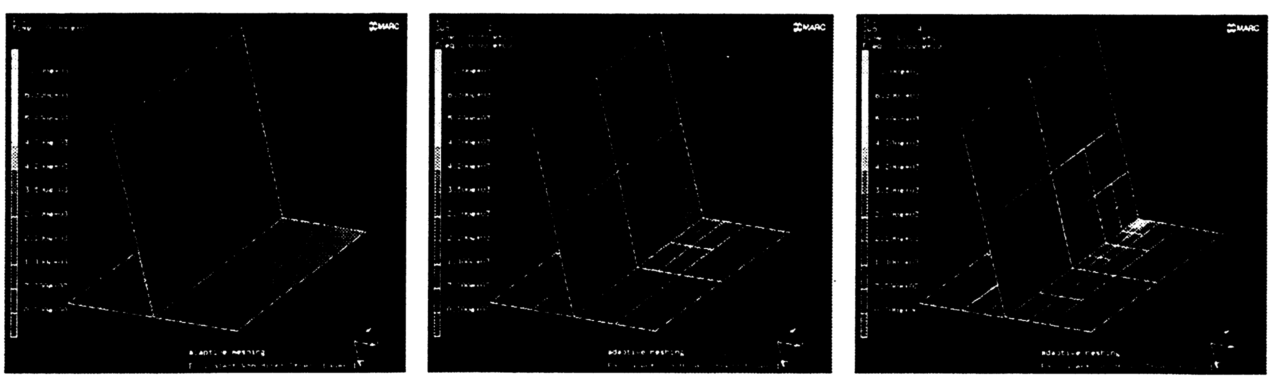
The separation criteria can now be based upon stresses as opposed to forces. For most problems, this is more robust and will result in a more accurate solution and a lower computational cost.

Rigid bodies can now be controlled in one of three ways: velocity controlled as in previous releases, position controlled, or force controlled. For position controlled bodies, one defines the location of the body as a function of time.

A new option has been added which facilitates the introduction of new contact bodies at later stages of the analysis. This is often useful for multistage forming analyses.

The geometrical description of the rigid bodies is now placed on the post file at the beginning of the analysis. If analytic (NURB) surfaces are used, this is the exact mathematical representation including the trimming curve information. The result is that the post file is smaller and the user has better visualization when using Mentat.

Adaptive Meshing



Adaptive Meshing of Intersecting Shells

Figure 2

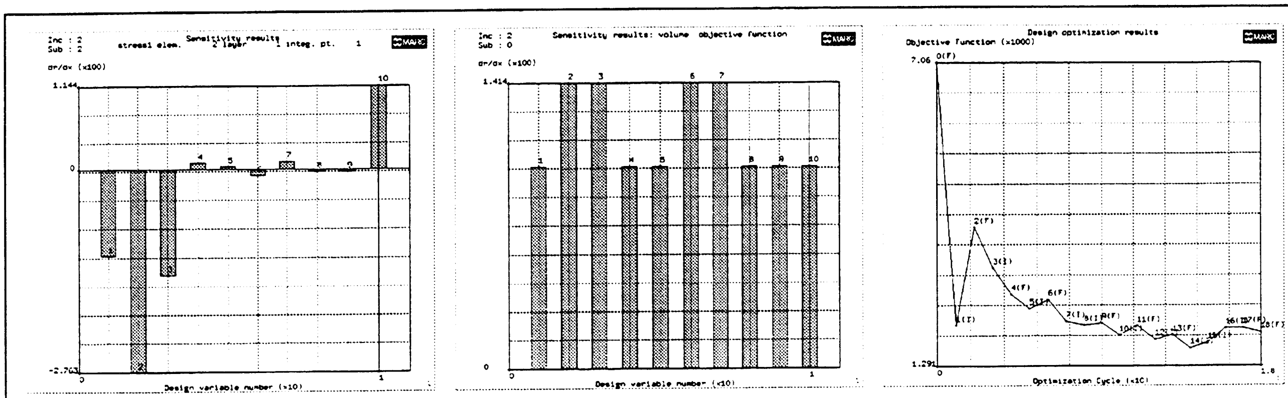
The adaptive meshing capabilities have been enhanced for shell elements. This is of particular importance when using intersecting shells or shells which are reinforced with beams.

Adaptive meshing will now allow refined elements to be merged back into the master element when used with box criteria. This reduces computational costs by eliminating elements in regions which are no longer significant.

Design Sensitivity and Optimization

The K7 release of MARC introduced a linear design sensitivity and optimization capability. This is the first step toward providing the user with a shape optimization capability for nonlinear analysis. The MARC procedure utilized innovative technology in that a minimum number of finite element analyses are required. While some procedures require 2^n analysis where n is the number of design variables or a random number of analyses, MARC performed n + 1 analyses per design iterations which lead to reduced computational costs. This will be particularly significant in future releases when nonlinear structural applications will be the focus of optimization.

In the K7.1 release, linear design optimization is available for resizing the design variables. In addition to the classic design variables (such as shell thickness, beam area, and beam moments of inertia), homogeneous elastic material properties (such as the Young's modulus, Poisson's ratio, and mass density) can be used. Futhermore, for composite shell structures, the ply orientation and ply thickness can be optimized. The user interface allows the user to associate material design variables with particular materials and geometric design variables with individual elements or group of elements. The user can select either to minimize the material volume, weight, or cost. The optimization is performed such that the final design satisfied the constraints on the multiple loadcases performed. These loadcases may include linear static analysis and/or eigenvalue analysis. Constraints can be placed on the resultant displacements, strains, stresses, and frequencies. Mentat will display the values of the objective function (volume, weight, or cost) during the iterative processes and indicate which designs are feasible or infeasible.



Sensitivity of Constraint | Sensitivity of Objective Function | Minimization of Objective Function

Figure 3

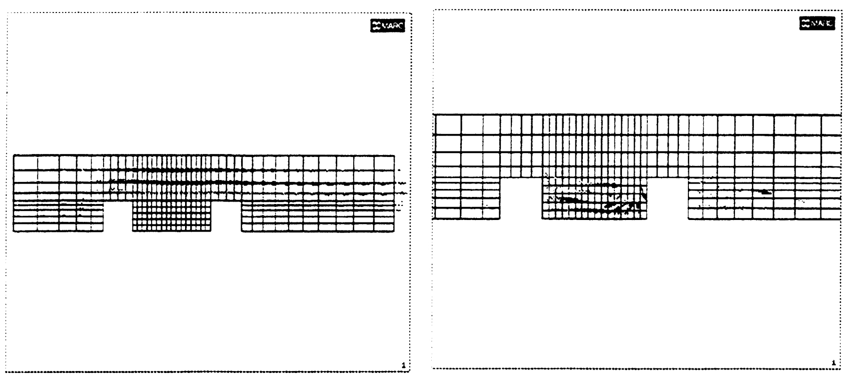
Often engineers are only interested in which design variables have the greatest bearing on the response. In such cases, a design sensitivity analysis can be performed at the current design point. The finite element contributions to the response and the derivative of the response quantities with respect to the design variables are computed.

Fluid Mechanics

The MARC K7.1 release allows the solution of three classes of fluid problems:

1. Isothermal Fluid Behavior
2. Thermally Coupled Fluid
3. Fluid Solid Interaction Analysis

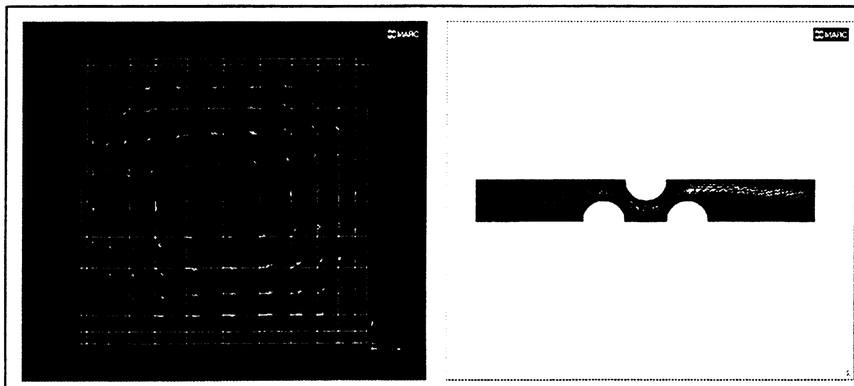
The Navier Stokes equations are used to solve for the velocity in the fluid region. The fluid is treated as either a Newtonian (constant viscosity) or a non-Newtonian; it can be applied to liquids such as air, water, oil, homogeneous slurries, cement, rubber solutions, biological fluids, etc. The computational procedure is restricted to domains where turbulence is not relevant. This is often acceptable for manufacturing simulations, cooling applications, food processing, and biomedical among others.



