3

# Modeling and Meshing Enhancements

- Rapid Surface Meshing
- Automatic Hard Points Creation
- Curvature Based Tetrahedral Meshing
- Tetrahedral Element Modification
- MSC/PATRAN LAMINATE MODELER Enhancements
- Fillet Curve Improvement

# 3.1 Rapid Surface Meshing

MSC/PATRAN Version 8 includes a rapid surface meshing process to create a single large surface over a collection of potentially discontinuous surfaces. This new surface, called a tessellated surface, can then be meshed, resulting in significantly fewer elements and higher quality.

When you mesh imported CAD geometry, the mesh conforms to the existing CAD model boundaries. For models with complex surfaces, such as airframes and automotive bodies, the resulting mesh may contain:

- too many elements.
- poor quality elements associated to degenerate geometry, such as sliver surfaces.
- adjacent meshes that do not share common boundaries.

These factors can lead to inaccurate results and inefficient analyses. Figure 3-1 shows imported CAD geometry containing gaps and sliver surfaces.

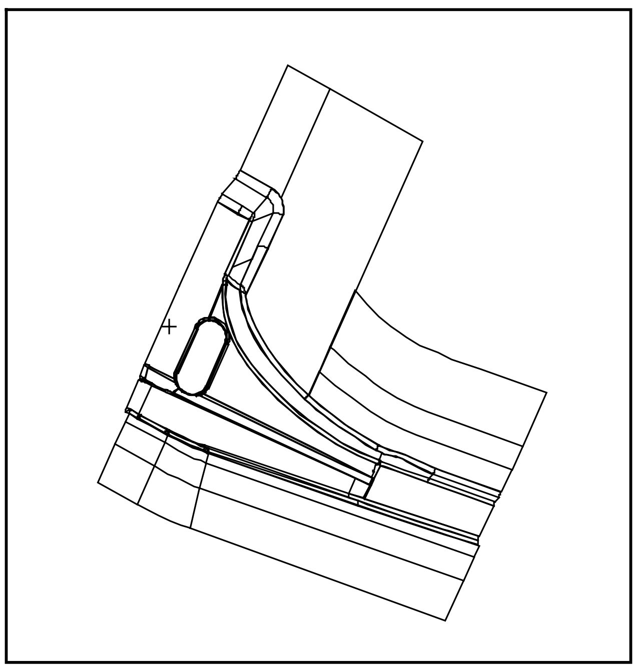


**Figure 3-1 Imported Geometry with Gap and Sliver Surface Discontinuities.**

The rapid surface meshing process quickly and easily:

- creates a continuous mesh from the CAD geometry, as shown in Figure 3-2.
- creates a tessellated surface from the continuous mesh.

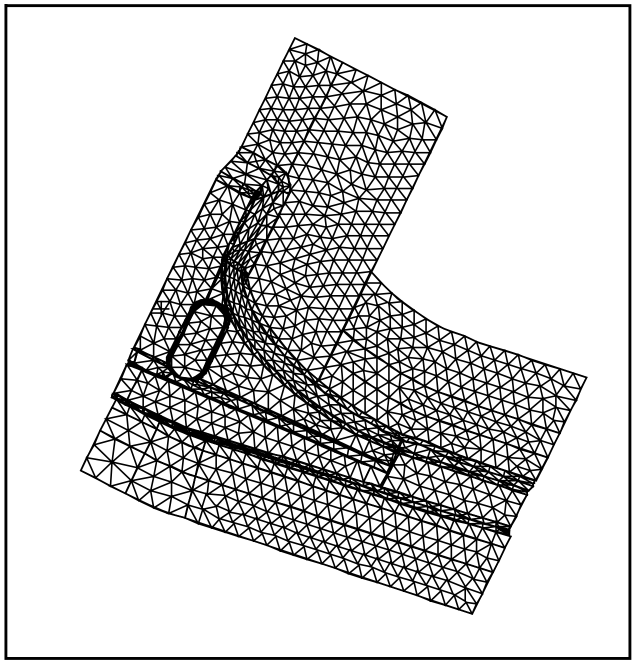


**Figure 3-2 Continuous Tria3 Mesh.**

You can then mesh the tessellated surface to create a high quality analysis model from the initial complex and dicontinuous CAD geometry, as shown in Figure 3-3.

