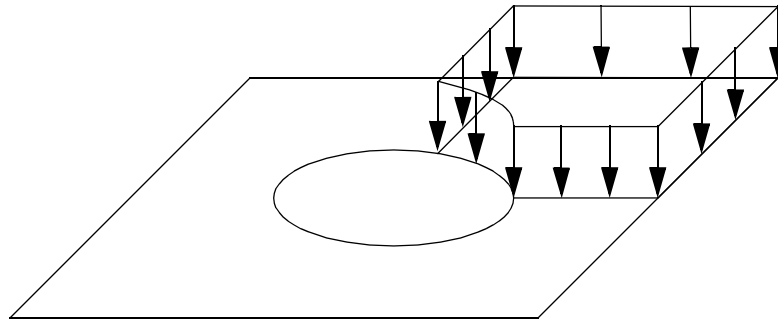

WORKSHOP 13

Analysis Set-up of a Static Analysis



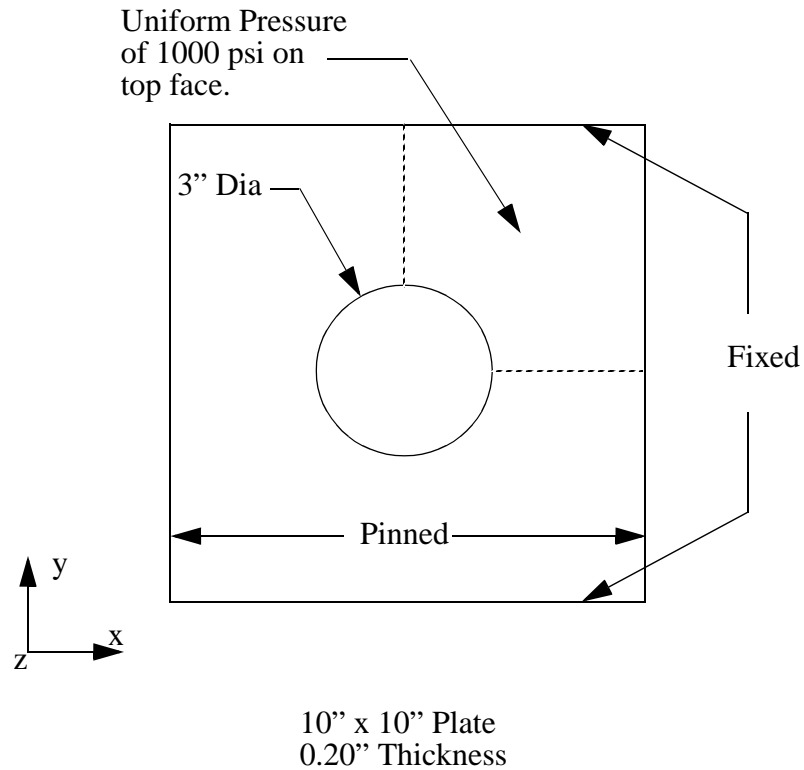
Objectives:

- 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 13-1**

Quarter Symmetry Model with mesh seeds.

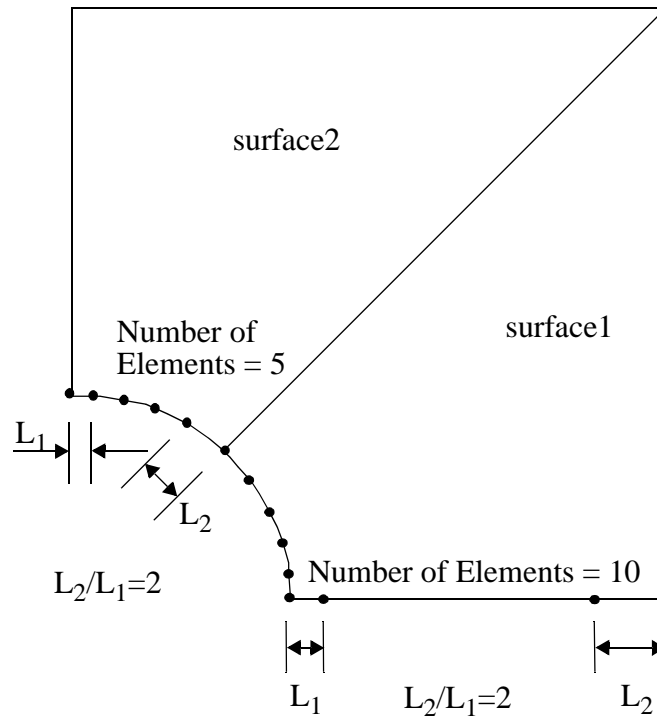


Figure 13-2

Table 13-1

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 13-2 and Table 13-1.
- 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 13-1.
- 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 13-1.