Flow over Electronic Chips

Close up of Flow showing Flow Reversal

Figure 4

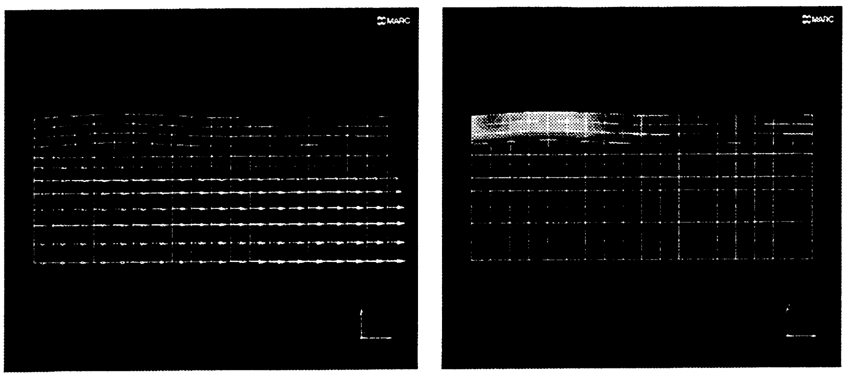


Recirculating Flow in a Cavity

Flow through a Heat Exchanger

Figure 5

In fluid-solid analysis, the solid region can contain any material nonlinearities but is restricted to small geometric motion. MARC is working on enhancing this capability such that arbitrarily large deformation is permitted.



Flow through a Deformable Tube

Stresses in Tube

Figure 6

The user has the choice of using a mixed method approach or a velocity/penalty method approach to satisfy the incompressibility conditions. The pressure stabilizing Petrov-Galerkin (PSPG) method is employed so that lower-order elements can be used, and the velocity and pressure use the same order of interpolation. The streamline upwinding technique (SUPG) is used to insure stability of the numerical procedures.

Element Technology

Two new thin shell elements have been added that have reduced computational costs when compared with element type 75.

Element 138 is a 3-node thin shell element.

Element 139 is a 4-node thin shell element.

Both elements are based upon discrete Kirchhoff theory and can be used for elastic, composite, or inelastic models.

The rebar elements have been reformulated, and rebar elements have been developed for lower-order elements and membrane elements. There is also a new user interface to the rebars, such that the orientation can be given in the input file. This complements the use of user subroutine REBAR. For orginally axisymmetric structures, a simplified input exists, which is of special importance to modelling tires and air springs.

These elements are as follows:

142	8-node axisymmetric rebar with twist
143	4-node plane strain rebar
144	4-node axisymmetric rebar
145	4-node axisymmetric rebar with twist
146	8-node three-dimensional solid rebar
147	4-node membrane rebar
148	8-node membrane rebar

General Technology

The MARC program now uses dynamic memory; hence, the program will allocate more memory as necessary with the intention of keeping as much of the data in-core as possible. If no number is placed on the SIZING parameter, a value based upon the include file (NORMAL) will be used.

The input has been changed such that the user can enter long integers (up to 10 digits) and real numbers with higher precision (up to 20 digits). This allows the solution of larger problems and may yield more accurate results. Mentat 3.1 outputs the MARC input file in the new format if required. Of course, old style inputs are still acceptable.

It is now possible to change some of the control values while the analysis is being performed. This includes the values entered through the CONTROL, POST INCREMENT, and RESTART INCREMENT options. This gives the user the flexibility to monitor the behavior of the computational analysis and tune the parameters as required.

The exit messages have been clarified and exist as a separate file. The user can change the text of the exit messages.

A material data base is available that contains a limited number of steel materials. This includes temperature dependent and rate-dependent properties. It is possible for the user to add his own set of materials to the data base.

A new adaptive load/time stepping procedure is available with this release. This procedure uses multiple criteria to determine if the step size should be reduced in the Newton-Raphson iterative procedure or changed for the next increment of analysis. The criteria may be based upon the change in strain, stress, plastic strain, or displacement among others. It has been shown to be very robust for large strain elasticity and plasticity problems, cracking analysis, or large displacement kinematics problems among others. It is easy to use and robust.

The automatic load stepping procedure, based upon the arc-length methods (AUTO INCREMENT) has been enhanced. In addition to the Chrisfield method, the Rik's method and Rik's-Ramm method can be used to solve buckling and post buckling phenomena.

Calculation of buckling collapse loads can now be performed using the Lanczos method as an alternative to the inverse power sweep method. This is advantageous for large analyses or if many modes are desired.

MARC can read in radiation view factors calculated by Mentat 3.1 and perform a heat transfer analysis including radiation.

A new utility routine is available for the user to obtain values such as stress, strain from the MARC data base within the user's user subroutine. All variables that could be put on the post file are available.

It is now possible to have user-defined vectors placed on the post file and visualized with Mentat 3.1. This is done with user subroutine UPOSTV.

It is possible to create a file with input data that the user desires to be the defaults in his analysis. This data is then no longer required in subsequent analysis.

Mentat 3.1

The Mentat 3.1 release contains major improvements over the previous release. These enhancements substantially improve the meshing capabilities, the graphics performance, and the integration to analysis.

CAD Integration

There is a new IGES reader in Mentat 3.1. This is much more reliable than the previous reader, faster, and requires less memory.

There is a new VDAFS reader in Mentat 3.1 as well.

The DXF reader has been completely rewritten. The new one supports many more DXF entities and has improved reliability.

Mentat 3.1 now uses the ACIS 2.1 geometric engine from Spatial Technology. This version is substantially more reliable and allows the import of SAT files from CAD systems that use this newer release.

There are new tools in Mentat 3.1 that allow the user to clean up the CAD geometry before meshing. These tools can be used to eliminate duplicate points, check for closed trimming curves, close trimming curves, and add trimming curves.

Geometry Clean-up

After a CAD geometry is imported into Mentat 3.1, it is often necessary to clean up the geometry before the automatic meshers are able to mesh it. The reason is that often, when creating a part, the user makes many small design changes that can result in very small geometric entities. Many CAD systems also create small entities, often called slivers, when performing intersections or blends.

These small geometric entities cause substantial problems for the automatic meshers because the meshers try to associate at least one element with each entity. Hence, the user obtains a mesh with far too many elements and a large amount of transitioning. The geometric clean up tools include:

- merging of points
- placing trimming curves on surfaces where they are missing
- removing curves which are not attached to a surface
- remove tiny curves
- replace a series of tiny curves with long curve
- eliminate gaps in curves
- breaking curves – split curves at the endpoint of associated curve to ensure that meshes are compatible across surfaces
- split curve – break a curve at its cusp point to be certain that the meshing is accurate
- check curves/surfaces – checks the topology and supplies the user with summary of geometry found

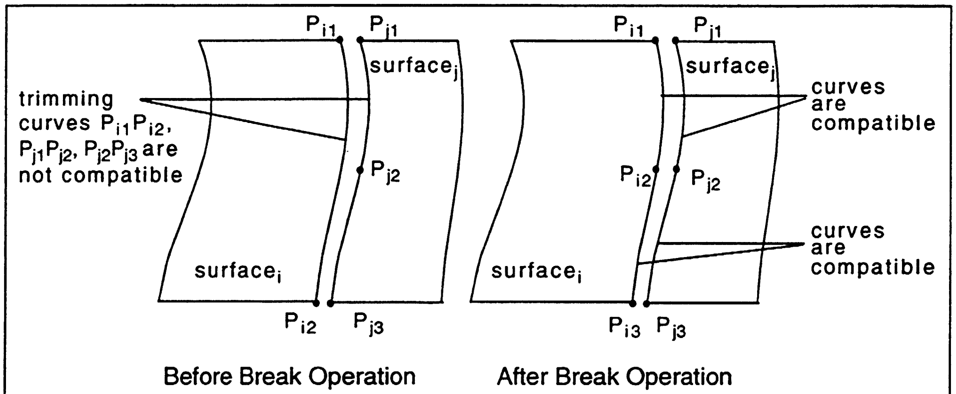
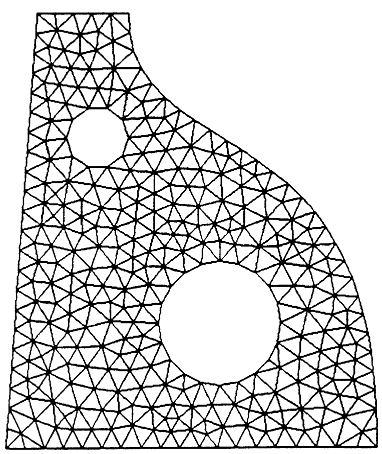