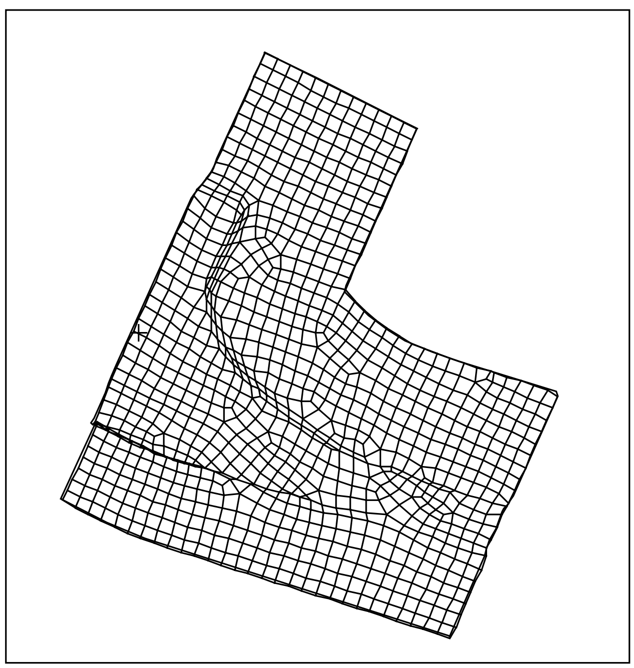


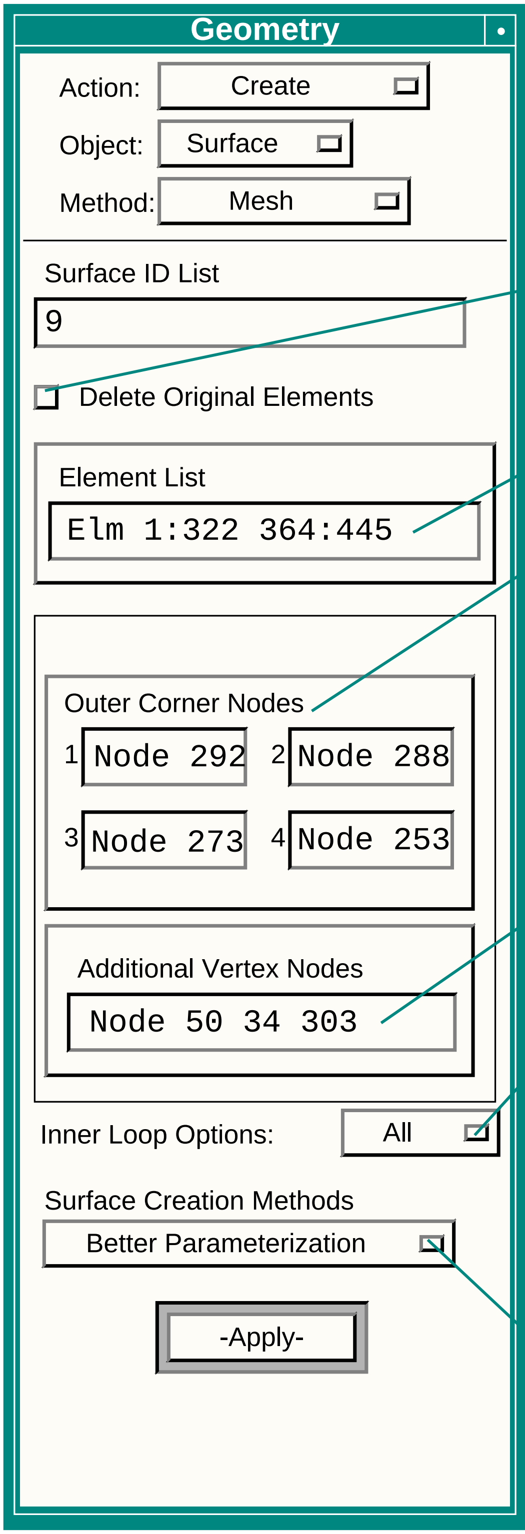
**Figure 3-3 Tessellated Surface with Mesh.**

### Additional Application of Tessellated Surfaces

Tessellated surfaces can be used to modify existing finite element models. For example, if an initially coarse model needs to be refined, a tessellated surface can be created from the original elements. This can be particularly valuable when importing an archived analysis model for design changes. Tria and Quad element types are supported.

## Tessellated Surface User Interface

Creating a tessellated surface is available from the Create / Surface / Mesh form under Geometry. An example of the form with explanations of its fields is shown below.




If toggled ON, the elements selected will be deleted when the surface is created.

Congruent element list that defines the surface.

Select four corner nodes that will define the four vertices of the resulting green surface or the parent surface of a trimmed surface. This input is not required. In general, leave these databoxes blank for automatic calculation.

If there are more than four vertices for the surface, the additional nodes can be listed in the Additional Vertex Nodes listbox. This can be helpful for connecting the surface to an adjacent surface edge.

By setting Inner Loop Options to All, None or Select, the holes in the resulting surface can be defined.

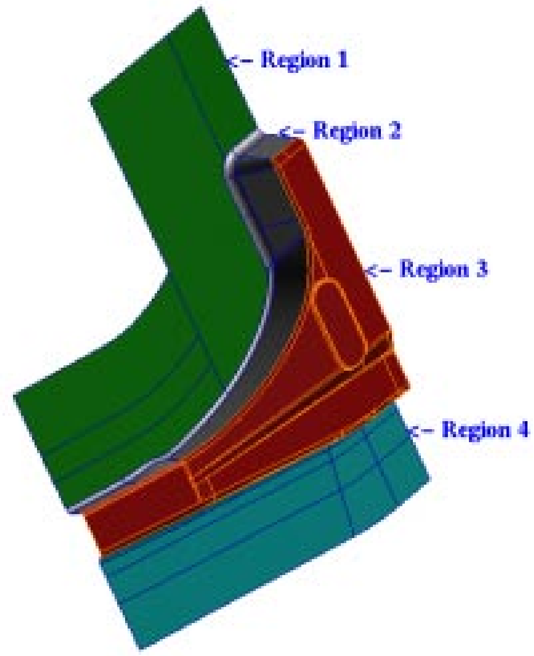
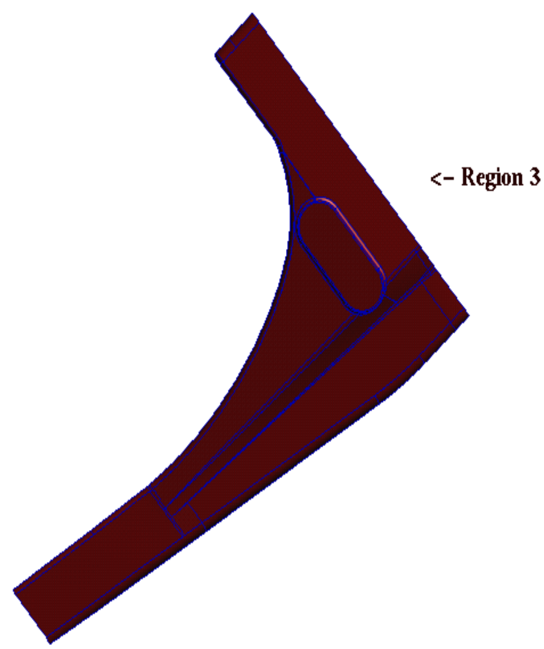
**Note:** When the Inner Loop Options is set to Select, a node listbox opens. Here any hole to be preserved can be identified by selecting at least one node on its edge. Any nodes not on the hole edge or on the outer boundary will be ignored.

By selecting the surface creation option, emphasis can be placed on parametrization or speed. For automatic meshing of the surface (i.e. Paver), “Fast” will usually suffice, otherwise use “Better Parameterization”.

## 3.2 Rapid Surface Meshing Procedure

The following table summarizes the rapid surface meshing procedure. Each step is described in detail on the following pages.

