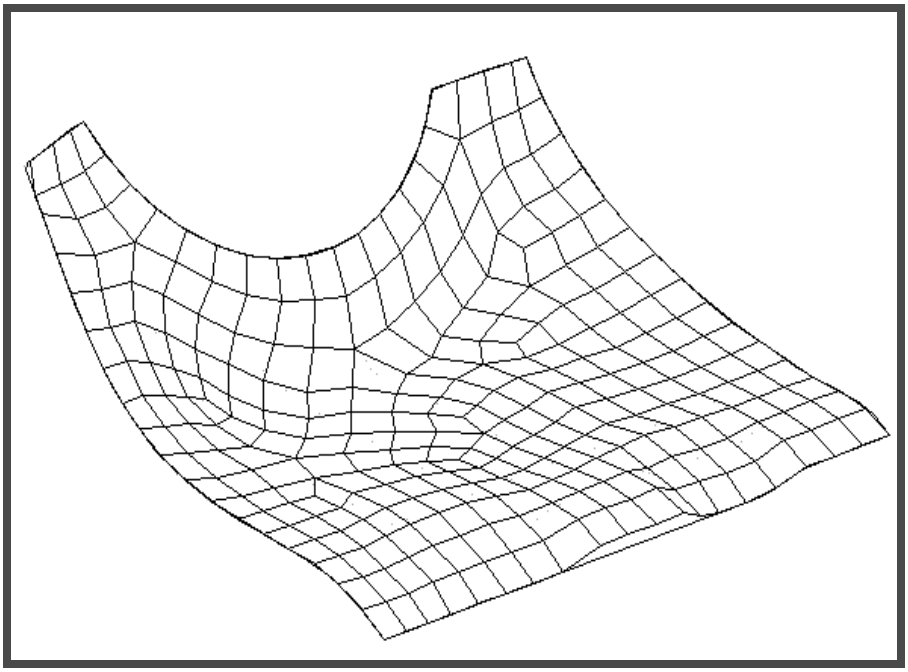
LESSON 12

Rapid Surface Meshing



Objectives:

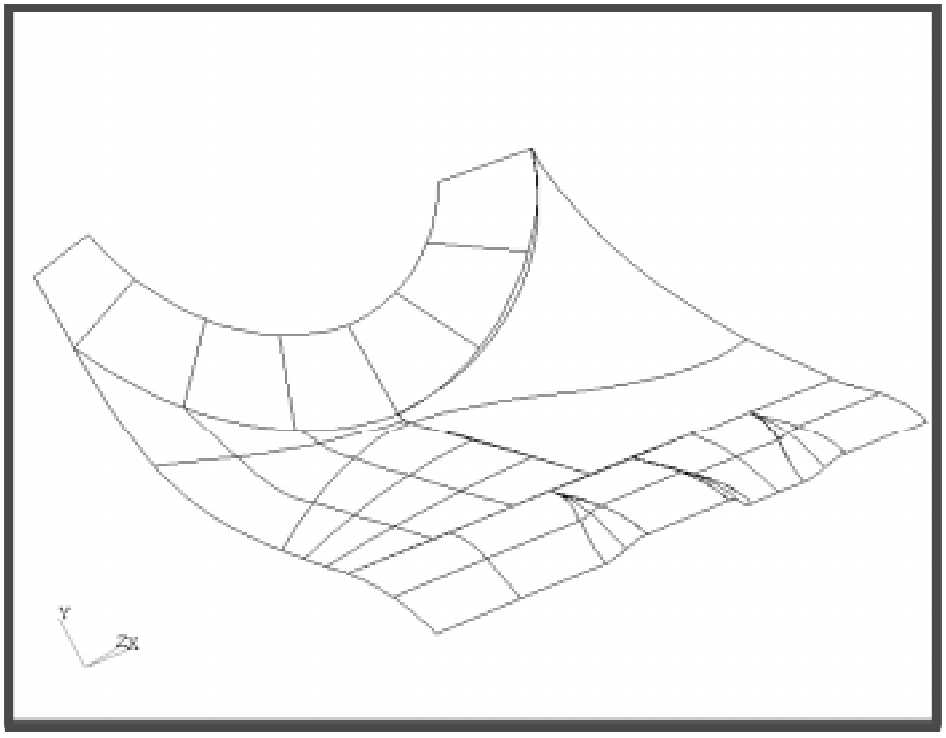
- Import a geometry from neutral file.
- Mesh the geometry and eliminate gaps.
- Recreate the geometry from the meshes and remesh.