Figure 7 Geometric Clean-Up Operations to Insure Compatable Meshes

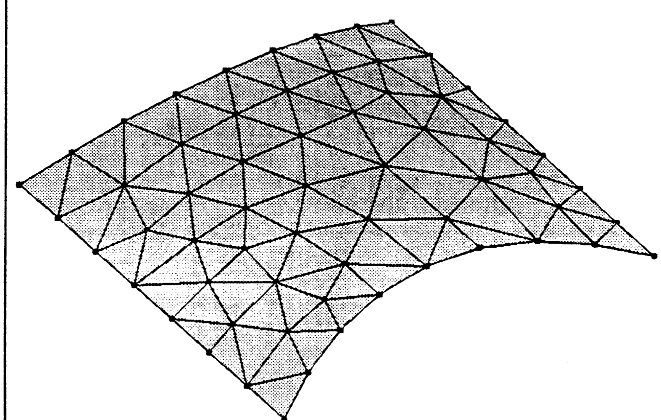
Meshing

A new triangular mesh generation has been developed for planar and surface geometries based upon Delaunay triangularization. This mesher is substantially faster than the previous mesher and generates better meshes. The user can control the number of seed points along the exterior boundaries and along lines (scratches) in the interior. The mesher produces nicely shaped elements with good transitional behavior. For three dimensional shell structures, the mesher can be used to create the model for the new thin shell element type 138.



Delaunay Mesher

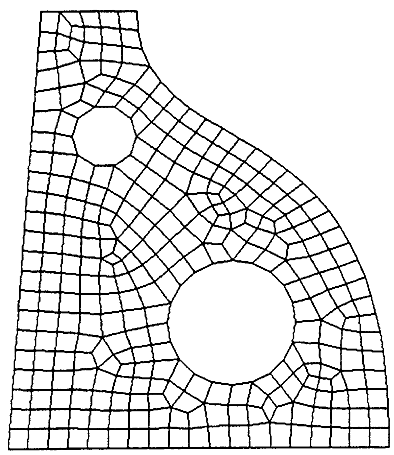
Figure 8



Delaunay Mesher for a Simple Surface

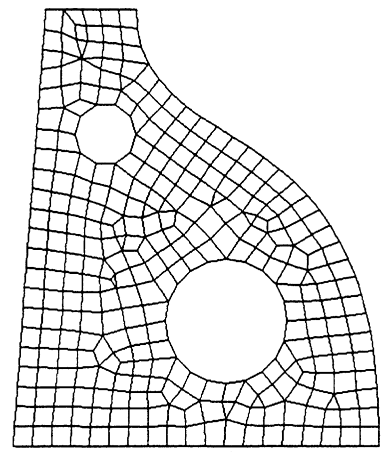
Figure 11

A new quadrilateral mesh generator has been developed for planar and surface geometries based upon the advancing front technique. This mesher can be used to generate all quadrilateral elements, or a mix of quadrilateral and triangular elements if it cannot generate all quadrilateral elements due to topology, or a mix of elements to satisfy constraints on element distortion.



Quadrilateral Mesher

Figure 12



Mixed Quadrilateral - Tetrahedral Mesher

Figure 13

The new mesher generates better quadrilateral elements over the previous mesher, and has better transitioning capabilities.

When using the meshers, it is necessary to place seed points along the boundary of the surfaces. Mentat 3.1 has a new capability where the number of speed points is generated based upon the curvature of the edges. As in the previous versions, the user can control the number of points explicitly or based upon spacing. To generate a completely quadrilateral mesh, it is necessary that the number of points along a closed loop be an even number. This constraint can be automatically imposed.

The expand capability has been enriched so the user can expand a finite element mesh along a curve.

The controls for the grid size, spacing, and orientation have been improved.

Graphic System

The graphic system of Mentat has been completely rewritten for the 3.1 release. This has led to dramatic improvements in the graphical performance. This will allow substantially larger models to be built and displayed in Mentat quickly.

The data base-graphics interaction has been modified so that only the modified data base is redisplayed. Operations such as adding or deleting nodes and adding labels now take a fraction of the time previously required.

The x-y plotting capability used for beam post processing, time history plots, and tables has been substantially improved.

The annotation compatibility now works with x-y plots, time history plots, and table plots.

Arrows are now elegantly drawn.

A bar chart capability has been added for design sensitivity and optimization. This capability can also be used for all x-y plots and time history plots.

The display of curves and surfaces is done to user-defined tolerances. This permits faster display of the geometry with greater accuracy.

Mentat now supports GIF, TIFF, BMP, and JPEG output standards as well as PostScript. The PostScript plotter has been improved to generate figures with greater accuracy.

The menu system has been completely rewritten so that it supports button highlighting and is much faster. It is now easier to customize the menus. The pop-up menus in the static menu (such as UTILS, VIEW, PLOT, FILES and SELECT) can be made to be permanently visible, which reduces mouse clicks.

The graphic window can now be resized at any time during the session.

Mentat 3.1 supports OpenGL on Sun Solaris 2.5 and PC/NT platforms.

Analysis Integration

Mentat 3.1 release supports the new features of the MARC K7.1 release including the new elements, design sensitivity and optimization, fluids, new materials, and REBAR definition. The MARC writer has improved reliability.

The MARC reader has been completely rewritten and now supports more of the MARC capabilities.

The definition of tying, servolink, and spring data is significantly easier.

Mentat 3.1 supports the MARC K7.1 post file format. For contact analyses using analytical rigid bodies, the rigid body is more accurately displayed.

Mentat will calculate radiation viewfactors for planar, axisymmetric, and three-dimensional regions. The cavities can be either open or closed, and symmetry surface capability is available. These geometric viewfactors are then exported to MARC for the radiation analysis.

The rubber and viscoelastic curve fitting program is now available through the Mentat user interface. This capability has been substantially improved both in models supported, accuracy in deriving the material constants, and the user interaction. After the constants are obtained, it is possible to view the original experimental data and the numerically obtained curve fit values. This feature can derive the material constants for Neo-Hookean, Mooney-Rivlin, Signiorini, Second or Third Order Deformation, Yeoh, Ogden, or Foam strain energy functions. Furthermore, the constants required for the Rubber Damage Models and Viscoelastic Prony series can be calculated. The constants obtained for the Odgen and Foam Model are checked to insure stability.

Mentat 3.1 includes a data base of steels and steel alloys. The user can expand this data base to include particular materials required for his own application.

There are now several ways to display the element results calculated at the integration points by MARC. This allows the user to contour the results without extrapolating the results to the nodes or averaging between the element contributions.

The GUI associated with the AUTO MESH and CONTACT options have been changed to support the new capabilities and they are more intuitive.

Binary files are now compatible between machines. Hence, the user can read a binary MARC post file created on UNIX on NT and vice-versa, The user can also exchange binary MUD files between machines unless solid modeling is included.

Operating System Support

The following table indicates which UNIX products are currently supported for the MARC K7.1 and Mentat 3.1 release. Windows/NT products are covered in a separate document. If you have an operating system earlier to those listed here, it is no longer officially supported by MARC Analysis Research Corporation. We suggest that you upgrade your computer system to a current revision of hardware provided operating system.

If you have a newer operating system than what is listed here, please use the operating system that is closest to your installed system. If you have any difficulties, please contact your local MARC support office.

Computer	Chip	OS	C Compiler	Fortran	MARC K7.1	Mentat 3.1	MARC/ Link-S 4.0	MARC/ Link-P 14+
sun	sparc	2.3	3.0	2.01	X	No	No	No
sun	sparc	2.4	4.0	3.0	X	X	X	X
sun	ultra2	2.5	4.0	4.2	X	X	X	X
hp	735	9.05	3.0	9.16	X	X	X	X
hp	735	10.01	3.5	10.00.01	X	X	X	X
hp	735	10.20	A.10.22	B.10.20.01	X	Use 735 10.01	Use 735 10.01	Use 735 10.01
hp-spp2	800	10.01	11.0.1	Exemplar 1.2.1	X	Use 735 10.01	Use 735 10.01	Use 735 10.01
ibm	6000	4.1.5	2.1.4	3.02.02	X	X	X	X
ibm	6000	3.2.5	3.1.4	3.2.5	X	X	No	No
sgi	4000	5.3	4.0	4.02	X	X	X	X
sgi	8000	IRIX64 6.2	6.2	6.2	X	Use r4000 5.3	Use r4000 5.3	Use r4000 5.3
sgi	5000	6.3	7.1	7.1	X	X	Use r4000 5.3	Use r4000 5.3
dec	alpha	4.0	5.6	4.1	X	X	No	X
dec	alpha	3.2	5.3	4.0	X	X	No	X

MARC K7.1 Release Notes

The MARC K7.1 release includes significant new features and enhancements to existing features. Moreover, nearly all reported defects in MARC K6.2 have been corrected. The table below briefly highlights some of the key enhancements.