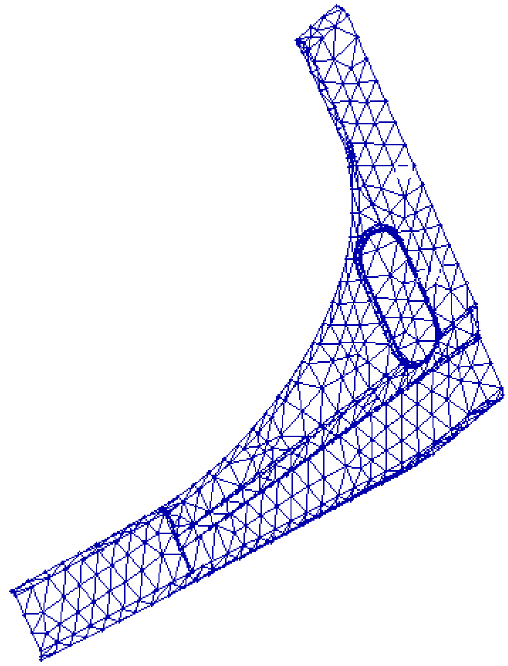
| Steps | Menu Selection |
|---|---|
| 1. Group the geometry (recommended, but not required). | Group / Create |
| 2. Display one group. | Group / Post |
| 3. Create the initial tria element mesh. | Finite Elements / Create / Surface / Mesh |
| 4. Review the mesh. | View / Select Corners |
| 5. Sew to create a continuous mesh. | Finite Elements / Modify / Mesh / Sew |
| 6. Verify the mesh boundary. | Finite Elements / Verify / Element / Boundary |
| 7. Edit the mesh (if required). | Finite Elements / Delete / Element or Delete / Mesh / Surface |
| 8. Create the tessellated surface from the continuous mesh. | Geometry / Create / Surface / Mesh |
| 9. Mesh the tessellated surface with quad elements. | Finite Elements / Create / Mesh / Surface |
| 10. Repeat the steps above for each group. | |

| Step 1. Group the geometry (recommended, but not required). | |
|---|---|
| Menu Selection: Group / Create |   |
| Group geometry regions such that boundaries are created where sharp changes in slope would otherwise result in warped elements. Skipping this step and working with too many surfaces at once can result in meshing difficulties. | |
| **Step 2. Display one group.** | |
| Menu Selection: Group / Post |   |
| Start with the group with the most complex surfaces. The image shows a group containing 66 surfaces and is particulary complex due to the gaps, overlaps, and sliver surfaces. | |

## Step 3. Create the initial tria element mesh.

Menu Selection:
Finite Elements / Create / Surface / Mesh

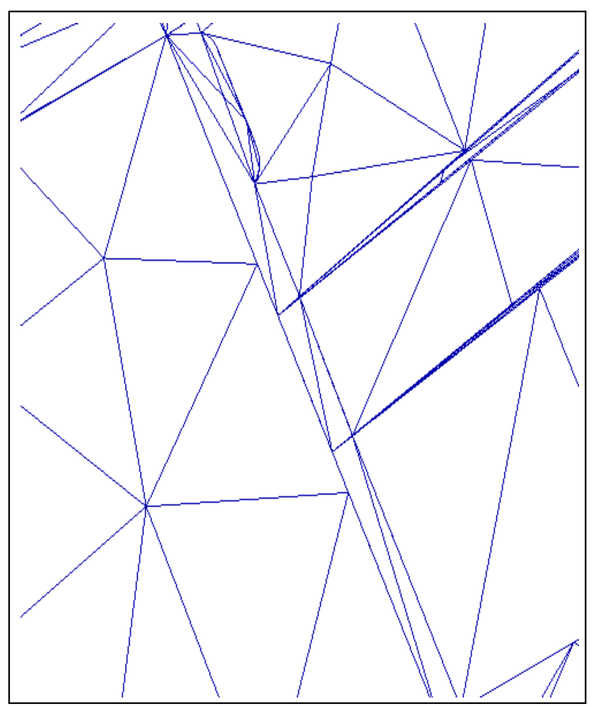
Use an element size that represents the geometric shape accurately enough for the particular application.

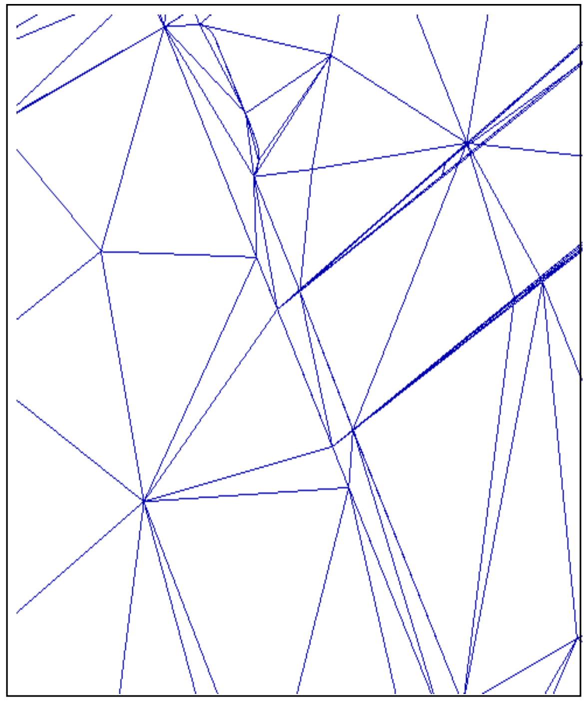


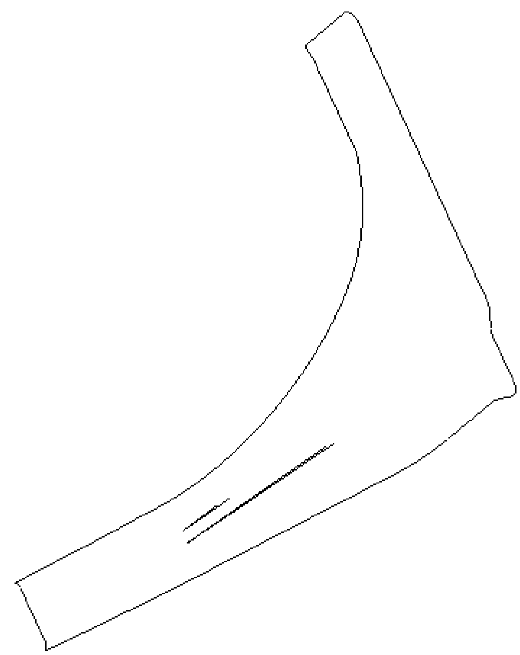
## Step 4. Review the mesh.