Model Description:

In most cases, finite element models are created with several component geometries. The purpose of this exercise is to model an oil pan, which is composed of several surfaces, with just one surface and mesh it.

Figure 1

Imported Geometry



Suggested Exercise Steps:

- Create a new database called **final.db**.
- Import the neutral file called **oilpan_prt.out.1**.
- Check the size of the model and select the global edge length.
- Mesh the surface with tri elements.
- Close the gaps with the **Modify/Mesh/Sew** option.
- Delete the original geometry before creating the new one.
- Create a surface from existing meshes that have gaps and small holes.
- Remesh the surface with Quad4 elements.

Exercise Procedure:

1. Create a new database called **oilpan.db**.

   File/New Database

   New Database Name: **final**

   OK

   In the *New Model Preference* form, set the following:

   Analysis Code: **MSC/NASTRAN**

   Analysis Type: **Structural**

   OK

2. Import the neutral file called **oilpan_prt.out.1**.

   File/Import...

   Change to the directory that contains the file **oilpan_prt.out.1.**

   Neutral Files: **oilpan_prt.out.1**

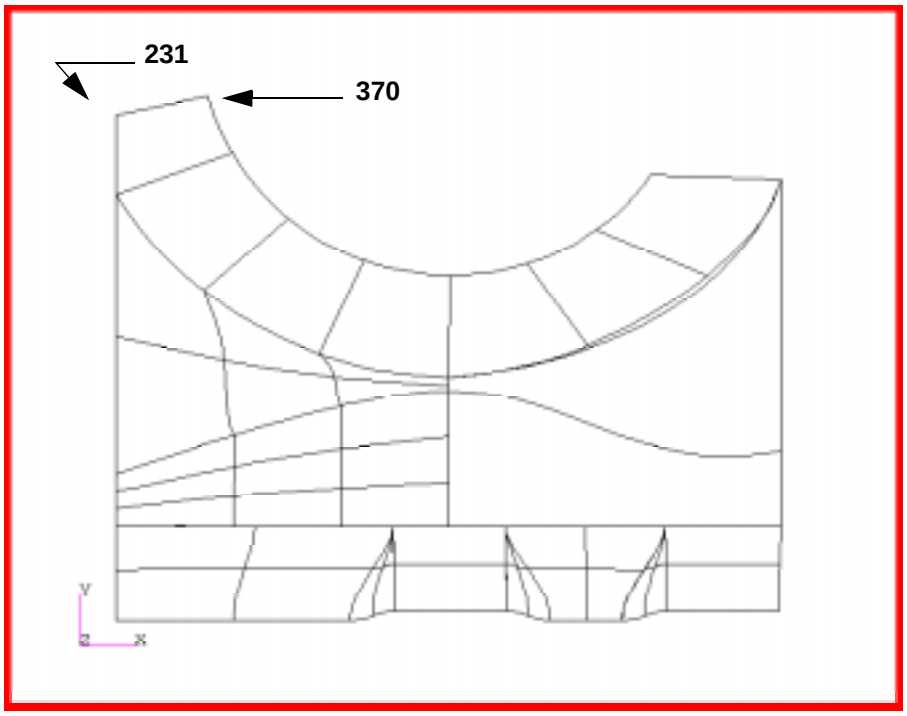
   Apply

   Yes

   When asked if this model should be committed to the database, select **YES**.

   Yes

Your display should look like the following:



3. Check the size of the model and select the global edge length.

   Prior to meshing, the proper global edge length should be determined.

   ◆ **Geometry**

Action:	**Show**
Object:	**Point**
Info:	**Distance**

   ❑ **Auto Execute**

   Select the two points in the upper left, shown in the above figure.

