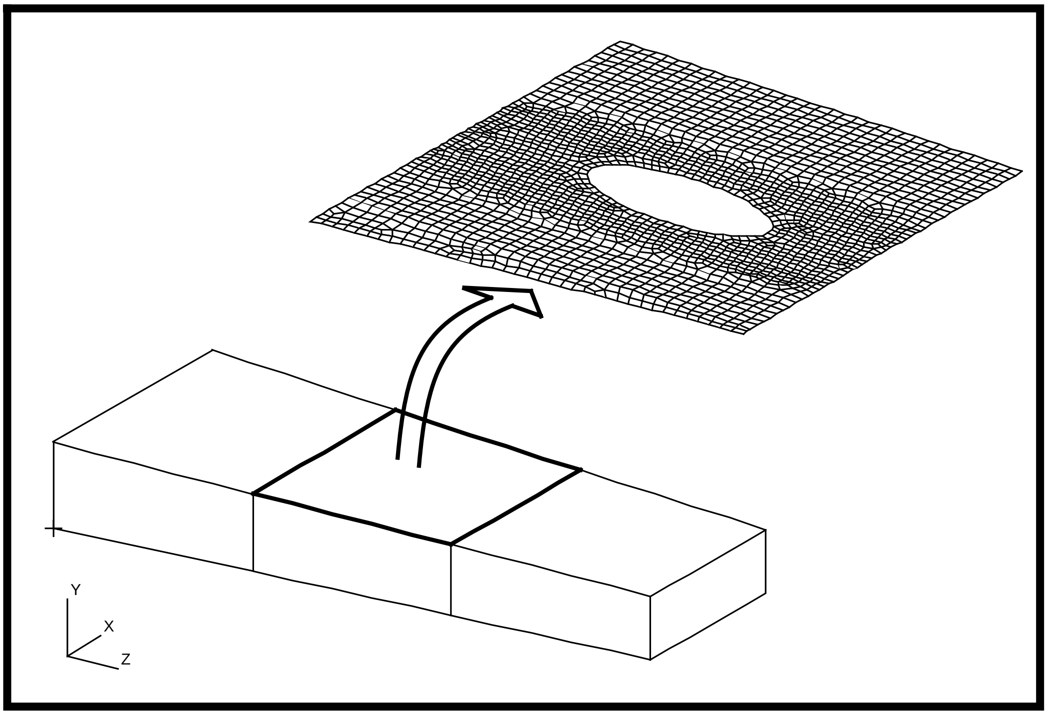
LESSON 14

Global/Local Model using FEM Fields



Objectives:

- Import a global loads model and corresponding results.
- Import local detail panel stress model.
- Create a displacement FEM Field based on the results of the Global Model for application as an enforced displacement on the local model.
- Create an MSC/NASTRAN input deck.

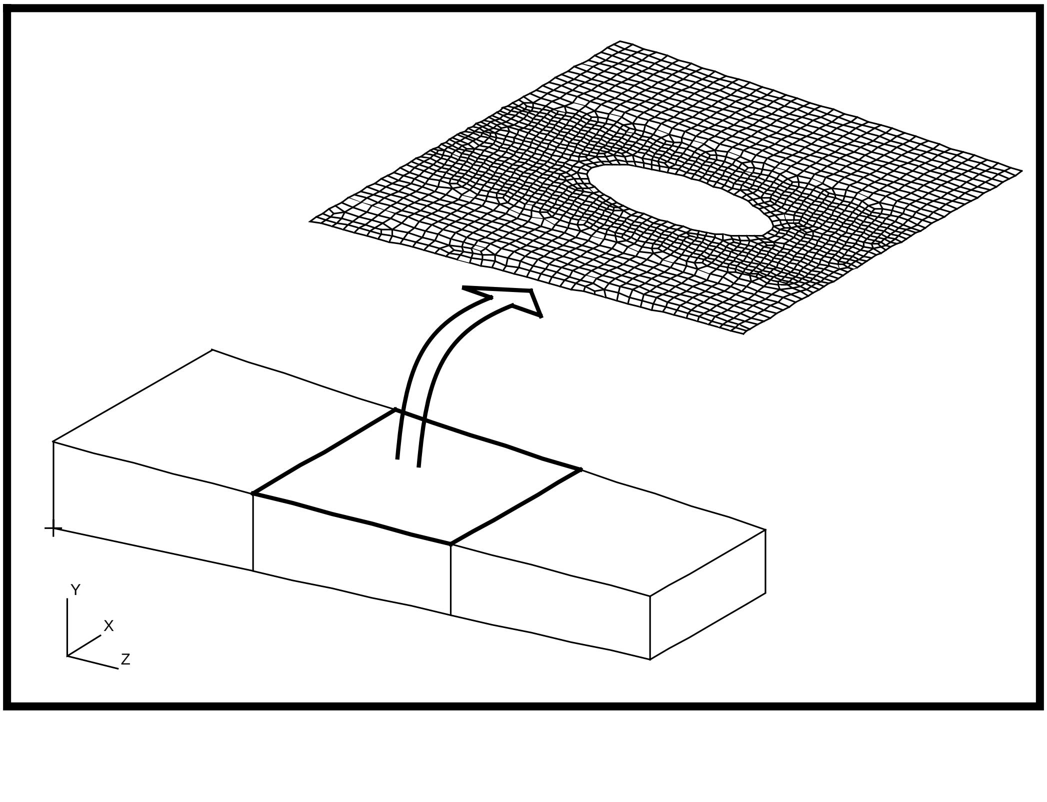
Model Description:

