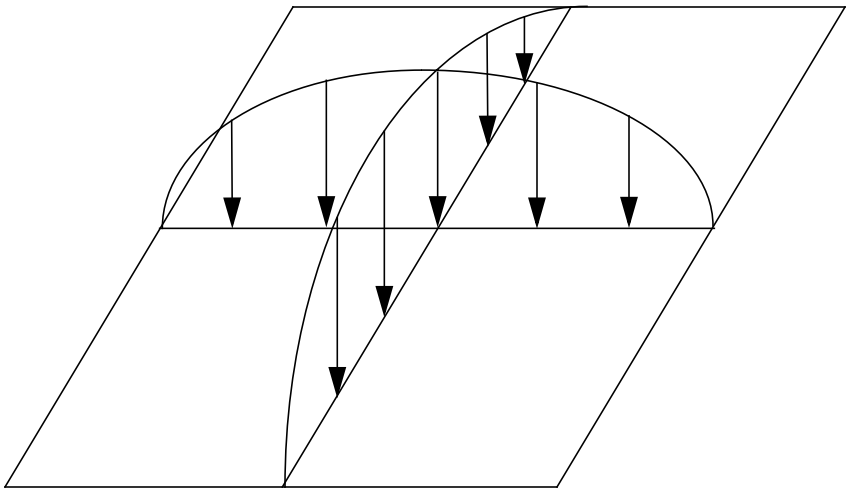
LESSON 10

Spatial And Temporal Variation of Loads

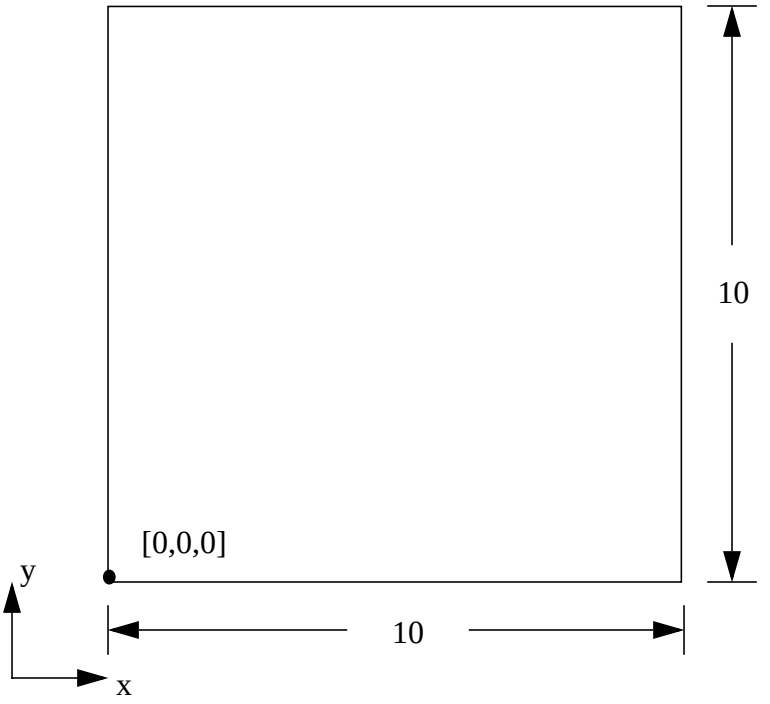


Objective:

- To model spatially and temporally varying applied loads.

Model Description:

In this exercise you will create a simple flat plate model and then apply a pressure load that is a function of both time and spatial location.



Analysis Code:	MSC/NASTRAN
Element type:	Quad4
Element Global Edge Length:	1.0
Pressure Loading:	

$$P(x,y,z,t) = 100\,\mathrm{sinr}(\pi x/10)\,\mathrm{sinr}(\pi y/10)\,\mathrm{cosr}(10t)$$

where, $0 \le x \le 10$; $0 \le y \le 10$; $0 \le t \le 2$;
use 30 time increments; π=3.14159

Figure 11-1

Suggested Exercise Steps:

- Create a new database named **`variable_loads.db`**.
- Change the Tolerance to Default and the Analysis Code to MSC/NASTRAN.
- Create the geometry and finite element mesh using the information in Figure 11-1.
- Create a time dependent load case named **my_load_case_1**.
- Define a Spatial field named, **pressure_spatial: 100*sinr(3.14159*'X/10)*sinr(3.14159*'Y/10)**.
- Define a Time-dependent field named, **pressure_temporal: cosr(10*'t)**.
- Verify both fields by showing an XY-plot of the fields.
- Create a pressure load, named **pressure_1**, and include it in the time dependent load case, **my_load_case_1**. Use the spatially and temporally varying fields to define the pressure variation and apply the pressure to the top surface of all the elements.
- Turn off the pressure labels so that only the pressure vectors are displayed.
- Turn off the pressure vectors and then verify the specified pressure loading by plotting contours of the pressure load.

Exercise Procedure:

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

File/New Database...

