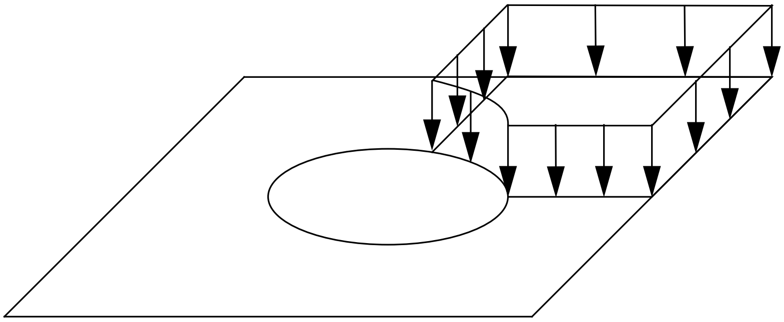
LESSON 14

Analysis Set-up of a Static Analysis



Objectives:

- Review all the steps necessary to build an analysis model.
- Understand how to setup a static analysis with MSC⁄PATRAN.

Model Description:

In this exercise you will build a complete MSC/PATRAN *Main Form* model and set up a static analysis run for MSC/NASTRAN.

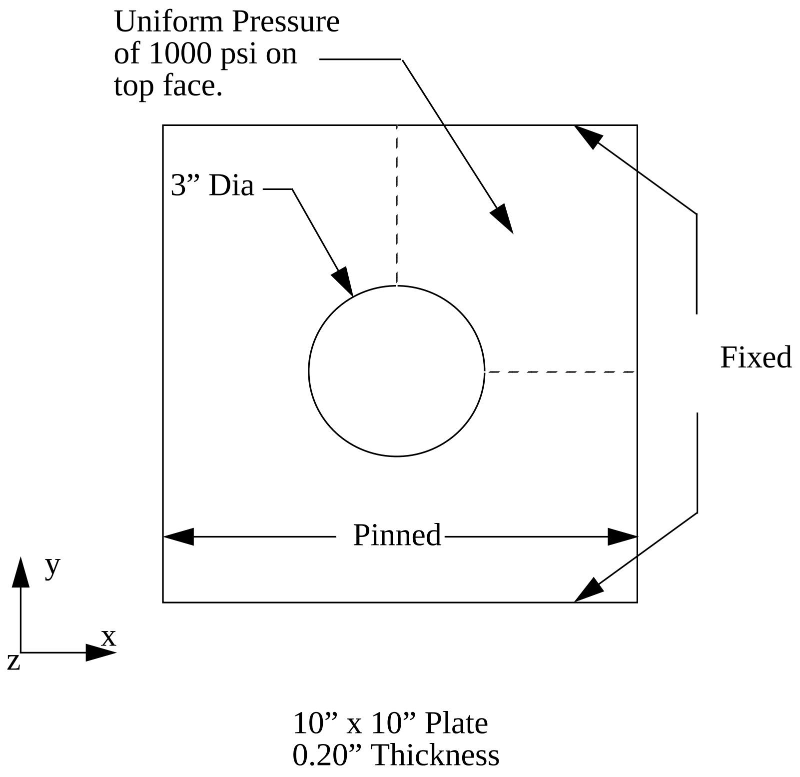


Figure 14-1

Quarter Symmetry Model with mesh seeds.