In this exercise you will read in a Global Loads model and the corresponding results of a multi-cell box beam from a MSC/ NASTRAN output file into a new PATRAN database. Then you will read in the local model, a detailed panel stress model that represents an outer skin of one of the cells in the global loads model.

The detailed model contains thickness buildups, detailed stiffer modeling and an access hole. The resulting deflections of the Global Loads Model will be applied to the perimeter of the Local Stress Model as enforced displacements. This will simulate the loads transferred across a free-body section of the global model where the local model resides. Additionally, panel pressures will be applied to the Local Stress Model to account for all the loading.

Finally, an input deck of the Detailed Stress Model which contains these new loads will be written out into a file.

The view of the global model is as shown here.



Suggested Exercise Steps:

- Open a new database called **global_local.db**.
- Create an empty group for the whole model.
- Read in both the model and the results of the Global Loads Model.
- Create an empty group for the panel part of the model.
- Read in the MSC/PATRAN input deck that represents the detailed panel stress model.
- Set an offset for the nodes and elements of the panel model from the global model so that they do not interfere.
- Make a fringe plot of the deflection of the global model.
- Make a vector plot and create a spatial field of the global model.
- Create a displacement constraints on the panel model.
- Create an uniform pressure load on the panel model.
- Use Analysis Deck to create a MSC/PATRAN input deck for the panel model.

Files:

All the files used in this exercise are listed below. Each listing includes the file, where it originated, its format (text/binary) and summary information as to how it relates to this exercise.

File	Supplied/Created	Format	Description
p3quickpick.def	supplied	text	This is an ASCII file that sets up the p3 supplied quickpick file. This file displays 2 columns X 16 rows of default p3 icons. A similar file is located in P3HOME.
multi_cell_box.op2	supplied	text	MSC/NASTRAN results output file that is in ascii format. This file is contains the model and results data for the global loads model used in this exercise and originated from a NASTRAN solution 101 run.
multi_cell_box.op2.bin	supplied	binary	This is the file that gets created when the multi_cell_box.op2 file is read into the PATRAN database. This is a binary representation of the op2 file.
panel.bdf	supplied	text	This is an ASCII MSC/NASTRAN input datadeck that is read into the PATRAN database. This file represents the detailed model of a cover panel for the multi_cell_body model and contains such details as panel access hole, stiffeners and caps.
global_local.db	created	binary	This is a PATRAN database (binary) created new for this exercise. The panel.bdf and multi_cell_box.op2, are read into this database. The displacements of the global loads model are applied to the detailed stress model. The detailed stress model (new_panel.bdf) is then regenerated.