Exercise Procedure:

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

File/New...*New Database Name***plate_hole****OK**

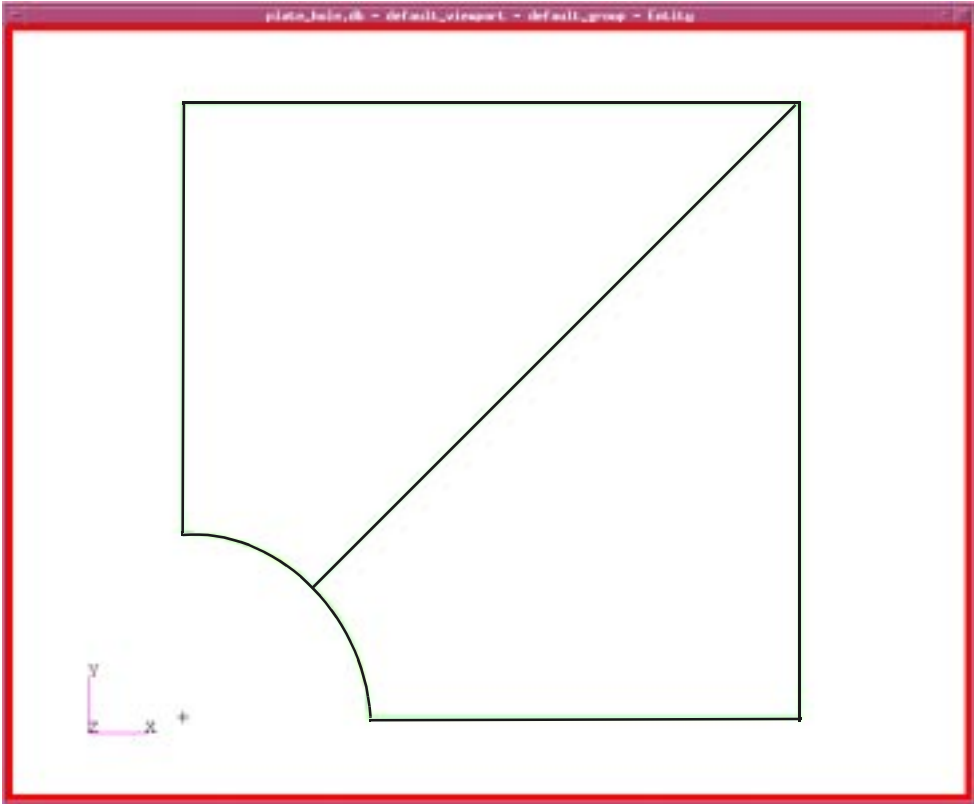
2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC/NASTRAN**.

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 13-2 and Table 13-1.

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



4. Create the mesh seeds and mesh the model

◆ Finite Elements

Action:	<input type="button" value="Create"/>
Object:	<input type="button" value="Mesh Seed"/>
Type:	<input type="button" value="One Way Bias"/>
Number =	<input type="text" value="10"/>
L2/L1 =	<input type="text" value="2"/>
Curve List	Select the bottom edge
<input type="button" value="Apply"/>	

For the bottom of the arc change:

Number =	<input type="text" value="5"/>
----------	--------------------------------

L2/L1 =

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

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Surface"/>
<i>Global Edge Length</i>	<input type="text" value="1.0"/>
<i>Element Topology</i>	<input type="text" value="Quad 8"/>
<i>Mesher</i>	<input type="text" value="IsoMesh"/>
<i>Surface List</i>	<input type="text" value="Surface 1, 2"/>

Apply

For a better view, erase all geometry.