1. Dynamic Memory Allocation:

   During the analysis, you may choose to have MARC automatically allocate the memory. The sizing parameter card is no longer necessary in the input data. In this case or when the sizing card is included but 0 words are specified, the program allocates the default size specified by keyword NORMAL in the include file in the tools subdirectory. If the size is greater than the one specified by MAXSIZE, then MAXSIZE words are allocated if permitted by the machine.

   In case the program fails to get the required memory at once, it will try to get the memory in chunks of about 1M words. If the job requires larger memory than allowed by the machine, the program will use an out-of-core equation solution or out-of-core element storage.

   In certain cases, e.g., bandwidth optimization, CONRAD GAP option and restart where the program cannot fetch the required words (same as the previous analysis) due to multi-tasking, the program will stop with an exit 4031.

   Finally, a word of caution: In going from xM words to yM words, the program allocates yM additional words and copies the old xM words to the beginning of the extra allocated yM words and then frees the original xM words. Thus at some moment, it requires (x+y) M words. Hence, for some machines or jobs it would be advisable for you to do the allocation yourself.

2. Post File Changes:

   If K6 style post file is used, then there are no changes except:

   (i) Frequency for harmonics and eigenmodes is in cycles/time
   (ii) New analysis types can be included
   (iii) Flexible contact bodies are written as sets

   For K7 style post file, one obtains the following additional information:

   (i) User defined nodal post codes via subroutine upostv
   (ii) Loadcase title can be included
   (iii)The rigid bodies in contact described by analytical surfaces get a true NURBSD representation
   (iv) Frequency for harmonics and eigenmodes is in cycles/time
   (v) New analysis types are included
   (vi) Flexible contact bodies are written as sets
   (vii)Last line has ++++ to notify end of the analysis

3. Unsupported Features:

   The following programs are no longer supported as their features are now included in Mentat 3.1.

   (i) Mesh3D

   (ii) Pipe

   (iii) Plot and

   The programs will be supplied on the magnetic media but there will be no associated documentation for them. The Laypack solver on Sun computers is no long supported as it used an excessive amount of memory.

4. Default Changes:

   The following defaults have been changed which may produce different results and/or behavior of the program:

   (i) Factor associated with the drilling degree-of-freedom for the shell elements 75 and 22

   (ii) Default bulk modulus for Ogden model for elastomers is now smaller (consistent with its corresponding 2 term Mooney-Rivlin representation)

   (iii) Default contact zone tolerance for shell and membrane elements is 0.25 times the thickness compared to previous value of 0.5 times the thickness.

5. Contact Changes:

   In general, the contact algorithms have gone through a significant modification and extension. In running the old input files, you may need to change some tolerances to achieve the earlier behavior of the program.

6. For analysis with very large models, 10 digit integers and 20 digit reals can be used with the use of the EXTENDED parameter card.

7. The RADIATION parameter card required you to enter the Stefan-Boltzmann constant in correct units as opposed to the scale factor for the units conversions earlier. Furthermore, this option needs to be now used in conjunction with the RADIATION CAVITY or VIEW FACTOR model definition option.

8. Nodes with point loads need to be defined under the POINT LOAD model definition option. As an alternative, the DIST LOADS parameter card can also be used to specify the maximum number of nodes with the point loads.

9. Material Data Fitting:

   (1) The feature of fitting and viewing of multiple data sets is available in the experimental data fitting menu (in Mentat) in a form which is different from previous fitting program, Curve. Only the data needed in fitting should be loaded, i.e. fitting only a subset of the loaded files is not an option.

   (2) Although the functionality is included in Mentat, the 'original' Curve program is also being supplied on the magnetic media.

10. Analysis with the new elasticity and plasticity material models using the mixed method will show a singularity ratio which is inversely proportional to the bulk modulus of the material. This is result of the condensation procedure used to make the nearly compressible analysis possible with standard displacement based elements.

11. The rebar element data can be given through the model definition card besides the user subroutine. Also, a simplified input option exists for originally axisymmetric geometries.

12. Do not use the Plasticity parameter card when modeling Powder or Soil materials since these materials have pressure dependent yield criterion and hence do not need the constant dilatation approach (which is used automatically with the Plasticity parameter card).

Mentat 3.1 Release Notes

Starting Mentat 3.1

A single startup script is now used to start all versions of Mentat. The OpenGL version of Mentat is run by including the "-ogl" command line option. For a complete description of all command line options see the Mentat 3.1-MARC K7.1:New Features user guide.

New Security System

MARC K7.1 and Mentat 3.1 use an entirely new security system, based upon the FlexLM license manager. This will allow floating network licences in addition to the CPU locked licensed available in previous versions. Please see the "Mentat Installation Manual" for complete instruction on using the new system.

Undocumented Features

The following features are late additions to the product which do not have any associated documentation.

Automatic Hexahedral Mesh Generator (HexMesh)

Automatically generates either a pure hexahedral (brick) or mixed hexahedral and pentahedral (wedge) finite element mesh. The input to the mesher must be an enclosure consisting of triangular and/or quadrilateral elements.

The feature is activated by issuing the command "*hexmesh" which prompts the user for the following information:

- *element size* - The desired size of the elements.
- *edge sensitivity* - Used in the automatic detection of model edges, which is done by measuring the angle between two elements in the enclosure. A value of zero means no edges will be detected and a value of one means all element edges will be considered model edges. A value of.5 usually produces adequate results.
- *level of coarsening* - The mesh generator will make large elements in the interior if this is set to a non-zero value. Tie will be automatically generated to allow this.
- *allow wedges* - Wedges may be optionally generated, which allows better hexahedral elements to be generated with a better representation of the edges.
- *element face list* - The list of element faces forming the enclosure. This list may not have any free edges (see the SWEEP feature) and should all be aligned such that the tops of the elements are facing the outside of the model (see the ALIGN SHELLS feature).

At this point messages will be printed out in the start-up window. After the meshing is completed elements used in the definition of the model will be deleted and the new mesh will be merged into the model. Undo may be used to revert to the input mesh.

The feature is highly experimental, and does not work on all models. In particular, models with narrowly spaced features may not successfully mesh. In addition, a large number of elements may be produced.

Fast Dynamic Drawing

This feature is designed to improve the response of the dynamic viewing option in Mentat. It does this by interrupting a draw of the model whenever the mouse is moved so that the new position of the model may be shown. In this mode it is possible that only part of the model may be drawn during dynamic viewing. The full model is always drawn after mouse motion ceases.

The feature may be turned on or off in the VIEW SETTINGS menu. In addition, the interrupt interval (default.1), expressed in seconds, may be set. Lower interrupt intervals have the potential of making dynamic viewing more responsive, but may cause less of the model to be visible.

The command line option "-qd on/off" is available to set this feature on start-up.

Using PseudoColor8 Visuals in X Windows

Mentat 3.1 uses a 24 bit color model. The advantage of this is that a large number of colors can be represented on the screen. In addition, it is possible to share colors with other applications, avoiding the colormap swapping seen in previous versions.

When an 8 bit or 16 bit visual is encountered, the situation is handled by allocating a static colormap immediately upon start-up such that a reasonable approximation to the desired color is available. The larger the colormap, the more accurately the colors will appear on the screen.

When there are 16 bitplanes available this colormap is quite large, and there will usually be a color very close to the requested color. However, when there are only 8 bitplanes available, the results may not be very good.

This is because with only 8 planes there will be at best 256 colors available, and in many circumstances (for example if other applications have allocated colors) there may be many fewer colors available. This causes Mentat to use a poor approximation to the desired colors.

Before starting Mentat, close other applications which may have grabbed too many colors. This will give Mentat access to as many colors as possible.

Alternatively use the "-ic nplane" command line option, which directs Mentat to install its own private colormap when there are fewer than "nplane" (default 6) bitplanes available. Changing the value to 7 or 8 instructs Mentat to install a new 256 color colormap for its own use, when there are fewer than 128 or 256 colors available.