Menu Selection: View / Select Corners

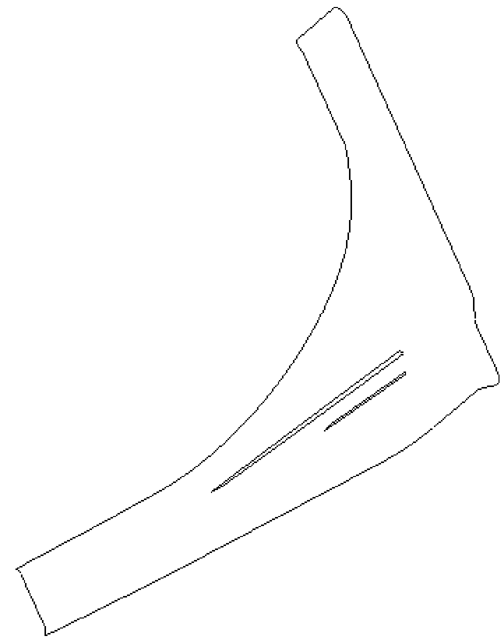
Note how adjacent element edges do not match up.



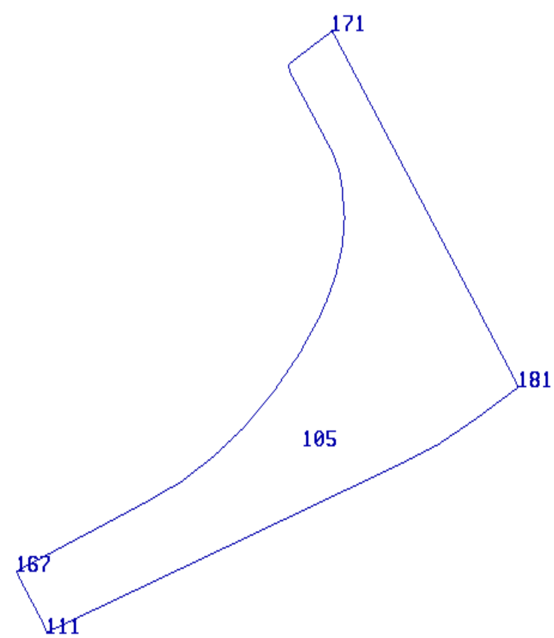
| Step 5. Sew to create a continuous mesh. | |
|---|---|
| Menu Selection:<br>Finite Elements / Modify / Mesh / Sew |  |
| Creates continuity from a potentially discontinuous mesh. Sewing performs nodal equivalencing and adds tria elements to fill small cracks. Elements are added to create continuity and are not intended for direct analysis since they are frequently sliver elements. Use a “Target Element Edge Length” of approximately the size you will use for quad meshing the resulting tesselated surface. The larger this value, the more the algorithm will assume that any small cracks should be sewn together. | |

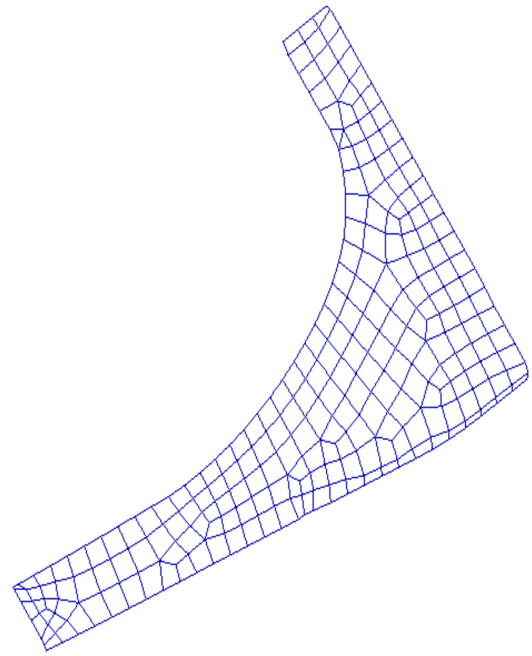
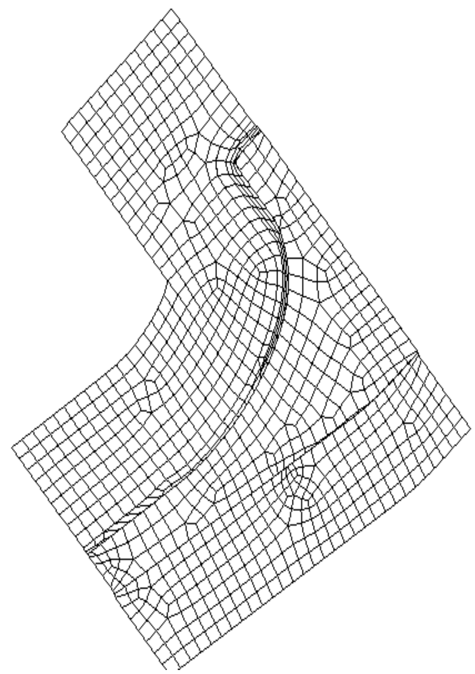
| Step 6. Verify the mesh boundary. | |
|---|---|
| Menu Selection:<br>Finite Elements / Verify / Element / Boundary |  |
| If internal cracks (unshared internal element edges) remain, often due to surfaces that significantly overlap each other, zoom into the cracked regions and delete elements in the overlapped region. | |

**Step 7. Edit the mesh (if required).**

| | |
|---|---|
| Menu Selection:<br>Finite Elements / Delete / Element<br>or<br>Finite Elements / Delete / Mesh / Surface |  |
| The simplest way to create continuity is to delete elements in the cracked region and create the tessellated surface (described below) over the hole created by the deleted elements. The image shows a boundary verification after the mesh is deleted from the overlapping surfaces. | |

**Step 8. Create the tessellated surface from the continuous mesh.**

| | |
|---|---|
| Menu Selection:<br>Geometry / Create / Surface / Mesh |  |
| The new surface can be meshed with high quality elements, independent of the internal CAD boundaries, and poorly shaped surfaces. This step requires continuity of the FEM mesh. Holes do not break the continuity but only two element edges may share any node pair. If this is violated, the offending nodes will be displayed. In this case, use:<br>List / FEM / Element / Association<br>Erase or delete the elements associated to the returned node list, then create the surface.<br>If the tessellated surface will share adjacent edges with another tessellated surface, be sure to select the “Outer Corner Nodes” to include these at the adjacent edge vertices. Otherwise, the mesh may not match at the edge. Set “Inner Loop Options” to “None” to fill in any undesired holes in the mesh. The “Additional Vertex Nodes” is optionally used for adding another vertex in the resulting surface. This can be useful if an adjacent surface has a vertex at this location to create a continuous mesh at the adjacent faces. | |

| Step 9. Mesh the tessellated surface with quad elements. | |
|---|---|
| Menu Selection:<br>Finite Elements / Create / Mesh / Surface |  |
| Use the Paver algorithm. See the Tips and Limitations section of the Version 8 Release Guide for meshing issues. | |
| **Step 10. Repeat the steps above for each group.** | |
| Displayed is the final mesh independent of the original CAD boundaries, sliver surfaces, and gaps. |  |