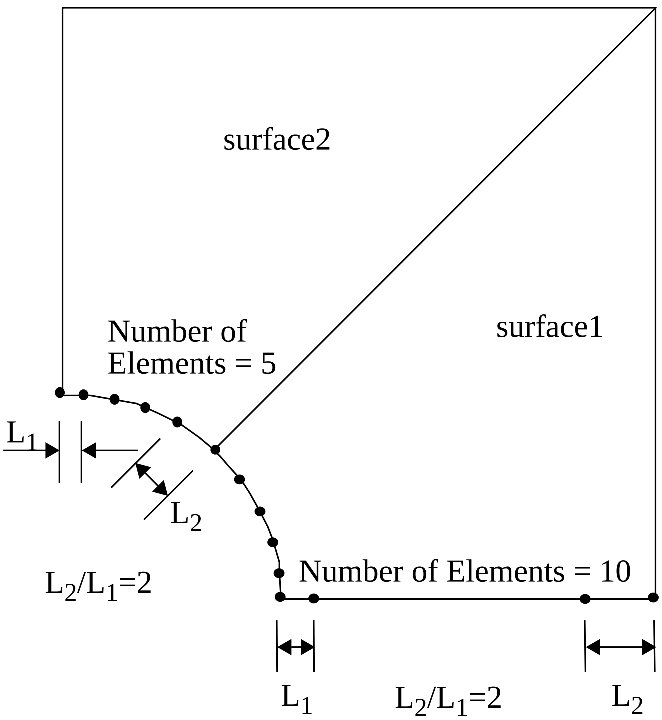


Figure 14-2

Table 14-1

Element type:	Quad8
Element global edge length:	1.0”
Material Constant Description	
Name:	Steel
Modulus of Elasticity, E (psi)	29E6
Poisson’s ratio, ν	0.30
Linear Elastic Isotropic material	
Element Properties:	
Name:	Prop1
Material:	Steel
Thickness:	0.2”
Analysis Code:	MSC/NASTRAN
Analysis Type:	Full Run, Linear Static Analysis
Analysis Solution Parameters:	Linear Static
Analysis Translator:	Text Output 2 format
Analysis Output Requests:	Displacements, Element Stresses, Element Strain Energies

Suggested Exercise Steps:

- Create a new database named **`plate_hole.db`**.
- Change the *Tolerance* to **Default** and the *Analysis Code* to MSC/NASTRAN.
- Create the quarter symmetry geometry and finite element mesh using the information in Figure 14-2 and Table 14-1.
- Equivalence and optimize the entire model. Verify that all element normals are in the same direction.
- Define the material and element properties using the information in Table 14-1.
- Assign a uniform pressure named **Pressure1** to the top surface of all elements.
- Assign the displacement boundary conditions to the appropriate edges of the model. Use the names, **disp_lf, disp_rt, disp_tp** and **disp_bt** for the left, right, top, and bottom displacement boundary condition set names.
- Prepare the model for a full analysis run using the information listed in Table 14-1.

Exercise Procedure:

1. Create a new database and name it **plate_hole.db**.

File/New Database...

New Database Name **plate_hole**

OK

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC/NASTRAN**.