First Point List:	**Point 231**
Second Point List:	**Point 370**

   Apply

   A *Show Point Distance Information* Menu should appear. It shows that the distance between these two neighboring points is approximately 15. Thus, a good global edge length would be 2, for it would produce a mesh dense enough for our purpose.

4. Mesh the surface with tri elements.

   These tris will not be used directly for analysis. The purpose is to create a continuous tri representation over potentially discontinuous CAD geometry.

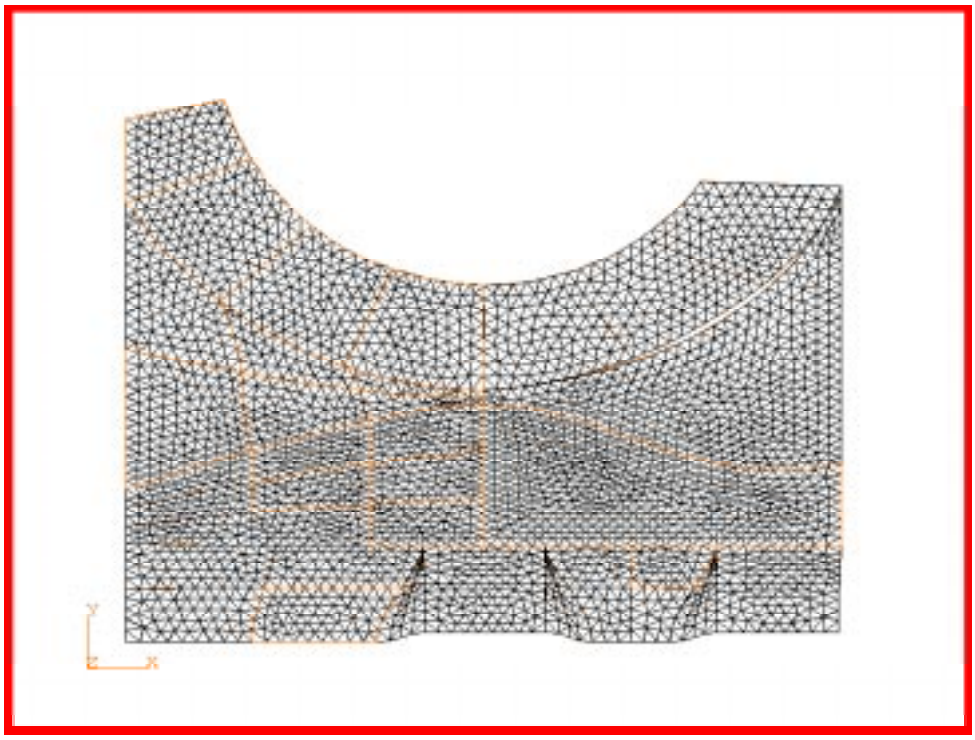
   ◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length:	**2**
Element Topology:	**Tria3**
Mesher:	◆ **Paver**
Surface List:	**<Select All Surfaces>**

   Apply

   The Warnings Menu will appear with information about mesh status.

   Your display should look like the following:



5. Close the gaps with the **Modify/Mesh/Sew** option.

   The usual step after meshing is to equivalence coincident nodes. However, since there are larger gaps that equivalencing will not be able to eliminate, the **Modify/Mesh/Sew** option will be used. The Sew option will automatically equivalence all coincident nodes at the same time close up element gaps that are within the Element Edge Length specified.

   ◆ **Finite Elements**

Action:	**Modify**
Object:	**Mesh**
Type:	**Sew**
Target Element Edge Length:	**6**
Tria Element List:	**<Select all elements>**