global_local.msg.01	created	text	This is an ASCII message file that gets generated when the multi_cell_box.op2 file gets read in. This file provides a summary of all the information that was read in. Also, any errors or warnings messages get included into this file.
naspat.msg.01	created	text	The current process for importing an input file is to first convert it to a PATRAN 2.5 neutral file then import it into the database. This is the message file for the first step for the conversion of the panel.bdf into a neutral file. This ASCII file contains information regarding MSC/NASTRAN cards conversion and information on unrecognized cards.
patran.pat.01	created	text	This is the neutral file representation of the panel.bdf file. This file gets created during the process of reading in the panel.bdf file.
new_panel.bdf	created	text	This is a newly created MSC/NASTRAN input deck. This file contains the panel.bdf and the perimeter global loads displacements and panel pressures.
new_panel.msg.02	created	text	This is an ASCII message file that gets generated when the new_panel.bdf file gets written out. This file provides a summary of all the information that was written out. Also, any errors or warnings messages get included into this file.
patran.prt	created	text	This is the PATRAN text report file created new from within this database. This file contains user specified results data and is in ASCII format.

Exercise Procedure:

1. Open up a new database and call it **global_local.db**.

Open a Database

File/New

New Database Name: **global_local.db**

OK

In the New Model Preference form set the following:

Tolerance: ◆ **Based on Model**

Approximate Maximum Model Dimension: **10**

Analysis Code: **MSC/NASTRAN**

Analysis Type: **Structural**

OK

2. Create an empty current group for the global model as follows:

Create a Group

Group/Create...

New Group Name **global_loads_model**

■ **Make Current**

■ **Unpost All Other Groups**

Apply

Cancel

3. Use *Analysis* to read in BOTH the model and results of the Global Loads Model.

Read In Mode and Results

◆ **Analysis**

Action: **Read Output2**

Object: **Both**

Method: **Translate**

Select Results File...

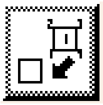
Selected Results File **multi_cell_box.op2**

OK

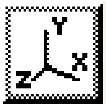
Apply

Turn off Labels and Change a View

4. Turn off all labels on the model and reset the view to isometric view using these Toolbar icons.

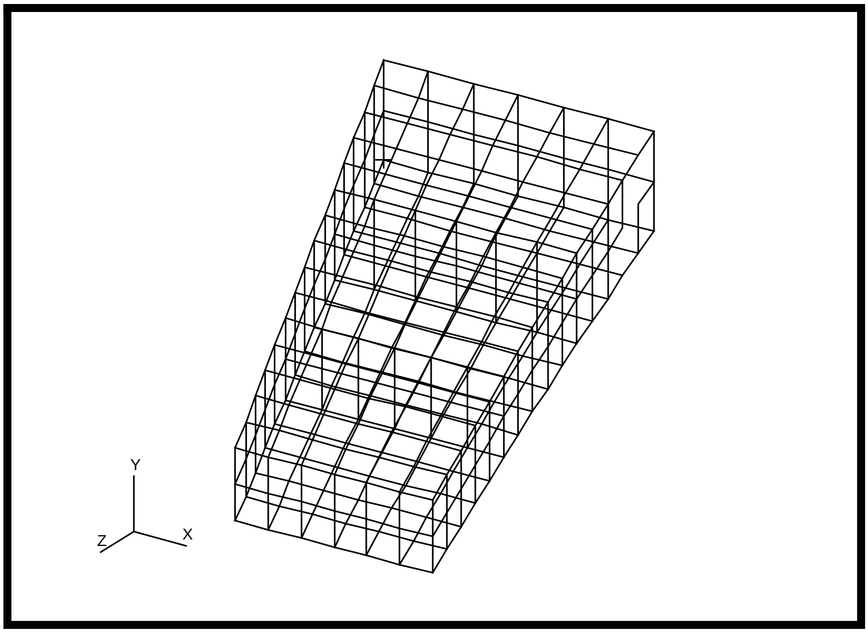


Hide All Entity Labels



Iso 1 View

The view of the model should be like the one shown below.



Create a Group

5. Create an empty current group by the name of Panel_Model.

Group/Create...

New Group Name **panel_model**

■ **Make Current**

■ **Unpost All Other Groups**

Apply

Cancel

6. Use Analysis to read in a MSC/NASTRAN input deck that represents the detailed panel stress model.

Read In an Input File

◆ **Analysis**

Action: **Read Input File**

Object: **Model Data**

Method: **Translate**

Entity Selection...