**Rapid Surface Meshing Tips and Limitations**

- A tessellated surface will exactly match the original CAD geometry at every node point. To more closely match the original CAD representation, create a finer initial mesh in the region of interest.
- When learning to work with the Rapid Surface Meshing process, do not attempt to create a single tessellated surface over a large number of surfaces (i.e. hundreds). Start with a tessellated surface for every 10-50 CAD surfaces until you gain experience using the sewing tools.
- To accurately capture sharp changes in interior curvature and avoid skewed quad elements, create tessellated surfaces such that the edges lie on boundaries that are parallel with the lines of curvature.
- If problems occur during meshing of highly complex surfaces containing sharp curvature changes, create more tessellated surfaces to represent the same region.
- When turning on the geometric display lines (Display-Geometry), the display lines of tessellated surfaces may not be smooth. This is an artifact typical of tessellated surfaces. When meshing with the Isomesh algorithm, the resulting mesh will follow these lines. Smoothing the mesh may help or creating another tessellated surface from the current mesh may improve the quality.
- If the Paver meshing algorithm returns a warning message "Unable to mesh in global space, meshing in parametric space", the resulting mesh may be of lower quality or may not complete successfully. Suggested options are:

  a) Use Modify / Mesh / Surface under Finite Elements for smoothing.

  b) If the quality is still not satisfactory, create more tessellated surfaces over the same region.

  c) Recreate the tessellated surface by using the Better Parameterization meshing option.

  d) Recreate the tessellated surface by picking four corner nodes that represent the approximate corner nodes of the surface. This may help the underlying parameterization of the surface
- If two adjacent tessellated surfaces appear to share a common edge, but the resulting mesh does not line up, then the edges are not congruent. To attain congruency

in mesh boundaries, recreate one of the tessellated surfaces by picking the four corner nodes. Two of these nodes should match the adjacent surface boundary nodes. Or the nodes of the adjacent mesh can be used as mesh seeds using the option Create / Mesh Seed / Tabular / Node and Point.

- Hard curves have limited support for Version 8. In some cases, the meshing success rate will be lower.
- Avoid using more than 10,000 elements when creating a surface. The performance of geometry and FEM operations on this surface will degrade. In such situations, create several surfaces using fewer number of elements per surface.
- A tessellated surface edge that shares a common boundary with an adjacent surface will recognize the adjacent mesh as a mesh seed to provide mesh continuity. If the Delete Original Elements toggle on the Create / Surface / Mesh form is off, then the element size ofthe original tria elements edges that share a common surface boundary will also be interpreted as a mesh seed.
- Solid shading of a tessellated surface in Version 8 does not accurately represent the curvature of the surface. Solid shade the resulting mesh instead.

## 3.3 Automatic Hard Points Creation

Using the Create / Mesh Control / Auto Hard Points form under Finite Elements creates hard points on a specified set of surfaces automatically. This program creates hard points at two kinds of points on surface boundaries:

- T Points
- Neck Points

These types of hard points are discussed in the following sections.

**T Points**

T points are interior points of a surface edge which are close to a vertex or an existing hard point on an edge in another surface within the T-point tolerance. The T-point tolerance is equal to one twentieth of the target element edge length.

Placing a hard point at a T-point helps to create a congruent mesh on a noncongruent model. A noncongruent model is shown in **Figure 3-4**.

The auto hard point creation program creates a T-point at the T-junction of three surfaces and marks it by a small triangle. The new hard point forces the mesher to place a node at the T-junction when meshing surface 1 and the mesh created is a congruent mesh (**Figure 3-5**). **Figure 3-6** shows the mesh without hard point creation.