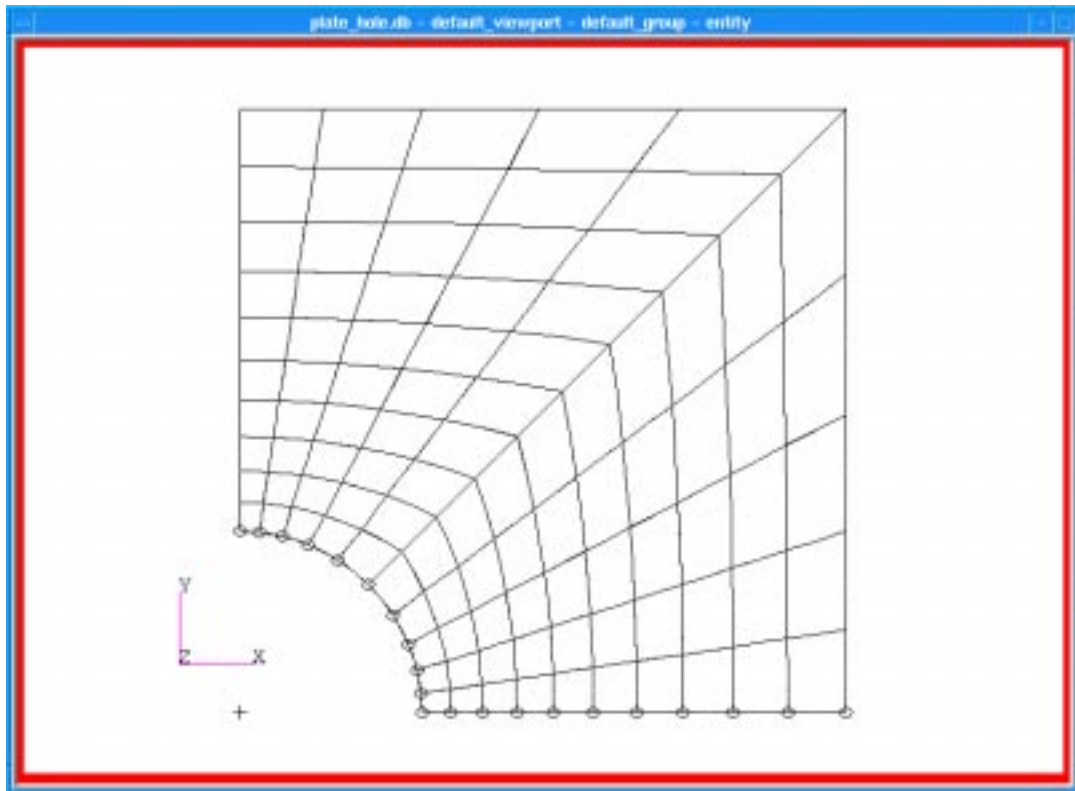
Display/Plot/Erase...

Geometry

Refresh graphics using the Refresh icon.



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



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

Equivalence

Action:

Equivalence

Object:

All

Method:

Tolerance Cube

Apply

Verify the element normals

Verify

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 **Verify/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.

- Define the material and element properties using the information in Table 13-1.

Create the Material Properties

◆ **Materials**

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name

steel

Input Properties...

Constitutive Model

Linear Elastic

Elastic Modulus

29E6

**Create the
Element
Properties**

Poisson's Ratio

0.3

Apply

Create the element property definition for the model.

Plot all posted geometry.

Display/Plot/Erase...

Geometry

Plot

Refresh graphics using the Refresh icon.



◆ 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

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

◆ **Load/BCs**

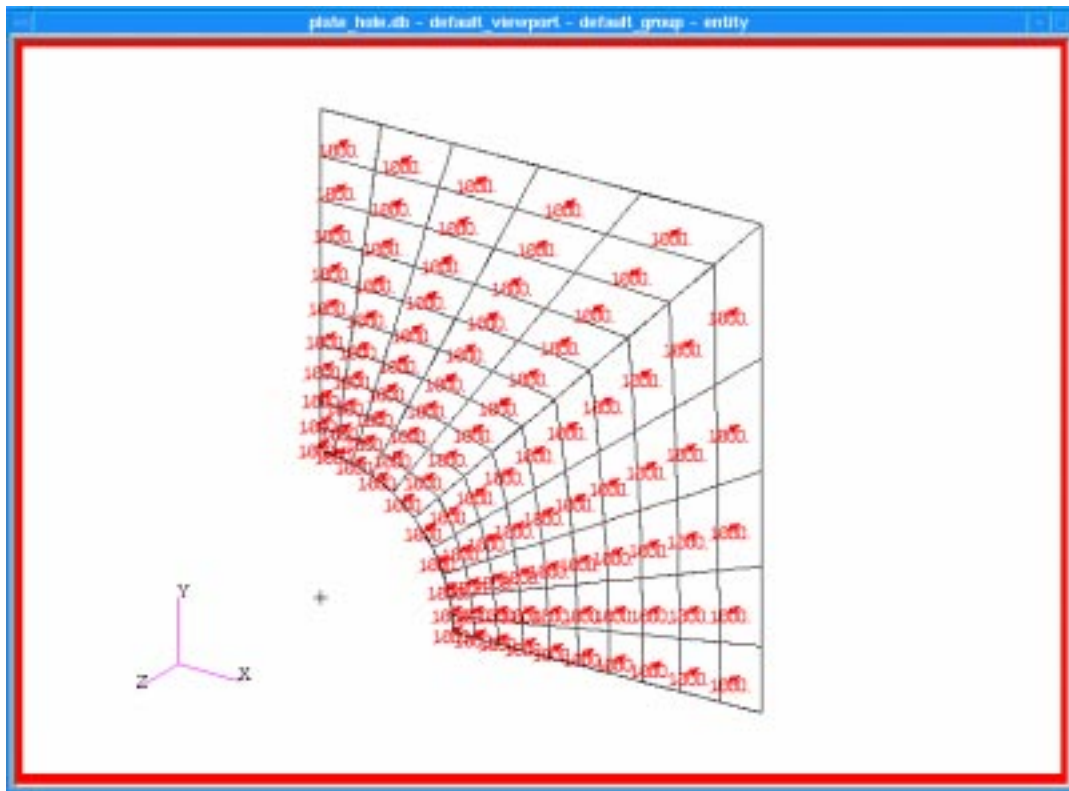
<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Pressure"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>Analysis Type</i>	<input type="text" value="Structural"/>
<i>New Set Name</i>	<input type="text" value="pressure1"/>
<i>Target Element Type</i>	<input type="text" value="2D"/>
<input type="button" value="Input Data..."/>	
<i>Top Surf Pressure</i>	<input type="text" value="1000"/>