This will provide Mentat with more colors at the cost of having colormap flashing as the mouse is moved in and out of the Mentat window. This is a consequence of Mentat having a different colormap than the window manager or other applications.

New Image Postscript Interface

Due to extremely poor performance, the previous implementation of the Postscript interface was replaced in favor of one based on arbitrary resolution z-buffering. This approach avoids the expensive graphical primitive sorting required to determine the proper draw order.

The great advantages of the new approach are its great speed, (in most cases less than 30 seconds for very large models), and consistent quality. In addition, for large models the size of the Postscript file produced is much smaller.

Several important drawbacks of the new approach which still are being worked on are:
As the resolution of the image is increased the size of the fonts decreases accordingly. This can be a problem at resolutions of 300 dots-per-inch or greater. One easy work around for this is to temporarily increase the size of the current text font while producing images.

For simple models the Postscript file produced is several times larger than would have been produced in the previous approach. This is despite the use of compression techniques and is an unavoidable consequence of the approach.

At low resolutions lines and fonts will appear jagged. Usually, for quality images it is best to avoid the 75 dots-per-inch setting.

Changes in Element Contouring

For the sake of optimization of the drawing speed, quadrilateral elements are now drawn with fewer triangles. The new version uses only one half the number of polygons as previous versions to represent the same mesh. This will cause the contours to appear different than in previous versions.

To improve the accuracy, increase the number of divisions using the Plot>Divisions>Elements command.

Experimental Material Data Fitting

The "X-Curve" program is now available directly in Mentat. Experimental data is input through tables, either manually or through data files. The format for the input data files needed for experimental data fitting needs the following format:

1st Column	2nd Column	3rd Column
eng. strain	eng. stress	volumetric information

The volumetric information is used for the foam model and is supplied as area ratios (for uniaxial test data) and thickness ratios (for biaxial test data). For other modes the third column defaults to one.

Please note that this format is different from the previous release of the curve program in that the 2nd and the 3rd column are switched. For an easy conversion one can use the unix utility awk to convert the old files to the new ones as:

(1) for data separated by spaces -

```
awk -F" " '{print $1 " " $3 " " $2}' old_file > new_file
```

(2) for data separated by commas -

```
awk -F, '{print $1 $3 $2}' old_file > new_file
```

MESA (OpenGL on X)

On some machines a special version of Mentat is provided which utilizes the MESA OpenGL library. This allows light source shading effects to be available on X-Windows displays without the need of specialized 3D graphics hardware.

The Mesa Graphics Library is a freely available cross-platform software implementation of OpenGL. This library has been included with Mentat in a pre-compile form.

To use the OpenGL (Mesa) version of Mentat, use the "-mgl" flag on the Mentat command line. Be forewarned, however, that the graphics may be significantly slower while using the Mesa Graphics Library, particularly when displaying across a network.

Information about the Mesa 3-D Graphics Library, including source code, patches, and version information, may be found at http://www.ssec.wisc.edu/~brianp/Mesa.html. The version included with Mentat 3.0 is 2.4, and is located in the bin/Mesa directory. Do not attempt to use any version prior to 2.4.

Dropped Features

Translucency

Translucency of elements and surfaces will be implemented for OpenGL for a future release.

Continuous Contours

A reasonable way to get the continuous contour effect is to increase the number of contour levels to 30.

Color Bar Sliders

This will be implemented in a future release. For now it is necessary to type in the red, green, and blue components of the desired color.

Opening/Closing Post Files

Before opening a post file, a copy of all model data is saved internally. In addition, the model name is changed to be that of the post file being opened. This has been done to prevent the overwriting of model files when using the save command.

Closing a post file now automatically restores the model existing prior to opening the post file, restores the model name, and turns off all post plotting. This allows quick edits of models after examination of the results.

To edit a model from a post file it is necessary to save the model before closing the post file. Then simply open the saved model.

As before, if memory usage is a concern it is a good idea to save and clear a model before opening a post file.

IGES Translation

Import

When importing an IGES file, sometimes it might be useful to turn on the "validate" option, especially if something appears to be wrong in model. The down side of this is that the translation might fail.

Export

Occasionally an exported model may have ambiguous or bad entities. If that happens, the IGES validation will fail and nothing will be exported. When this occurs, it might be helpful to turn the validation off using "-voff" in the stand alone mode to get as much information out of Mentat as possible.

Overlay Planes for OpenGL

The OpenGL version of Mentat 3.1 uses overlay planes to draw the picking box, cross hairs, and lasso. Some machines may not have overlay planes, in which case blending is used. However, some machines may need to have their overlay planes enabled. Specifically:

SUN

On the Sun, enable the overlay visuals:
1. su root
2. /usr/sbin/ffbconfig -sov enable
3. reboot

IBM

1. su root
2. vi /usr/lpp/X11/defaults/xserverrc (or use a preferred editor)

Change the OpenGL extensions line to read:

EXTENSIONS="-x mbx -x abx -x GLX -layer 0" (for older systems)
or
EXTENSIONS="-x dbe -x abx -x GLX -layer 0" (for newer systems)
3. reboot

DEC

On the DEC, make sure you have the latest open3D440 release installed.

External Analysis Interfaces

NASTRAN Bulk Data Writer

A new external program has been added for exporting the FE model to NASTRAN Bulk Data format. The details of entities supported are included in the Mentat 3.1-MARC K7.1:New Features user guide.

I-DEAS Universal File Reader

- The new coordinate frame (packet type 2420) is now supported. Also transforms are created in the Mentat database and added to appropriate nodes.
- Reading of beam sections (packet type 776) is enabled.
- Node and Element groups (packet type 2429) are now read as sets.

- A bug in reading fixed temperature information is now fixed.
- Displacement coordinate ID's are added to nodes.
- The new load case (packet type 2428) format is now supported.
- New logic has been added for axisymmetric analysis.
- Support for spring elements is added.
- The loops for adding of geometric and material information to elements are now optimized for a ten fold performance gain.

PATRAN Neutral file Reader

- Geometric property Shell thickness is now read.
- Logic has been added to combine loads and boundary conditions of the same value into one Mentat apply.
- Some bugs in reading pressures applied to edges are now fixed.
- Material plasticity flag does not default to on.
- Element ID's are assigned to elements as read from the Neutral file.
- The program no longer core dumps on DEC UNIX.

NASTRAN Bulk Data Reader

- New logic has been added to dynamically allocate memory overcoming an artificial limit on the file size that could be read.
- Support for large format has been added. As a result, Mentat can read the files that it writes.
- Element types are more closely matched.

List of Obsolete Commands

The following commands are no longer supported:

- all_iges_import_layers
- animation_formatted
- button_save
- calcomp_color
- check_curve_loops
- check_surface_loops
- clear_all_iges_import_colors
- clear_composite_material
- color_mode
- composite_material
- convert_solid_faces_surfaces_discrete
- copy_composite_materials
- curve_bias
- curve_divisions
- duplicate_comp_material
- flyby_curves
- gray
- gray_mode
- hpgl2_color
- hpgl_color
- iges_import_layers

- mono_mode
- origin_xy
- pick_list_next
- pick_list_prev
- pick_list_top
- PostScript_center
- PostScript_height
- PostScript_trans_x
- PostScript_trans_y
- PostScript_width
- set_all_iges_import_colors
- set_curve_divisions
- set_iges_import_color
- set_surface_divisions
- solid_elements
- solid_solids
- solid_surfaces
- translucent_elements
- translucent_solids
- translucent_surfaces
- write_calcomp_color
- write_hpgl2_color
- write_hpgl_color
- write_postscript_color_all
- write_postscript_gray_all

New MARC Reader

The new MARC reader, based upon the MARC K71 source code, replaces the MARC reader in Mentat 2.3.1, which is no longer supported. However, it is still available as an unsupported command, "*read_marc <filename>".

ACIS Support

Mentat 3.1 uses version ACIS 2.1R7 or 2.1R8 from Spatial Technology. This is significantly improved over the previous releases. The DEC UNIX 3.2 Version does not include solid modeling support.