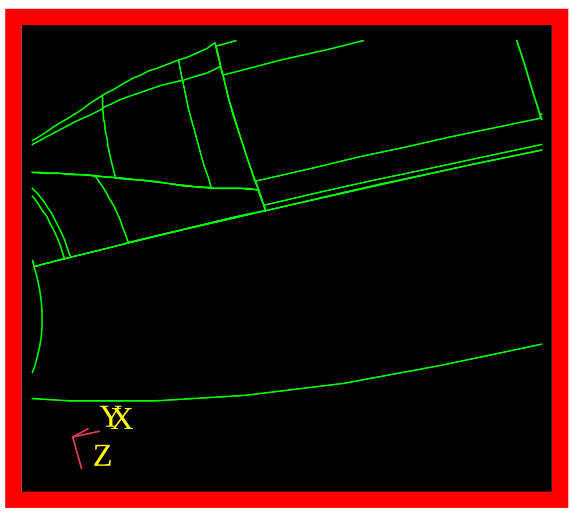



**Figure 3-4 Incongruent Surfaces Demonstrating Need for Hard points**

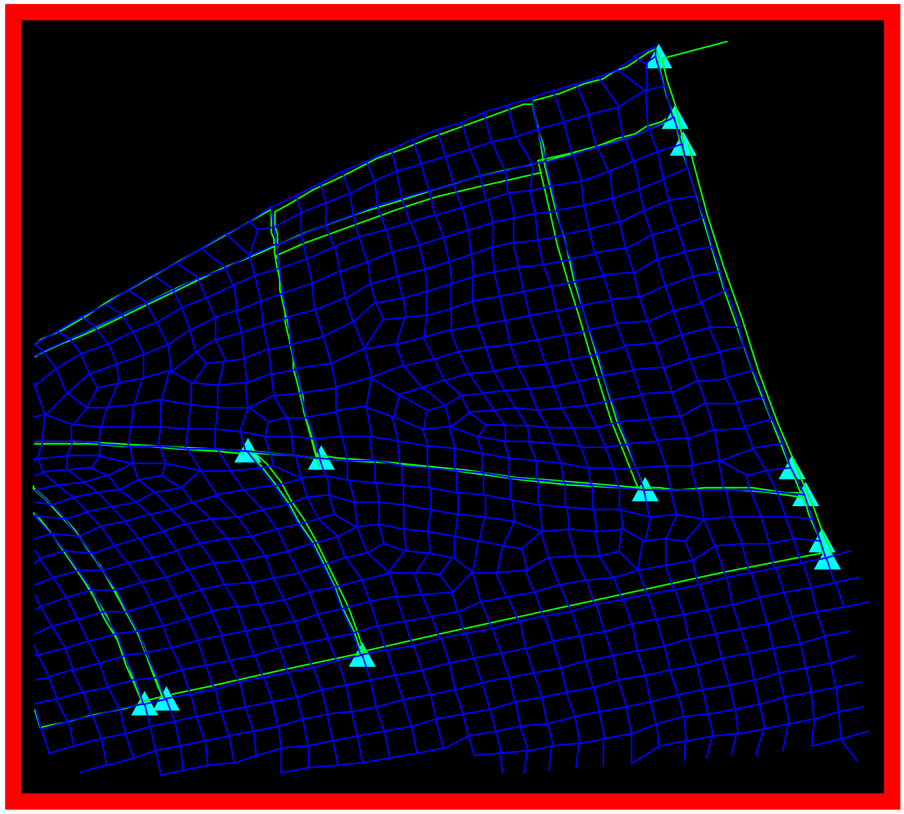


Figure 3-5 Mesh with Hard Point Creation Resulting in a Congruent Mesh

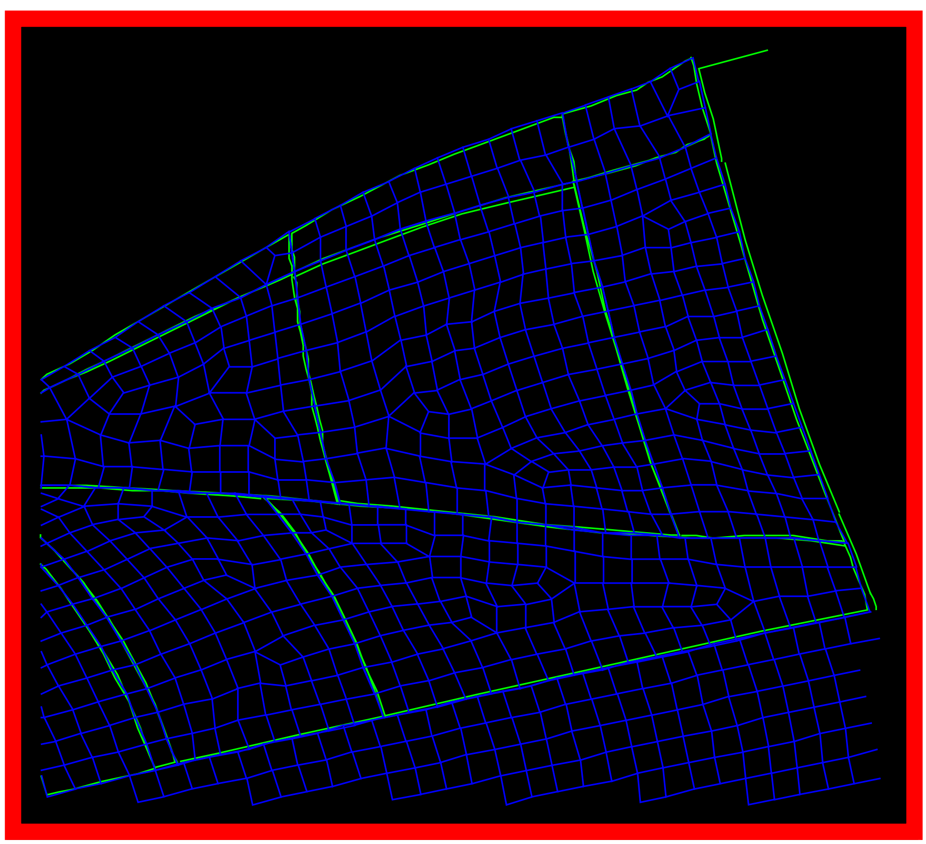


Figure 3-6 Mesh Without Hard Point Creation

## Neck Points

Neck points are end points of a short cross section on a surface. A cross section on a surface is short if its length is less than the neck-point tolerance. The neck-point tolerance is equal to 1.5 times the target element edge length.

Placing a hard point at a neck point creates better meshes on narrow surfaces. Neck points can be created recursively by neck-point propagation.

In **Figure 3-7**, the two neck points on the boundary of surface 1 were created first and the remaining four neck points were created by neck point propagation from one small surface to another until the path reached the outer boundary of the model.

The new hard points will force the mesher to line up the boundary nodes and create a good mesh on the narrow surfaces (**Figure 3-8**). **Figure 3-9** shows the mesh without hard point creation.