   Apply

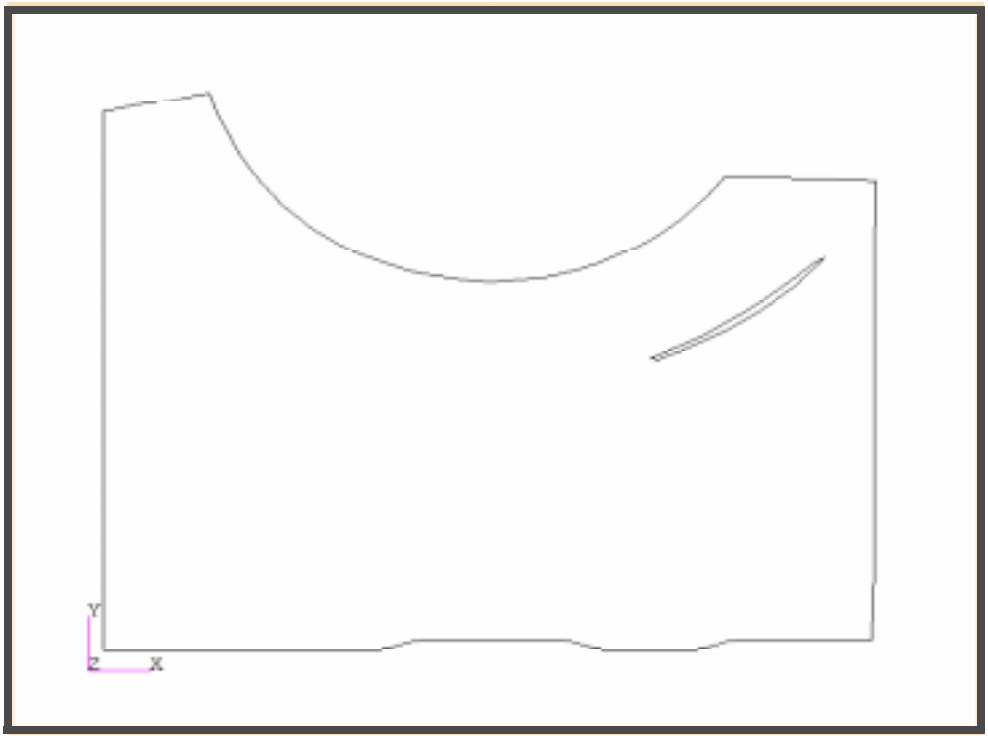
6. Verify the element boundary edges.

   ◆ **Finite Elements**

Action:	**Verify**
Object:	**Element**
Test:	**Boundaries**
Display Type:	◆ **Free Edges**

   Apply

Your display should look like the following:



Higher Target Element Edge Length will close larger gaps. The users will see that a value of 20 will close up this gap completely. Nevertheless, using a higher value may run the risk of collapsing the elements.

Models, which have gaps that are too large to be sewn without collapsing elements, will require a different approach. To demonstrate this point, we will close the last gap without using the sew option.

7. Delete the original geometry for visual clarity. (optional)

◆ **Geometry**

Action:	**Delete**
Object:	**Surface**
Element List:	**<Select all Surfaces>**

Apply

8. Create a surface from existing meshes that have gaps and small holes.

Create/Surface/Mesh allows the user to create a surface from finite elements. The Inner Loop Options allows the user to either eliminate all or selected gaps/holes within the surface. Users who wish to do Isomesh or have finer meshes should set Surface Creation Method to Better Parametrization. Although this will greatly increase the processing time, the resulting surface will be more refined.

We will set **Inner Loop Options: None**, so there will be no gaps within the model. Also, we will set **Surface Creation Method: Fast**, because we will use Paver mesh to mesh the model and this will expedite our process.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Test:	**Mesh**