New Model Preference

Tolerance ◆ **Default**

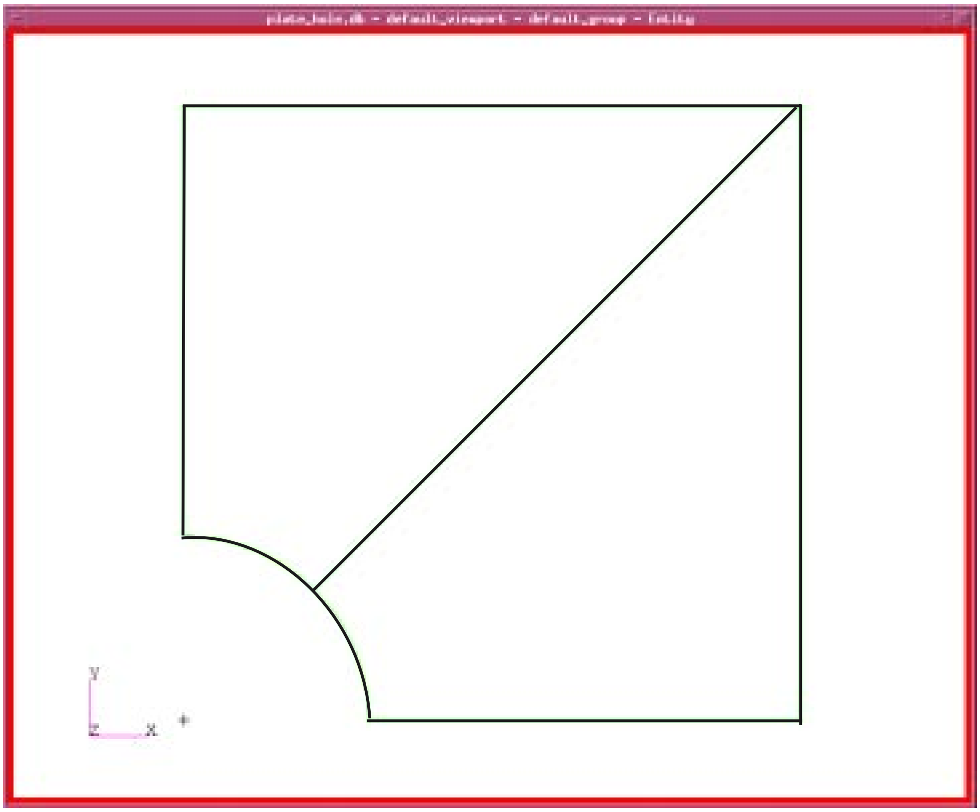
Analysis Code: **MSC/NASTRAN**

OK

Create the Geometry

3. Create the quarter symmetry geometry and finite element mesh using the information in Figure 14-2 and Table 14-1.

The surface representing the geometry of the plate is shown below:



4. Create the mesh seeds and mesh the model

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**One Way Bias**
Number =	**10**
L2/L1 =	**2**
Curve List	Select the bottom edge

Apply

For the bottom of the arc change:

Number =	**5**

L2/L1 =	**2**
Curve List	Select the bottom half of the arc

Apply

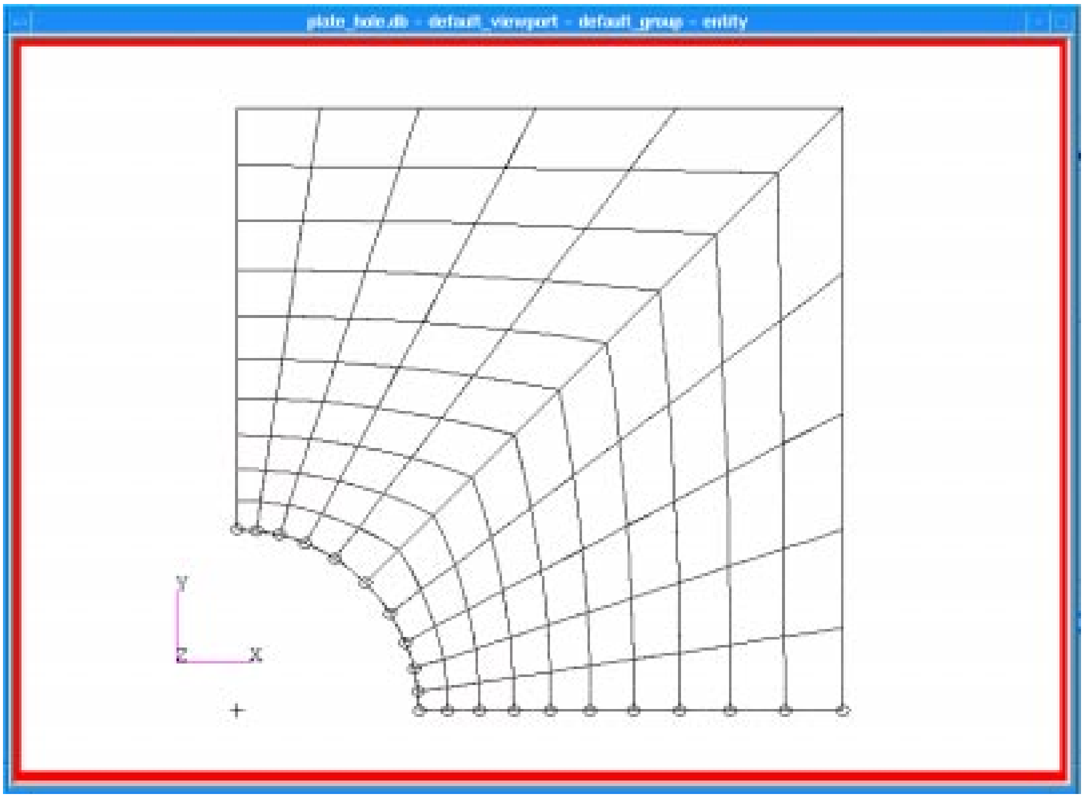
Change *L2/L1* to **-2** and click on the top half of the arc.If necessary, click **Apply**.