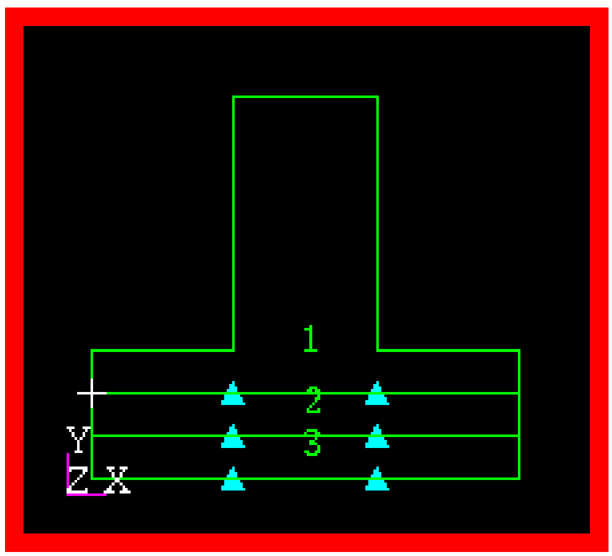



**Figure 3-7 Surface with Auto Hardpoints**

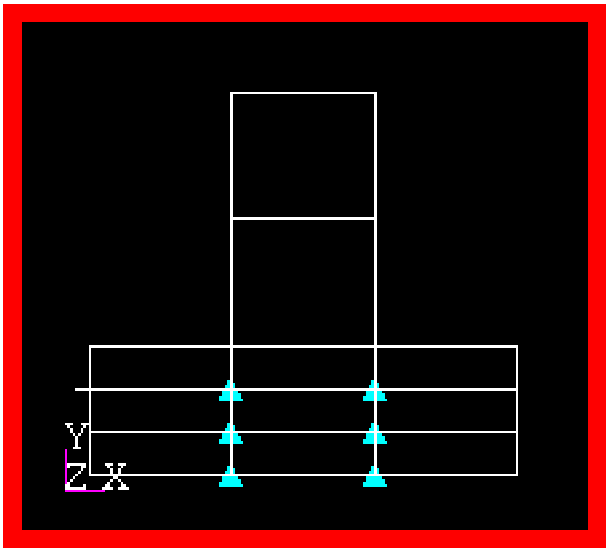



**Figure 3-8 Mesh with Hard Point Creation**

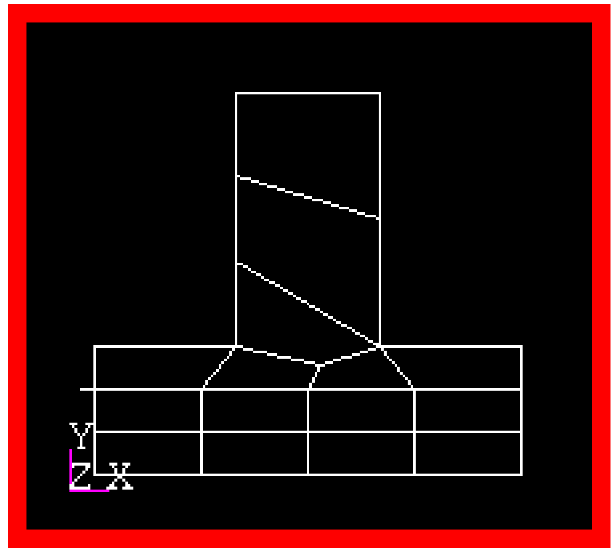
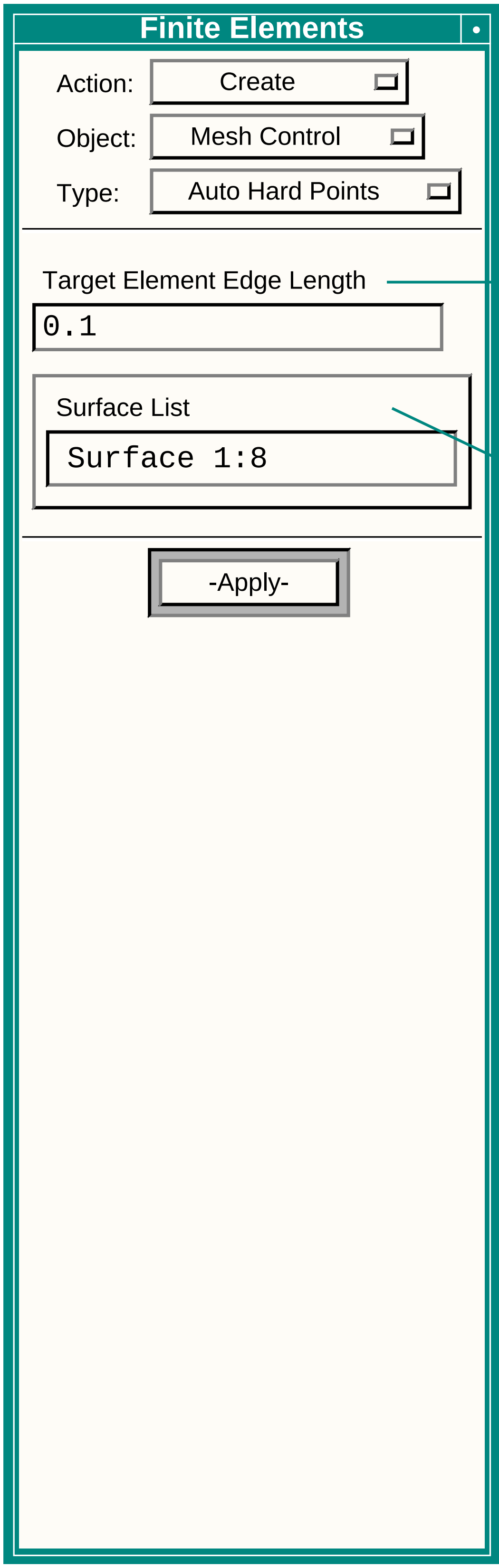



**Figure 3-9 Mesh Without Hard Point Creation**

More




Specify the target element edge length which you will use as a global edge length when you mesh the model. The target element edge length is used to compute the T-point tolerance and neck-point tolerance.

Specify a set of surfaces on which to create hard points, either by cursor selecting them or by entering the IDs from the keyboard.

# 3.4 Curvature Based Tetrahedral Meshing

Mechanical component failure frequently occurs in areas of high curvature. For accurate analyses of these components, you typically need to specify finer surface meshes in curved regions of a model before automatically meshing a part with a coarser element size. This time consuming process has been replaced with curvature based tet meshing that automatically refines the mesh in areas of high curvature. You can control how much refinement occurs in each curved region.

Refinement is limited to 1/5 of the global element size to avoid excessive element creation at near 90-degree angles. Figure 3-10 shows a model with curvature based refinement.