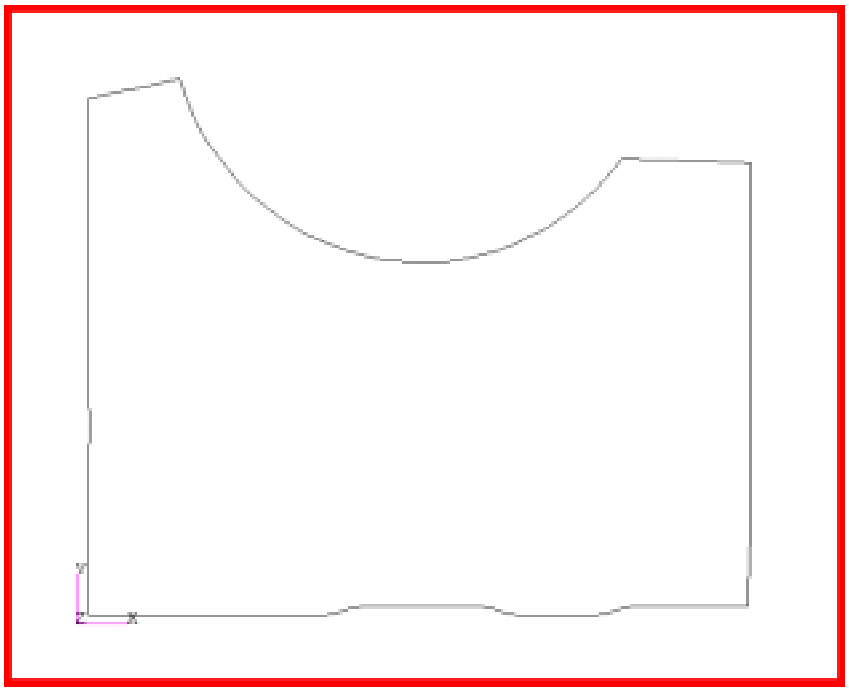
■ **Delete Original Elements**

Element List:	**<Select all elements>**
Inner Loop Options:	**None**
Surface Creation Methods:	**Fast**

Apply

All the gaps should now be closed and only one continuous surface should be in your display.

The model should appear as follows:



9. Remesh the surface with Quad4 elements.

◆ **Finite Elements**

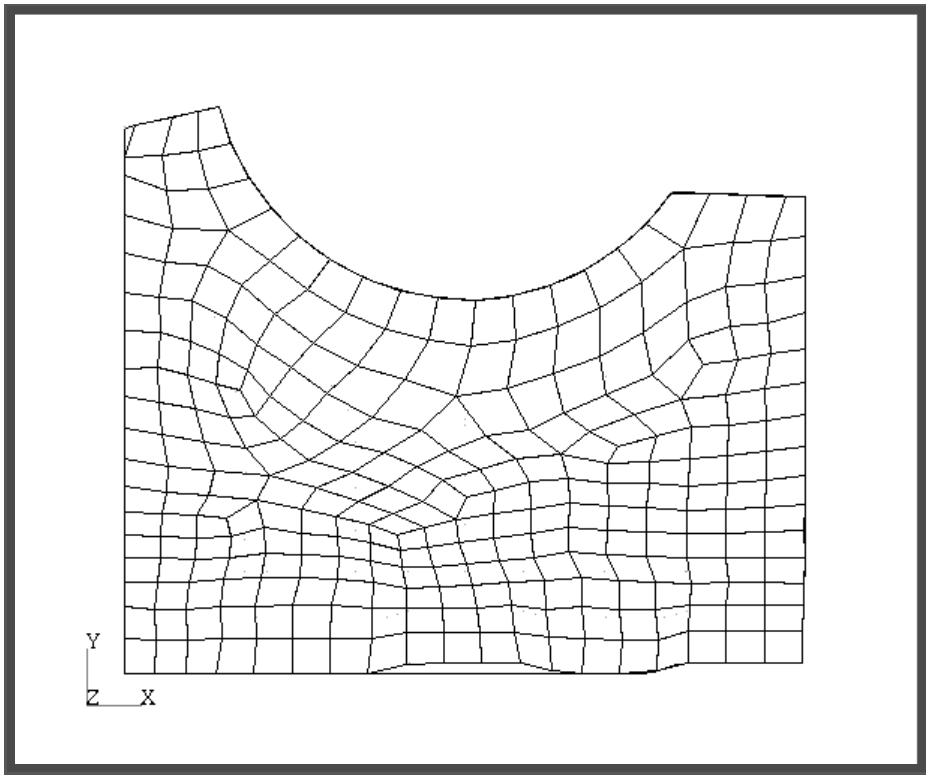
Action:	**Create**
Object:	**Mesh**
Global Edge Length:	**6**
Element Topology:	**Quad4**

◆ **Paver**

Surface List:	**<Select the Tessellated Surface>**

Apply

The model should appear as follows:



This concludes this exercise.

To quit MSC/PATRAN, select **File/Quit**.