Define Offsets...

Offset the nodes and elements by 1000 so they do not interfere with the nodes and elements of the Global Model. Click the cell under *Offset* in the *Nodes* row.

Setting an Offset for Nodes and Elements

Input Offset Value **1000**

<Return>

Click the cell under *Offset* in the *Elements* row.

Input Offset Value **1000**

<Return>

Your *Entity Label Offset Definition form* should appear as follows:

Existing ID Range in Db		New ID
Mininum	Maximum	Offset
1	922	1000
1	412	1000

OK

OK

Select **`panel.bdf`** as the input file.

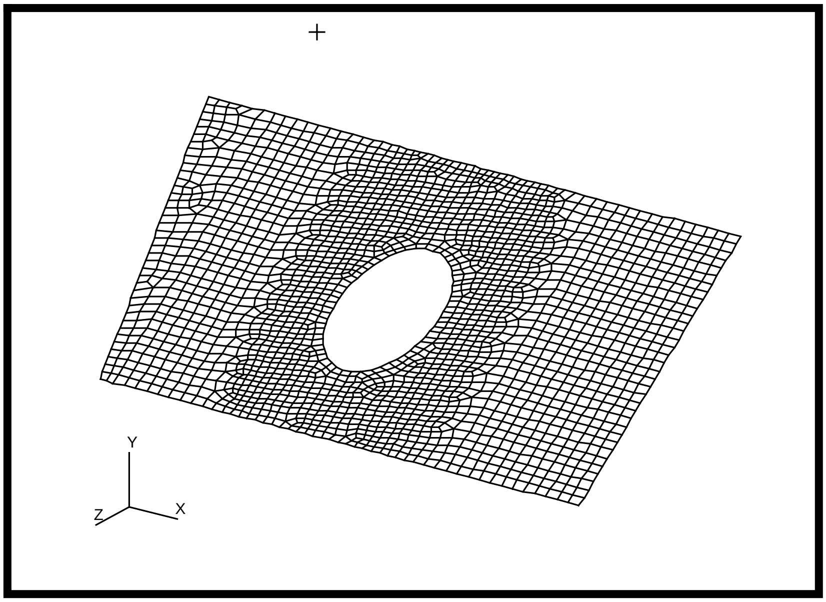
Select Input File...

Selected Input File **`panel.bdf`**

OK

Apply

Here is the display of the panel model.



Post a Group

7. Now post only the Global_Loads_Model group.

Group/Post...

Select Groups to Post **Global_Loads_Model**

Apply

Cancel

Change the view of the model as follows:

Viewing / Angles...

Angles **25 60 0**

Apply

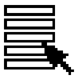
Cancel

Display a Fringe Plot

8. Take a look at the deflections and stresses of the global loads model using basic Results form.

◆ **Results**

Click on the **Select Results** icon



Action: **Create**

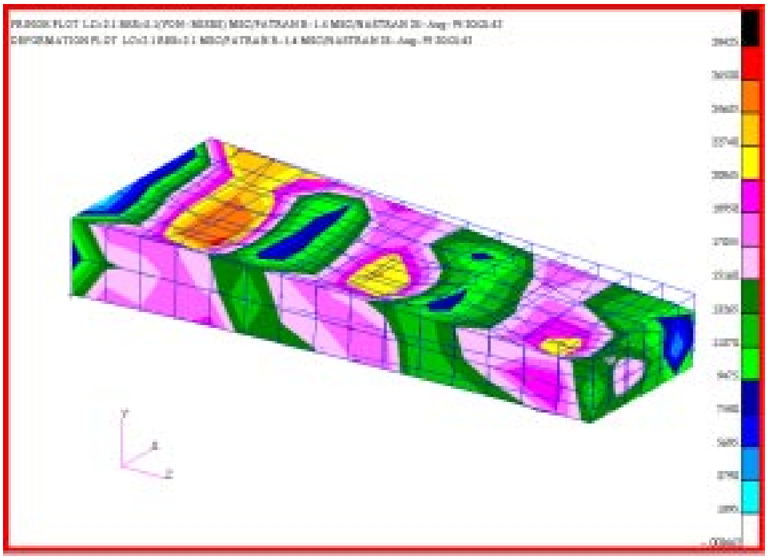
Object: **Quick Plot**

Select Fringe Result **Stress Tensor,**

Select Deformation Result **Displacements, Translational**

Apply

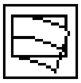
The fringe plot of the results is shown below.