<i>Geometry Filter</i>	◆ FEM
<i>Select 2D Elements or Edges</i>	Select Entire Model

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



The uniform pressure load is shown below. Of course, the orientation of the pressure load will depend on original orientation of the element normals. Change to front view before proceeding to next step.



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

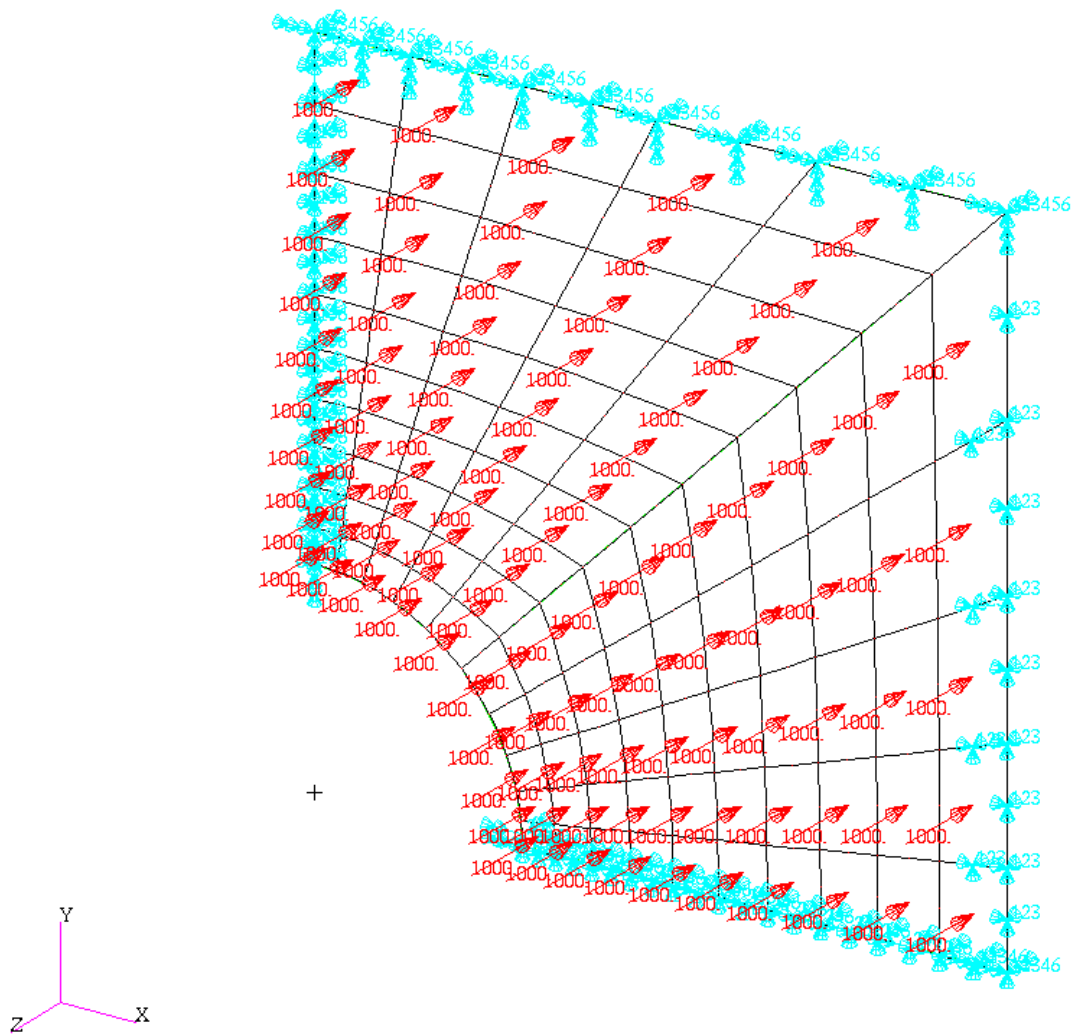
Apply

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.



Set-up the Analysis

- Prepare the model for a full analysis run using the information listed in Table 13-1.

◆ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Full Run

Translation Parameters...

OUTPUT2 Format:

Text

Solution Type◆ **Linear Static**

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

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...