Now mesh the surface

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**1.0**
Element Topology	**Quad 8**
Mesher	**IsoMesh**
Surface List	**Surface 1, 2**

Apply

Your model's finite element mesh should look like the one shown in the figure below.



Equivalence

5. Equivalence the entire model. Verify that all element normals are in the same direction.

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

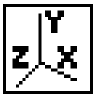
Verify

Verify the element normals

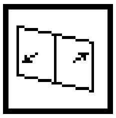
Action:	**Verify**
Object:	**Element**
Test:	**Normals**
Display Control	◆ **Draw Normal Vectors**

Apply

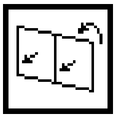
You may need to change the view to **isometric_view** by clicking on this icon in the toolbar.



All elements normal must point in the same direction. In this exercise we choose them to point in the positive Z-direction. If the normals are not pointing in the same direction there are two methods to reverse element normals. The first is under V**erify/Element/Normals**. Under *Test Control* click on **Display Only**



This will change to **Reverse Elements**



Guiding Element	

Select a guiding element that has a normal pointing in the direction you desire then click on **Apply**. All of the normals will then point in that same direction.

The second method is found in **Modify/Element/Reverse**. Here Patran will simply reverse the normals of any elements selected.

Create the Material Properties

6. Define the material and element properties using the information in Table 14-1.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**steel**
Input Properties...	
Constitutive Model	**Linear Elastic**
Elastic Modulus	**29E6**

Poisson's Ratio	**0.3**

Apply

Create the Element Properties

Create the element property definition for the model.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**prop1**
Options	**Homogeneous**
	Standard Formulation

Input Properties...

Material Name	**m:steel**
Thickness	**0.20**

OK

Select Members	**Surface 1, 2**

Add

Apply

Apply Loads and Boundary Conditions

7. Assign a uniform pressure named **Pressure1** to the top surface of all elements.

◆ **Load/BCs**

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
Analysis Type	**Structural**
New Set Name	**pressure1**
Target Element Type	**2D**

Input Data...

Top Surface Pressure	**1000**

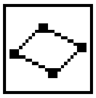
OK

Select Application Region...

Geometry Filter ◆ **FEM**

Select 2D Elements or Edges Select Entire Model

Click on the **Tri or Quad Element** icon in the select menu then screen select the entire model.