Setting a Results Scale Factor

9. Now set the displacements to show true values (scale factor = 1.0, direct multiplication)

   Select the **Deformation Attributes** icon

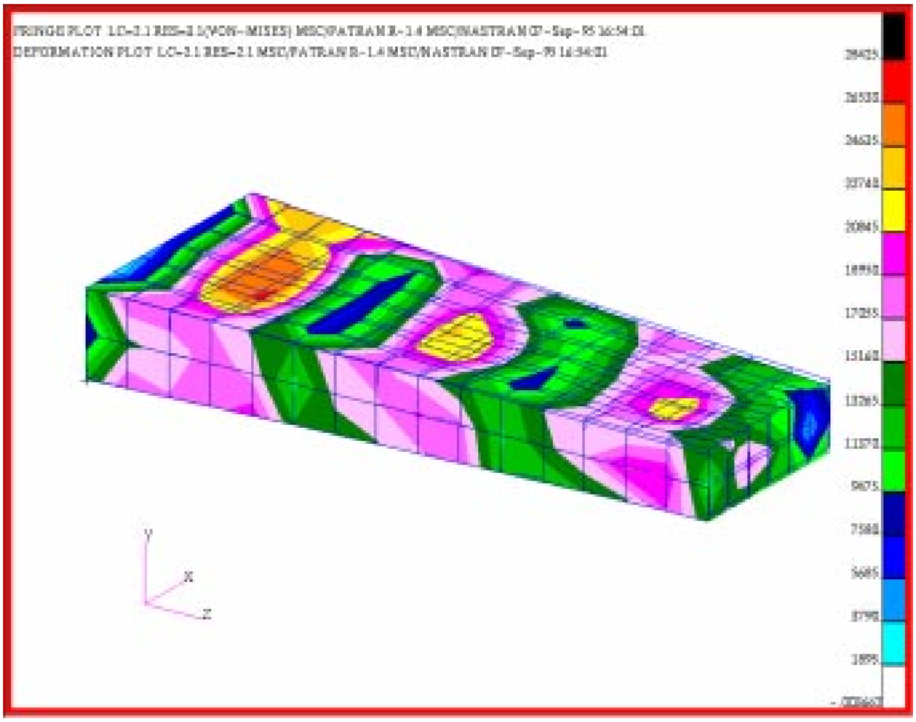


◆ **True Scale**

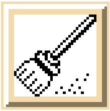
Scale Factor **1.0**

Apply

The fringe plot is now as shown here.



Use the Broom Icon to reset the graphics.

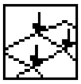


10. Using advanced Results form, create a Vector Plot of Displacements.

Display a Vector Plot

Action:	**Create**
Object:	**Marker**
Method:	**Vector**
Select Result	**Displacements, Translational**