Executing Auxiliary Standalone Programs for DXF, IGES, HEXMESH, IDEAS, IGESOUT, NASTRANOUT, PATRAN, VDA

To execute the auxiliary standalone programs, use the "mrun" shell script in the mentat 310/bin directory in the following syntax:

```
mrun program options
```

Automatic Generation of Hexahedral Meshes

The automatic generation of fully hexahedral meshes has been the goal of many researchers and developers in the computer-aided engineering industry. Incorporated into the Mentat 3.1 release is an optional first version of such a tool. This mesher will create a finite element mesh consisting of hexahedrons and optionally wedges (collapsed hexahedrons). The creation of a hexahedral mesh is extremely challenging, given the requirements of:

1. creating minimally distorted elements,
2. creating as few elements as possible to minimize the analysis computational costs
3. having the mesh generator execute in reasonable computational time and within a minimal amount of memory.

Distorted elements are usually created either because of surface geometries (high curvature), or because of the requirements of creating a transition mesh. Transition regions occur either because of the need to go from fine regions to coarse regions, or as a ramification of the mesher working from multiple surfaces and meeting somewhere in the middle.

This new mesh generation technology attempts to circumvent these challenges by reducing the meshing to a few simple concepts.

The user needs to provide a volume that is fully enclosed by piecewise linear patches (either triangular or quadrilaterals elements). The volume may be multiply connected. In general, this is performed by undertaking the following steps.

The first step is to create a geometry in Mentat using either the solid modeling capability of Mentat or simply as a collection of Mentat surfaces that bound a solid region. For an externally created model, read in a geometry using the file import command within Mentat. This may be a geometry created in I-DEAS®, Pro/Engineering®, AutoCAD®, or other ACIS® based geometric system. It is also possible to bring in a geometry using IGES files or VDAFS. As only a representation of the surface will be used, it is even possible to begin with an already created tetrahedral finite element mesh or surface mesh.

If the volume is an ACIS representation, it is necessary to convert the faces into surfaces using the CONVERT command under the MESH GENERATION menu.

If the volume is represented by surfaces, it is necessary to discretize these surfaces into triangles or quadrilaterals. The AUTO MESH capabilities should be used here. Note that the geometry clean-up tools are generally required to insure that there are no cracks between surfaces. The finite element mesh on these surfaces is only used to represent the body, the resultant hexahedral finite element mesh is independent of the size of the surface elements. The discretization of the surface does influence the location of edges in the final mesh. The SWEEP command should be used to insure that there are no duplicate nodes in the model, and that the volume is fully enclosed. By using the OUTLINE EDGES mode in the PLOT menu, the user can determine if cracks exist in the mesh. Lines should appear only where true edges exist. The elements on the exterior surface should be aligned so they all have the same outward pointing normal. The ALIGN SHELL command under the MESH GENERATION>CHECK menu can be used to perform this task. If an existing tetrahedral mesh is being used, then this is not necessary, as the normals are naturally aligned (assuming a valid tetrahedral mesh).

The user then types *hexmesh and is prompted for a number of parameters.

The most important is the desired element size of a typical element. The mesher will proceed to make a grid of this size, aligned with the global x, y, and z axes. The cubes associated with this grid are called atoms. All atoms outside the volume are deleted, all atoms within the volume are saved. Those atoms near the surface are manipulated so that the mesh captures the surface and the edges of the body.

Note that in this version, the mesh generator will create a different mesh depending on how the body is oriented with respect to the global x, y, z axes. Rotating the geometry with respect to the Cartesian frame can result in better meshes. The atom size is reasonably critical, as too large of a size will result in an inability to capture small details, while too small of a size will result in an excessive number of elements.

When creating the hexahedral finite element mesh, the system attempts to place element nodes at real vertices of the body, and element edges aligned with the edges of the body. The edges of the body are determined by evaluating the angle between adjacent elements in the original surface mesh. The user can control the sensitivity of detecting edges with a parameter. A value of zero means no edges will be detected and a value of one means any edge will be considered model edge. A value of 0.5 usually produces good results.

It is very difficult to achieve the condition that a hexahedral element is aligned with both the surfaces and edges of a body. To improve the representation of the solid, the user should usually permit wedges in the mesh. These wedges (collapsed bricks) will only appear on the outside surfaces. These wedges will transition into one brick element. Note that no new nodes are introduced, so that displacement continuity is preserved.

Because of the large number of elements that are generated in the interior, an automatic element transition procedure has been included. Up to two levels of element coalescing is possible. In the limit, this could reduce the number of elements in the mesh by afactor of 64. Rather than making transition elements, which will have a high degree of distortion, constraint equations are formed between the elements. When MARC obtains the mesh data from Mentat, it therefore includes CONNECTIVITY, COORDINATES, and TYING data.

The list of faces that enclose the model needs to be identified. This is done by using the all SURFACE command.

This version is limited to creating 99,999 elements or nodes.

This is the first version of this mesh generation technology. MARC felt that is was important to make it available as quickly as possible to the engineering community.

It is our plan to enhance it so among other things, it directly uses the surface data within Mentat and has additional capability to model small details, without generating an excessive number of elements. We are looking forward to receiving your feedback.

Online Documentation

Online Documentation

Contents of Online Documentation

The following documents are now available online:

- Vol. A: Theory and User Information
- Vol. B: Element Library
- Vol. C: Program Input
- Vol. D: User Subroutines and Special Routines
- Mentat 3.1-MARC K7.1:New Features

Formats for Online Documentation on the CD-ROM

The formats for online documentation on the CD-ROM are:

- FrameMaker 5.0.1 (browse with FrameViewer)
- Acrobat 3.0 (browse with Acrobat Reader)

To view the online documentation, you must have FrameViewer or Acrobat Reader installed in your machine. You can install Acrobat Reader using this Online Documentation CD-ROM or directly from the Adobe Systems, Inc. website (http://www.adobe.com/acrobat).

Browsing the Online Documents using FrameViewer

To browse the online documents using FrameViewer:

1. Launch FrameViewer.
2. Choose File>Open.
3. Choose the file, **contents.fm**, in the **frameview** folder.
4. Use the links in the contents page to browse a specific document.

Browsing the Online Documents using Acrobat Reader

To browse the online documents using Acrobat Reader:

1. From the static menu area in Mentat, choose Utilities>More>System Command.
2. Type **acroread**.
3. In Acrobat Reader, choose File>Open.
4. Enter the pathname to the file, **contents.pdf** in the **acroview** folder,.
5. Use the links in the contents page to browse a specific document.

Known Problems in K7.1 Release

Beam element type 31 does not support thermal strains.

Potential core dump if selecting post codes that are not appropriate for analysis type. For example asking for creep strains, when creep is not included in the model.

In a coupled analysis, when using tying, regardless of the tie type, except (31 ,32, 33, 34) during the heat transfer portion, the temperatures are tied using type 100.

Thermally driven CONTACT problems will converge slowly. It is best to set the 7th field of the 2nd data block of the CONTACT option to 3. - Mentat Increment Splitting - Iterative on the JOBS>MECHANICAL>CONTACT CONTROL menu.

Do not define a setname as a numeric value in Mentat. This will result in an input error in MARC.

Reduced integration hourglass elements give poor results for plasticity analysis.

Known Problems in Mentat 3.1

The following is a list of known problems in the 3.1.0 version:

Sometimes it is necessary to issue the "*regenerate" command to update the graphics area. Normally either the graphics area is updated automatically, or a "*draw" command is sufficient.

Translation of solid faces sometimes gives erroneous results, especially for spherical parts.

At times the plotting of trimmed surfaces produces lines or polygons which extend beyond the trimming curves.

Surface/surface intersection may take a long time and give poor results, especially for high order surfaces, or surfaces with singular points. It is useful to reduce the order of surfaces by converting them to interpolated type.

Reading MARC input files with nonconsecutive node ids will produce extra nodes.

Changing directories after opening a model file with a relative path name will cause subsequent saves to write to a different file. Therefore, it is best to either start Mentat in the desired directory, or to change directories before opening a new model.

For post files which have different post data on different increments the currently plotted scalar, vector, and tensor, may change as these increments are viewed.

Post files with rezoning and rigid dies composed of curves or surfaces may not be able to display properly the undeformed positions of those dies.

Binary model files (.mud) having solids are not portable to other hardware platforms. Especially between DEC and PC hardware to the rest of the Unix world.

Display of rgb files on 16 plane display devices may produce scrambled colors.

Post files having elements with non-ascending ids will display wrong results.

Contour or numeric plots of principal values in combination with isolate will give erroneous results.

Running the OpenGL version of Mentat across heterogeneous machines may not work correctly. For example, running Mentat on a DEC workstation while viewing on an SGI display may not work correctly.

Problems Detected in MARC K6.2 - Resolved in K7.1

Performing a BUCKLE analysis with CONTACT. Only the first collapse mode was calculated. Result could have been Nan's for other modes.