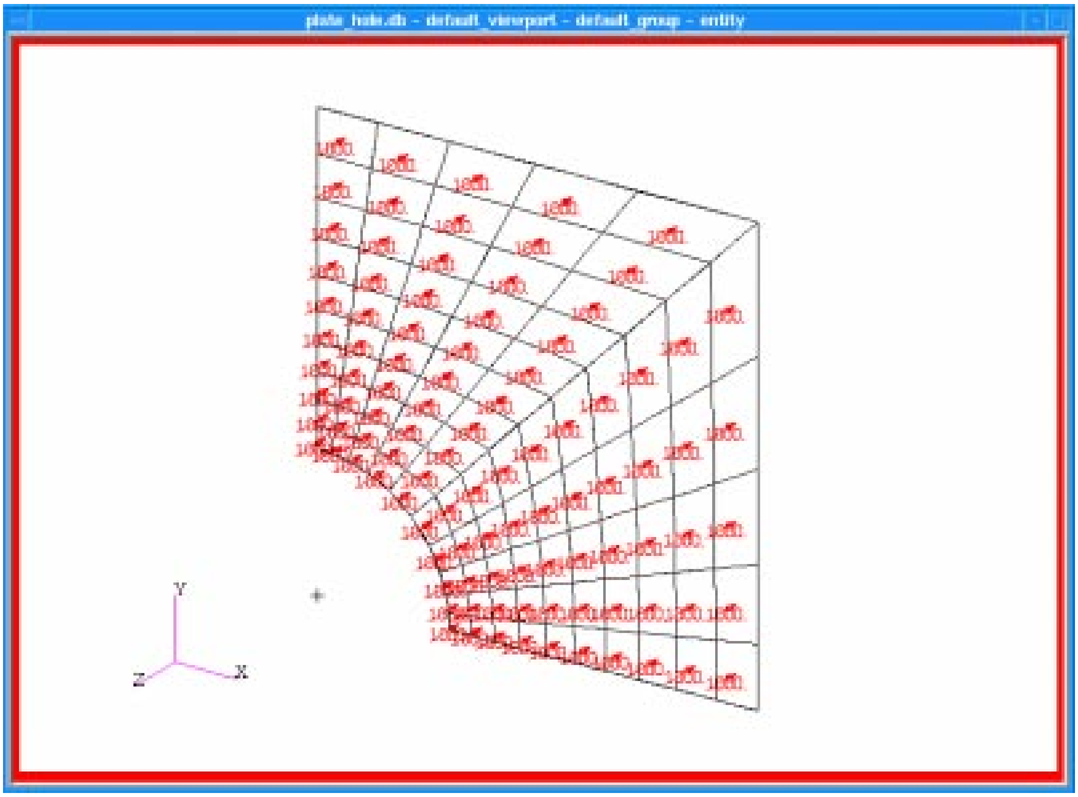


Add

OK

Apply

The uniform pressure load is shown below. Of course, the orientation of the pressure load will depend on original orientation of the element normals.



8. Assign the displacement boundary conditions to the appropriate edges of the model. Use the names, **disp_lf, disp_rt, disp_tp** and **disp_bt** for the left, right, top, and bottom displacement boundary condition set names.