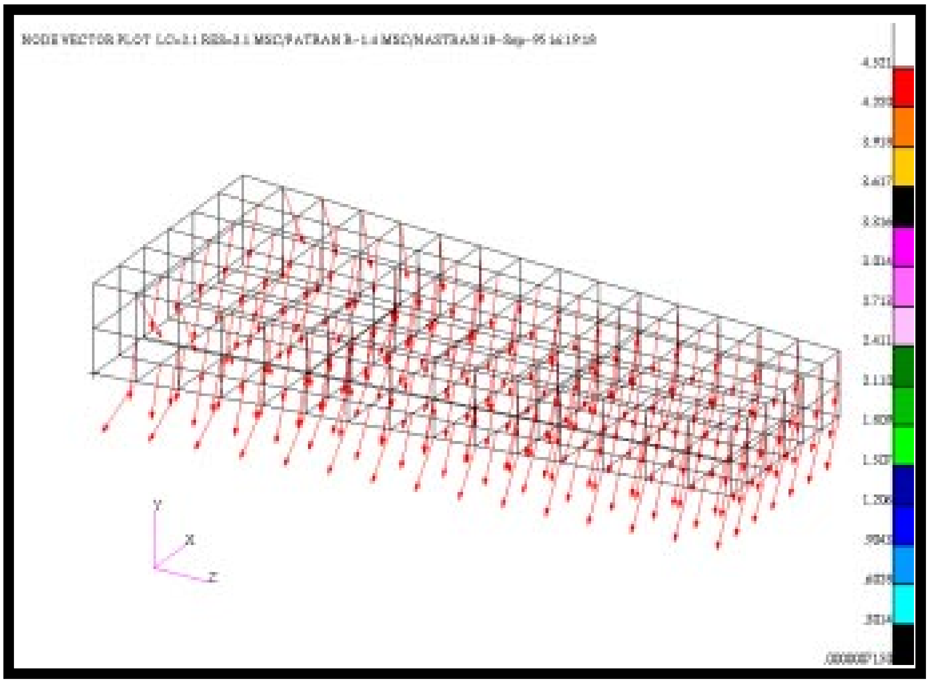
Select the **Display Attributes** icon



❐ **Show Vector Label**

Apply

The resulting vector plot is as follows:



Create a Spatial Field

11. With the vector plot displayed, create a continuous, spatial, vector FEM Field by the name of Displacements_Global_Model.

◆ **Fields**

Action:	**Create**
Object:	**Spatial**
Method:	**FEM**
Field Name	**Displacements_Global_Model**
FEM Field Definition	◆ **Continuous**
Field Type	◆ **Vector**
Mesh/Results Group Filter	◆ **All Groups**
Select Group	**Global_Loads_Model**

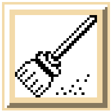
[Options...]

Extrapolation Option	**Linear Extrapolation**

OK

Apply

Use the Broom Icon to reset the graphics.



12. Now post only the detailed Panel_Model group.

Post a Group

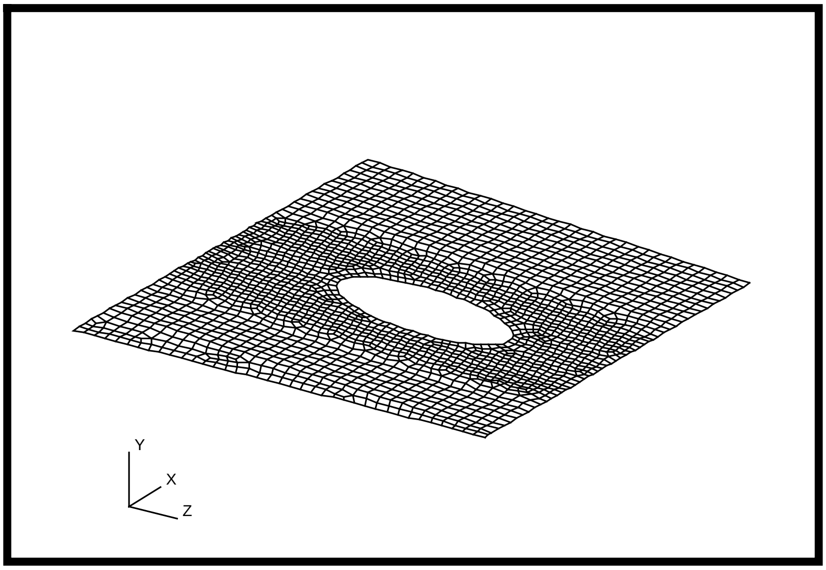
Group/Post...

Select Groups to Post **Panel_Model**

Apply

Cancel

The Panel model is shown here.



13. Create a group by the name of Panel_Model_Edge_Nodes.

Create a New Group

Group/Create...

New Group Name **Panel_Model_Edge _Nodes**

■ **Make Current**

■ **Unpost All Other Groups**

Entity Selection	**Node 10000T#**

Apply

Cancel

Setting Node Size to 5

Using the slidebar under the **Display/Finite Elements...** option, set the node size to 5 to make the nodes more visible.