Potential floating point error if rotational velocities were prescribed to a rigid body.

The RELEASE NODE option did not work as intended. The reaction forces were not removed correctly.

The RELEASE, 1, (gradual release) did not work as intended. The reaction forces were not removed correctly.

Program gave EXIT number 40 "the normal direction of tied node xxx has the same direction as B.C." when it was not necessarily true.

Conversion of NURB surfaces to patches did not always accurately represent the surface.

CONTACT using NURB surfaces had problems when the node was in contact with the edge of the NURB surface.

When shells came into CONTACT, contact would often be detected with the wrong shell surface.

The normal to a contact patch was not always accurate, if the patch was highly distorted. A flat rectangular patch would produce the least errors.

Exit 40 "check your die speed" would occur when approaching the deformable body in increment zero, if the rigid bodies had a very small velocity.

The Contact Force reported in the output does not include the friction force. The output has been changed in MARC K7.

Contact analysis had potential problems if nonconsecutive node numbers were used.

Bodies do not release properly if chatter suppression option 2 is used in CONTACT.

CONTACT analysis with AUTO INCREMENT was incorrect for either deformable to deformable or deformable to rigid contact.

Poor behavior occurred if FOLLOW FORCE stiffness used in a CONTACT analysis.

TEMPERATURE CHANGE in a COUPLED CONTACT analysis is incorrect if the number of nodes is less than was initially prescribed.

Debug print messages are going to the console in CONTACT analyses, with analytical rigid surfaces, on SUN, DEC, IBM machines.

CONTACT occasionally goes wrong if a node is in contact with two rigid surfaces and one deformable body.

CONTACT occasionally goes wrong if a node is in contact with a rigid surface and then contacts a deformable body.

CONTACT analysis may core dump if element types 12, 97, 31, or 68 are present in the analysis.

CONTACT analysis may core dump if the SOIL model is included.

Potential floating invalid messages on DEC -Alpha in CONTACT analysis.

Interference fit problems worked poorly in three dimensions.

In CONTACT Analysis, if very large motions are taken, with analytical rigid surfaces it is possible that contact is not detected.

Node is not placed exactly on a rigid surface in CONTACT if the APPBC option is used or the sparse iterative solver, solver 1.

In CONTACT Analysis, occasionally during increment splitting the time step is reduced to 0.0. The program than gives an EXIT 38, because no time step is defined.

Potential penetration when two nodes of two bodies should come into contact, but are virtually coincident.

Poor convergence occurred with Herrmann elements in CONTACT analysis when increment splitting occurred.

Poor convergence occurred in R-P FLOW analysis using CONTACT when increment splitting occurred.

Possible error in CONTACT detection after REZONING.

The SORT NODE option was very slow in a CONTACT analysis.

CONTACT would indicate that a edge had a zero length edge, when in fact it was merely very small.

Extrapolation of element values to nodal values for distributed friction calculation was not correct for all elements.

Error when using POWDER materials in a CONTACT analysis. Possible negative square root in transient DYNAMICS with two-dimensional CONTACT.

Problems with CONTACT when using higher order collapsed elements.

Combination of user subroutine UTRANS in a CONTACT analysis caused problems.

Friction behavior in increment after rezoning is very poor which may lead to slow convergence or a bad solution for this increment.

There was a limit to the number of data points for Xcurve data fitting program.

Xcurve program had problem fitting Ogden data if four or more terms were requested.

Reading in data for viscoelastic SHIFT FUNCTION goes wrong if using the power law procedure, and the number of terms is greater than one.

User subroutine UPOWDR was only called if Young's modulus was zero. It was not called during heat transfer portion of coupled analysis.

OGDEN and FOAM models could not be input for COUPLED analysis.

Thermal strain calculation was not correct for MOONEY model or FOAM model or viscoelastic Mooney model.

Cracking analysis may give incorrect analysis if principal values of the stress are the same.

Viscoelastic orthotropic behavior is not correct. The program does not pick up the values of E11,E22,E33, p12,p23,p31 correctly.

Viscoelastic behavior is not correct for composite elements.

TSHEAR option in combination with ORTHOTROPIC does not give the correct results.

Potential problems if multiple COMPOSITE materials in the model and they have different number of layers.

Progressive failure via FAIL DATA was not correct if the failure was based upon shear stress or shear strain.

CRACKING in conjunction with COMPOSITES may lead to the wrong crack information passed to user subroutine ELEVAR. The results are correct.

The VISCO PLAS material type under ISOTROPIC results in incorrect memory allocation.

Error in ORTHOTROPIC material in combination with ORIENTATION and UPDATE.

Error if the SHELL SECT option and COMPOSITE option is used in analysis, if the number of layers in COMPOSITE option is an even number and larger than value specified on SHELL SECT option.

MARC would fail if both FOAM or OGDEN model, and TEMPERATURE EFFECTS (for other material types), and the FOAM or OGDEN material data was before the TEMPERATURE EFFECTS data.

Unable to reference second, third etc. state variables in user routine HOOKLW.

Temperature increment is always zero in user subroutine HOOKLW if used with the ORTHOTROPIC option.

Thermal loads or creep with element 4,8, or 24 are incorrect.

Use of ORIENTATION with element 4,8, or 24 is incorrect.

When inputting "2" on 2nd field of 3rd card of ANISOTROPIC option in HEAT transfer, overwriting of standard input occurs.

Explicit dynamics, DYNAMIC,5 had an addressing error that may have resulted in a core dump.

A MODAL SHAPE analysis immediately after a REZONING step is incorrect.

A MODAL SHAPE analysis immediately after restart, if the RESTART LAST option is used is incorrect. If the RESTART option was used, there is no problem.

Combination of COUPLE and DYNAMIC causes job to fail.

EBE solver, type 1 gives incorrect results for transient DYNAMICS (Newmark or Houbolt), in increment zero, hence incorrect initial accelerations.

EBE solver, type 1 potentially aborts for transient DYNMAICS (Newmark or Houbolt).

If a HARMONIC analysis is requested, with the sparse solver, SOLVER type 4, and the optimizer was set to type 10, a core dump would occur.

Problem if a HARMONIC analysis is done directly after increment zero in an analysis which includes CONTACT.

Applied imaginary loads in a HARMONIC analysis with CONTACT were not applied correctly if they were applied on a node that came into contact with a rigid body.

Application of imaginary distributed loads was incorrect.

Tying in a complex HARMONIC analysis was not correct.

Element type 80 is incorrect with HARMONICS.

Post file is incorrect for HARMONICS in CONTACT analysis.

Prescribed displacements not applied correctly in HARMONIC analysis with OPTIMIZE.

HARMONIC analysis using PHI FUNCTIONS and axisymmetric elements may result in divide by zero.

Conversion of user input of cycles/time to radians/time was not exact when inputting range of frequencies for Harmonics.

User routine USPCHT was not being called for heat transfer shell elements.

MARC pre-reader did not read ORTHOTROPIC data in HEAT transfer correctly or RADIATION correctly, the result could have been that data errors were flagged, when they did not exist.

If INITIAL TEMPERATURES and INITIAL PLASTIC STRAINS are both used in a job to read values from the post file, the wrong data is read in.

Third component of heat flux for a two dimensional problem was not zero on the post file.

Error in applying fixed temperatures in a JOULE heating analysis. Temperatures were overwritten.

User subroutine IMPD was not called for HEAT transfer analysis.

Convergence testing is not complete for increment 1, cycle 0 in HEAT transfer analysis.

In a HEAT transfer analysis with shell elements, if POST is included, but no element post codes are entered, the result is EXIT 13.

Third component of heat flux for a two dimensional problem was not zero on the post file.

Integration point coordinates were not correctly passed into user subroutine FILM for element types 124-135.

COUPLED analysis with SPRINGS gives incorrect results.

In a COUPLE R-P FLOW problem, temperatures may increase in the first heat transfer step due to artificial plasticity.

Error with RESTART if non consecutive element numbers were Friction heating does not work for nodal based friction models.

Scaling of mechanical loads was not correct in COUPLED analysis using TRANSIENT option.

Scaling of mechanical loads was not correct when used in conjunction with AUTO THERM and CHANGE STATE.

Nodal FLUXES in the output file and post file are incorrect for COUPLED deformable-deformable CONTACT analysis. Results are correct.

COUPLED CONTACT analysis was not possible with beam element type 5.

Flux and gradient output was not correct in a COUPLED analysis. Temperatures were correct.

Possible error if the TYING model definition block was used more than once.