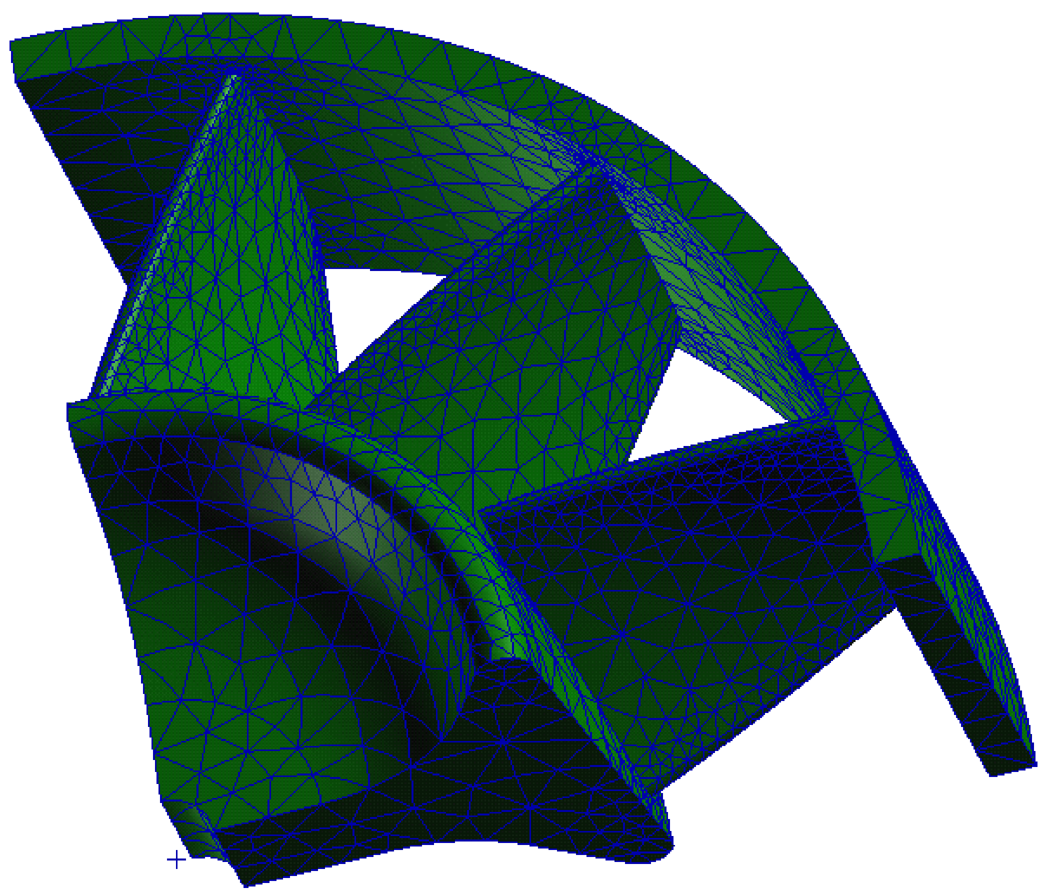


**Figure 3-10 Automatic Curvature Based Refinement**

# 3.5 Tetrahedral Element Modification

Version 7.5 of MSC/PATRAN added a new tool for improving Tetrahedral (Tet) meshes by locally remeshing to remove collapsed elements. Version 8 introduces an additional tool integrated into the same form, which locally remeshes to eliminate elements with negative jacobians. Both tools are used simultaneously to avoid degrading the mesh.

When creating a default Tet mesh in MSC/PATRAN, it is possible to generate an element with a negative jacobian due to complex geometric constraints. To allow the element to pass the quality verification process, the element edge is automatically straightened by the mesher, eliminating the negative jacobian. However, these element edges no longer follow the geometric curvature of the solid model.

To avoid this problem, applying local mesh control with a smaller element size and re-meshing may prevent straightening. Another option to avoid edge straightening when negative jacobians are encountered, is to activate the toggle "Create P -Element Mesh" (located in the Create/Mesh/Solid-tetmesh Parameters subform). Then use the new functionality to fix the Tet mesh. This new form (Modify/Mesh/Solid) can be used to locally remesh if negative jacobian elements are encountered.

# 3.6 MSC/PATRAN LAMINATE MODELER Enhancements

This section describes the enhancements to the MSC/PATRAN LAMINATE MODELER for Version 8.

**Support for Windows NT**

MSC/PATRAN LAMINATE MODELER now runs on all platforms supported by MSC/PATRAN, including Windows NT. The module is also implemented as a shared library, facilitating the delivery of incremental upgrades and reducing hardware requirements.

**Enhancements of Draping Simulation**

The draping simulation has been enhanced as follows:

- The definition of the principal axes of draping and the extension method to be used once draping extends past the principal axes can now be specified individually. For example, geodesic principal axes may be specified together with extensions oriented to minimize shear strain energy.
- Draping can proceed on separate surfaces in a defined order. For example, if a structure consists of two intersecting cylindrical surfaces, the first surface can be draped completely with zero resultant shear, followed by draping on the second surface in a manner which is compatible with the draped pattern on the first surface.

**Results Sorting on the Basis of Ply Material ID**

Results sorting on the basis of physical plies can now be done if these plies can be identified through the use of a unique material ID. This complements the material sorting capabilities already available if the model is built up from plies within the MSC/PATRAN LAMINATE MODELER.

**Laminate Generation Using Coordinate Systems and Vectors**

The user was previously given the option of generating laminates when a layup is created. Now, a separate “Create Laminates” function has been added to allow more control over the laminate creation process. In particular, the orientation of the laminates can be controlled using different methods as allowed by the appropriate Analysis Preference. The prefix and ids of the required laminates and properties can be specified, and the necessary number of laminates, properties and coordinate frames previewed. A summary of the options for each Analysis Preference is given in the table below:

| Analysis Preference | Laminate Orientation |
|---|---|
| MSC/NASTRAN | Default (along first edge of element)<br>Angle (from first edge of element)<br>Vector<br>Coordinate System |
| ADVANCED FEA | Default (projection of X or Z axes)<br>Coordinate System |
| ABAQUS | Default (projection of X or Z axes)<br>Coordinate System |
| ANSYS | Default (along first edge of element) |

**Modification of Materials**

The user can now modify LM_Materials, even if these are referenced by plies. This is useful for conducting "what-if" investigations into the use of alternative materials.

**Import of Plies from External Programs**

Users can import plies from Layup files generated in previous MSC/PATRAN LAMINATE MODELER sessions, or by third party software (e.g. FiberSIM). This can be used even if the mesh in the external file is different to that in the MSC/PATRAN database. This also allows the user to remesh the model after the initial Layup file is generated.

**Ply Book Creation**

A ply book can now be created from a Layup at any time. This consists of a summary report, postscript files showing application on the model, and DXF files defining the flat or draped patterns. The user can customize the amount of data generated.

**Minor Enhancements**

There are a number of minor enhancements to the user interface, such as the ability to show a ply from the Layup spreadsheet.

# 3.7 Fillet Curve Improvement

Version 8 creates more accurate fillet curves. This improved capability is available on the Create / Curve / Fillet form in the Geometry application. The form itself has not been modified, but the PCL enabling this fuctionality has.

In previous versions, the curves were created using an asm definition. The asm_const_line_fillet PCL command was used to generate the fillets. This PCL command is upward compatible, so that existing session files played in Version 8 will generate the same fillet as previous versions. In Version 8, the fillet is generated using the sgm_const_curve_fillet PCL command, which uses a more accurate sgm definition.

For detailed information on this capability, see Create Actions (Ch. 4) in MSC / PATRAN User's Guide, Part 3: Geometry Modeling. As documented, the endpoints in the Curve / Point evaluation are used to determine what part of the curves you want to keep if trimming is desired.