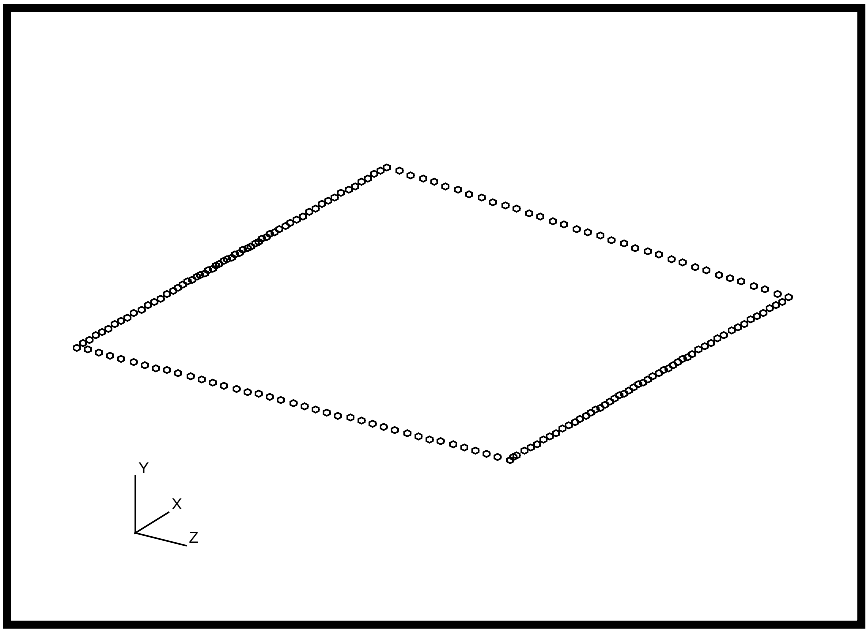
Display/Finite Elements...

Node Size	**5**

Apply

Cancel

The panel edge nodes are displayed here.



Create a Load Case

14. Use Load Cases to create a static load case by the name of Panel Model Loads. Make sure this load case is set current prior to creation.

◆ **Load Cases**

Action:	**Create**
Load Case Name	**Panel_Model_Loads**

■ **Make Current**

Load Case Type:	**Static**

Apply

15. Create a Load/BC by the name of Panel Enforced Displacements. Use the newly created Spatial FEM Field, Displacements_Global_Model, to define the translations.

Create a Displacement Constraints

◆ **Load/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name	**Panel Enforced Displacements**

Input Data...

Translation <T1 T2 T3>	**(Pick Displacements_Global_Model in the Spatial Fields databox)**

OK

Select Application Region...

Geometry Filter	◆ **FEM**
Select Nodes	**Node 10001:10180** **(Screen pick all the nodes in the viewport)**

Add

OK

Apply

Set the node size back to 1.

Setting Node Size to 1

Display/Finite Elements...

Node Size	**1**

Apply

Cancel

Turn off the vector values to have a cleaner display.

Turn Off the Vector Values

Display

Load/BC/Elem. Props ....

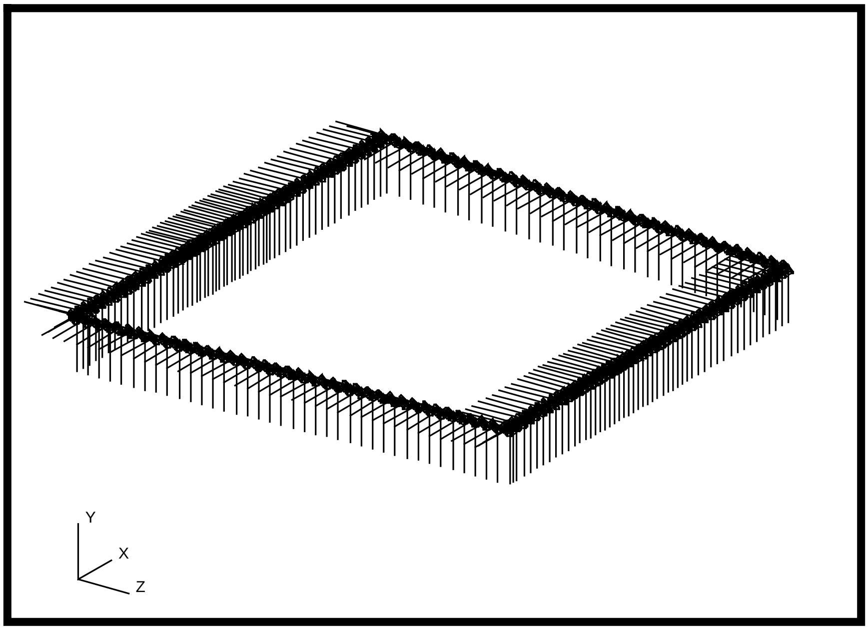
Vectors/Filter...