TYING goes wrong with sparse solver type 4 if out-of-core. This is also a problem for deformable to deformable contact.

FOLLOW FORCE and AUTO INCREMENT with gravity load type 102 gave incorrect result.

Centrifugal or Coriolis with FOLLOW FORCE stiffness and PROCESSOR option gave incorrect result.

Using Assumed Strain formulation with Herrmann element type 84 gave poor results.

Centrifugal or Coriolis loads with element type 117 give incorrect results.

When nonuniform distributed loads are used with element type 7, where the direction must be specified, if the direction is not specified, no warning is given. Results are wrong.

Use of element 20 and 83 at the same node gave messages *** warning - node number xx shared by elements with different nodal degrees of freedom. This is not a problem for these two element types and message should not have appeared.

Control over the contribution of the deviatoric stress to the initial stress stiffness matrix does not work.

Membrane elements type 18 gives slightly incorrect results if degenerated and PROCESSOR option is used.

Centrifugal loads go wrong with element type 51.

Lumped mass matrix goes wrong for element type 49.

Improve element type 20 for use with LARGE DISP.

Incorrect results with generalized plane strain elements when LARGE DISP is included in the model.

Fix up pipe bending, element type 31 element stiffness and mass calculations.

Adaptive meshing with element type 75 and more than one level automatically subdivided all elements.

ADAPTIVE meshing with tetrahedral elements type 134 or 135 give MARC EXIT 5003 or 5007 or 5010.

Possible problems with ADAPTIVE and collapsed brick elements.

Sparse solver type 4 has problems if during an analysis it needs to switch from in-core to out-of-core.

The use of the POINT TEMP option was incorrect if optimization was used.

The use of the POINT TEMP option was incorrect for shell elements 22,49 and 72.

POINT TEMP is not working as history definition option.

Sparse iterative solver (type 2) with incomplete Cholesky preconditioner goes wrong after restart. Convergence checking in MAGNETOSTATIC analysis is not correct.

Possible infinite loop in MAGNETOSTATIC analysis with B-H RELATION.

When using AUTO THERM, occasionally one would get one more increment in than analysis than one should.

Using the CYLINDRICAL option for 3D model resulted in incorrect analysis.

Using the CYLINDRICAL option for all nodes in a model, results in transformations being applied to all nodes. The analysis was very slow.

Error when using LORENZI method with either AUTO INCREMENT of AUTO THERM.

Error in AUTO CREEP with prescribed non-zero displacements.

CREEP EXTRAPOLATE did not begin with the correct time, it had an extra timing added on.

NODE SORT option, where the user specified node ranges, in combination with OPTIMIZE did not work.

Changing the post codes during a restart job, if RESTART LAST is used goes wrong. This is not a recommended procedure.

Herrmann or Gap elements may lead to computational problems with sparse solver, Solver type 4. The Lagrange multiplier gets ordered into wrong position. If singularity ratio is less than 1.e-6, switch to Solver type 0.

Gap elements may lead to computational problems with the Sloan optimizer. If singularity ratio is less than 1.e-6, switch to Cuthill McKee optimizer.

Cuthill McKee optimizer error if the number of tries requested is greater than 28.

Program core dumped if sparse iterative solver (type 2) selected and minimum degree optimizer (type 10) selected.

Rezoning with shell elements gave incorrect results.

Rezoning is incorrect for four node and ten node tetrahedral elements.

Incorrect results after performing a REZONE step in a SOIL analysis.

Error in REZONING with reduced integration hourglass elements.

The post file was corrupt if the job was restarted using RESTART LAST and ADAPTIVE meshing was used in the model.

POST code 411 gave incorrect stresses.

Post processing a ACOUSTIC analysis with Mentat, will show a title of displacements instead of pressures. The values are correct.

If the number of post codes was greater than 200, or the number of distributed load lists was greater than 200, there could be an overwrite when creating a binary POST file.

The RESTART analysis potentially core dumps if the number of post codes is greater than 30. Note that tensor codes, internally produce 6 codes.

The nodal results written to an SDRC Universal file are not correct if transformations are present.

The information written to an SDRC Universal file are not correct if nonconsecutive elements or node numbers are used in the model.

When MARC writes a SDRC Universal file with beam elements, SDRC cannot post process it.

I/O error occurred when writing SDRC Universal file on HP computers.

In a CONTACT analysis, if an increment splits, the writing of an SDRC Universal file will be incorrect.

PRINT NODE STRESS gives incorrect stresses for tetrahedral elements.

PRINT ELEMENT NODE gives incorrect results for element type 5.

Print out of messages in output was not correct when values were less than threshold and program switched from relative testing to absolute testing.

There was a problem with the security system if "hostname" could not be found.

Core dump on DEC with SUBSTRUCTURE option.

Cray version required input file to be in lower case. Conversion from upper to lower case did not work.

Routines fsetup.f and intint.f provided in user directory were incorrect on some machines.

PLDUMP program did not work on Windows 95 and NT.

When using the maintain script to create a new executable, if there is no fortran compiler, one will destroy the old executable and end up with nothing. You will need to redo the installation or retrieve from system back-up.

Was unable to read the value of SIZING, if the number of elements was specified on the SIZING line and it was a five digit number.

If the data file name contains uppercase letters MARC will give an EXIT 4004.

Mentat 2.3.1 Problems That Have Been Fixed

Mentat 2.3.1 Problems That Have Been Fixed in This Release

The following problems existed in Mentat 2.3.1 and have been fixed for Mentat 3.1:

Post file sub increments which contained results core dump.

Post file increments which did not have element results core dump.

Curves which are trim surfaces should not have hash marks on them when using the identify contact plotting option. The post rezone option core dumps.

Film boundary conditions were written with incremental ambient temperatures instead of with absolute ambient temperatures, as required by MARC.

Auto creep is improperly written to the history definition of MARC.

Temperature data is written in fields of 10 instead of 15 as required by MARC.

The *system_command, which can be used to issue shell commands, does not work properly.

A divide by zero sometime occurred when computing aspect ratios.

Changing an element class from penta15 to hex20 crashed.

Reading SDRC Universal files with links core dumped.

Tensor values were computed differently in MARC and Mentat due to differences in extrapolation.

Opening a post file after another one was previously opened sometimes crashed due to the second post file having fewer variables available.

The PATRAN Neutral file reader sometimes crashed due to data that was not initialized.

Post codes 7 and 27 had their labels switched, as did 8 and 37.

Using the command line option "-ar0.7" caused a crash.

Overwriting a model file could occur after post processing and accidentally doing a save instead of a restore.

Flipping elements which had boundary conditions on them changed the location of the boundary conditions.

Beam sections parameters would not get written to MARC when the rigid-plastic flow option was used.

Duplication of trimming curves, which should be disallowed, gave erroneous results.

Translation of solids into trimmed surfaces sometimes gave erroneous results or crashed.

Collection of history data during post processing did not reset the increment to that prior to the collection.

Sweep of surfaces having trimming curves did not eliminate duplicate surfaces.

When viewing a new increment of a post file with rezoning, elements previously made invisible would become visible again. The same was true of selected and isolated elements.

Creation of bezier or composite curves sometimes crashed when an empty list of curves was given.

When displaying history plots, the values at the nodes are not taken from the isolated elements, but from the non-isolated elements, for those nodes which are part of both isolated and non-isolated elements.

The MARC reader is not able to read free format fields.

The collection of history data from a multiple domain post file sometimes crashed.

Nodes sometimes were not visible if they were not part of an existing element.

Contours of models from adaptive analyses were sometimes discontinuous.

When writing a MARC input file angular velocities could have the wrong sign. Also, the variation of angular velocities using a table did not work.

Planar quad meshing in some cases crashed.

The overlay mesher could not properly mesh curves when the local coordinate system was not aligned with the global coordinate system.

Node ids were not unique after a merge of model files.

An infinite loop may occur when writing a MARC input file while using adaptive time stepping in a coupled analysis. Reading MARC input files into Mentat sometimes crashed.

Solitary nodes were sometimes not visible during post processing.

The RELEASE NODE option in the MARC input file was missing the number of sets on the next card.

The yield stress at zero degrees instead of a the lowest temperature was written to the MARC input file ISOTROPIC option.

Creep strain and creep stress were switched in the MARC input file.

Postscript plotting sometimes gave bad results when surfaces or elements were close to being coplanar.

Nodal temperatures were not read properly from universal files.

Some post files crashed when stepping to the first increment.

Shell thickness was not read in from Patran Neutral files.

At times the triad in the graphics window was incorrect.