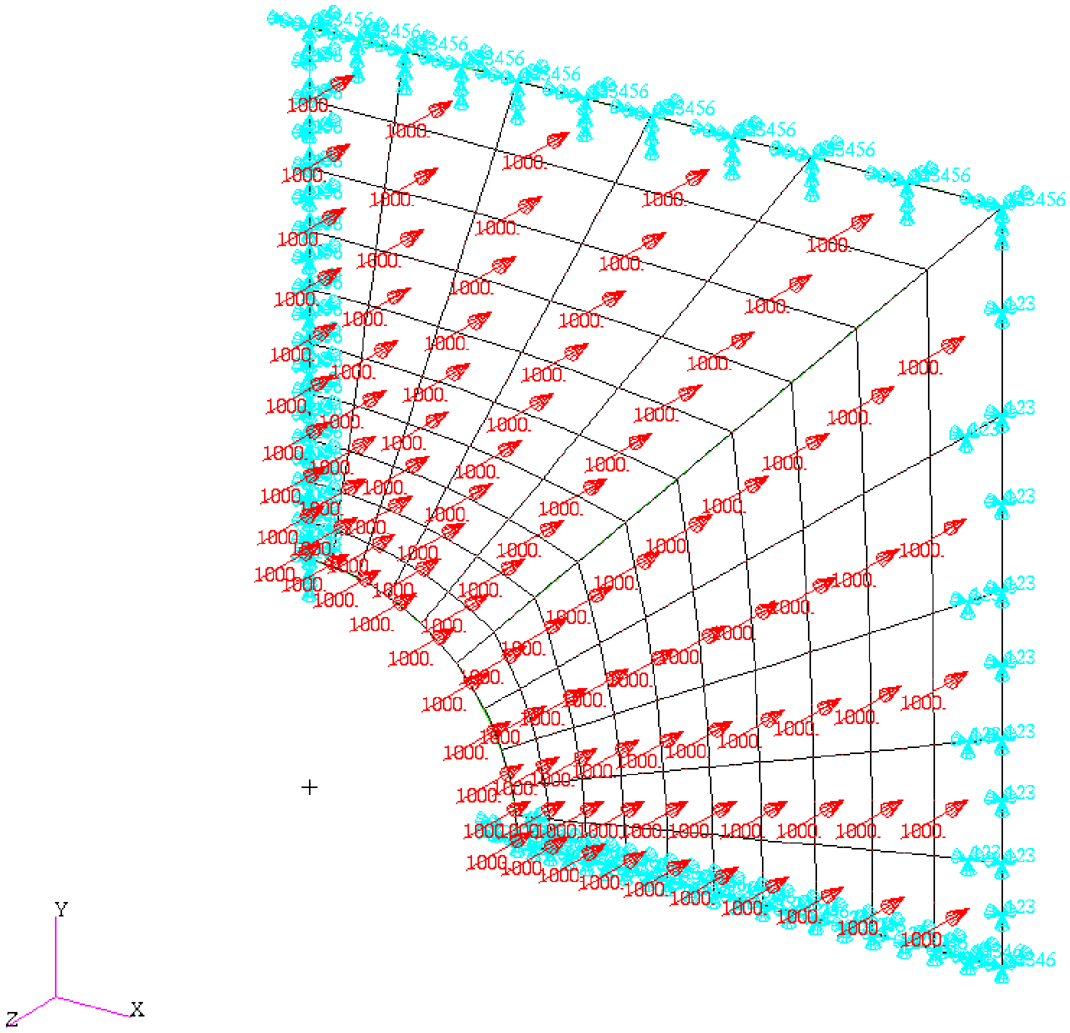
Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**disp_lf**
Input Data...	
Translations	**<0, , >**
Rotations	**<, 0, 0>**
OK	
Select Application Region...	
Geometry Filter	◆ **FEM**
Select Nodes	Select the left edge
Add	
OK	
Apply	

Using the Table below, define the remaining displacement boundary conditions.

Table 14-2:

Name	Translations	Rotations	Application Region
disp_rt	**<0,0,0>**	**< >**	**Nodes on right edge.**
disp_tp	**<0,0,0>**	**<0,0,0>**	**Nodes on top edge.**
disp_bt	**<,0,>**	**<0, ,0>**	**Nodes on bottom edge.**

When you are finished your model's displacement boundary conditions should look like those shown in the figure below.



9. Prepare the model for a full analysis run using the information listed in Table 14-1.

Set-up the Analysis

◆ **Analysis**

Action:	**Analyze**
Object;	**Entire Model**
Method:	**Full Run**

Translation Parameters...

OUTPUT2 Format:	**Text**
OK	
Solution Type...	
Solution Type	◆ **Linear Static**
Solution Parameters...	

Review the form, but do not change its default settings.

OK

OK

In MSC/NASTRAN, the subcases provide a tool to associate loads and boundary conditions, output requests and various other parameters depending on the solution type selected. These subcases are essential to perform portions of a full run like performing nonlinear analysis and analyzing a model with super elements.

Click on the **Subcase Create...** button, you will notice a subcase already created. The name of the subcase is the same as the loadcase which is **Default**. This subcase consists of the Default load case, and the requested outputs that can be inspected by pressing the *Output Requests* button.

When done inspecting the form, you may press the **Cancel** buttons.

File/Quit