❑ **Show LBC/EI. Prop. Values**

Apply

Cancel

The displacement load/BC's is shown here:



16. Post and make current the Panel Model.

Post a Group

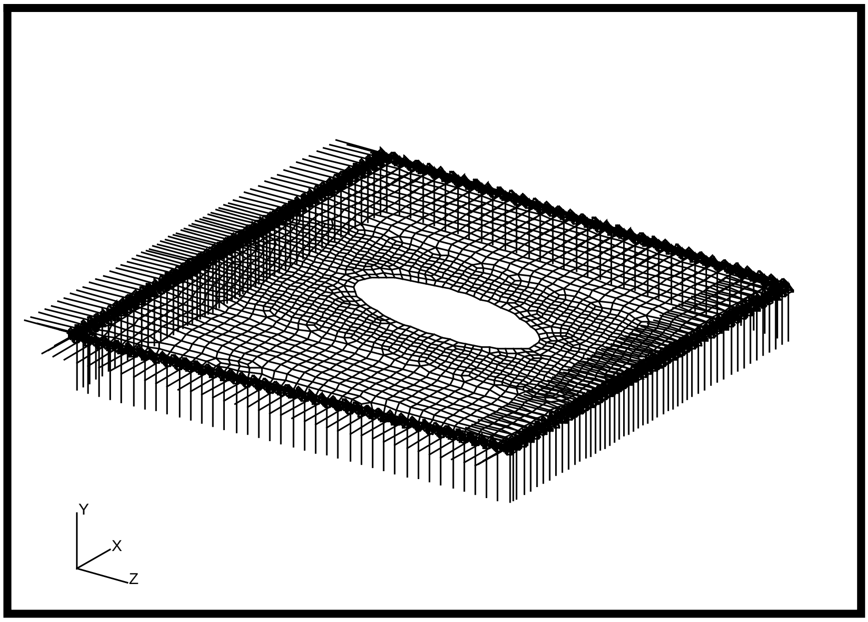
Group/Post...

Select Groups to Post **Panel_Model**

Apply

Cancel

The viewport now displays this:



17. Create a uniform pressure called Panel Pressure which has a value of 2.25 applied to all 2D quads on the top surface.

Create a Uniform Pressure BCs

◆ **Load/BCs**

Action:	**Create**
Object:	**Pressure**
Method:	**Element Uniform**
New Set Name	**Panel Pressure**
Target Element Type:	**2D**

Input Data...

Top Surf Pressure	**2.25**

OK

Select Application Region...

Geometry Filter ◆ **FEM**

Select 2D Elements or Edges	**Element 1001:3071 (Screen Pick all the elements in the viewport)**

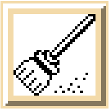
Add

OK

Apply

Red arrows representing the pressures will appear on the surface pointing downward.

Use the Broom Icon to reset the graphics.



Create an Analysis Deck

18. Finally, use Analysis Deck to create a MSC/NASTRAN input deck. Call the job name to new_panel.

◆ **Analysis**

Action:	**Analyze**
Object:	**Current Group**
Method:	**Analysis Deck**
Job Name	**new_panel**

Translation Parameters...

OUTPUT2 Format:	**Text**

OK

Solution Type...

Solution Type	◆ **Linear Static**

Solution Parameters...

Plate Rz Stiffness Factor=	**1.0**

OK

OK

Subcase Create...

Subcase Name	**Panel_Model**

Available Load Cases **Panel_Model_Loads**

Apply

Subcase Select...

Subcases Selected: **Panel_Model**

OK

Apply

19. <Optional> If you have MSC/NASTRAN on your Network, you can submit the new_panel.bdf for analysis and import the results file, new_panel.op2, into the database and look at model stresses and deformation.

When the analysis is finished, a message will come up in the *command window* to state that the translation has completed successfully.

Quit PATRAN

20. Close database and quit p3.

File/Quit