New Database Name **variable_loads**

OK

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

New Model Preference

Tolerance **Default**

Analysis Code: **MSC/NASTRAN**

OK

3. Create the geometry and finite element mesh using the information in Figure 11-1.

Create a surface

Geometry

Action: **Create**

Object: **Surface**

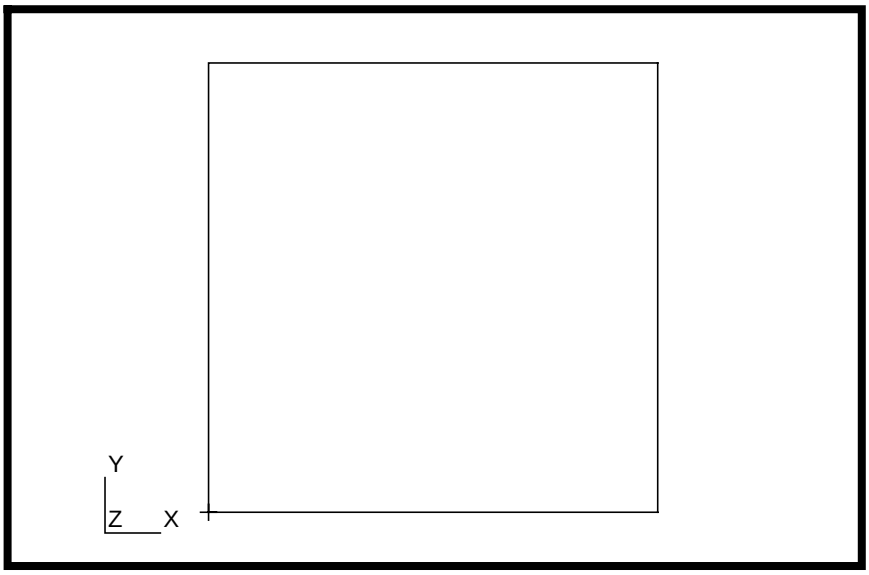
Method: **XYZ**

Vector Coordinate List **<10, 10, 0>**

Origin Coordinate List **[0, 0, 0]**

Apply

The surface is shown in the figure below.



Mesh the model

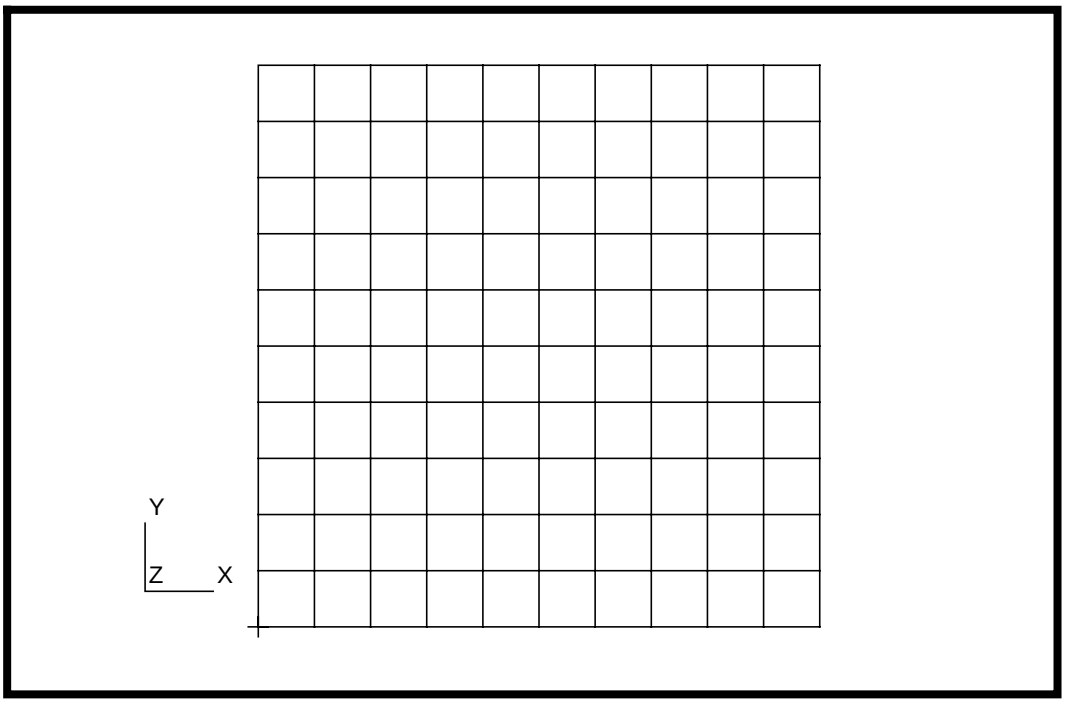
Now create the mesh for the model.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**1.0**
Element Topology	**Quad 4**
Surface List	**Surface 1**

Apply

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



4. Create a time dependent load case named **my_load_case_1**.

Create Load Case

Before you create the time dependent pressure load you must create a time-dependent load case.

Load Cases

Action:	**Create**
Load Case Name	**my_load_case_1**
Load Case Type	**Time Dependent**

Apply

The temporal and spatial fields will be created in two separate fields.

Create a Spatially Dependent Field

5. Define a Spatial field named, **pressure_spatial: 100*sinr(3.14159*'X/10)*sinr(3.14159*'Y/10)**.

Fields

Action: **Create**

Object: **Spatial**

Method: **PCL Function**

Field Name **pressure_spatial**

Field Type **Scalar**

Scalar Function ('X 'Y 'Z)

100*sinr(3.14159*'X/10)*sinr(3.14159*'Y/10)

Notice that the X and Y are preceded with a single quote and they are capitalized. In addition, the acceptable PCL syntax is written above the *Scalar Function* databox.

Below the *Scalar Function* databox, the **Independent Variables** are listed. Selecting any of these variables will automatically place it into the equation with the appropriate syntax.

Apply

Create a Time-Dependent Field

6. Define a Time-Dependent field named **pressure_temporal: cosr(10*'t)**.

Action: **Create**

Object: **Non-Spatial**

Method: **Tabular Input**

Field Name **pressure_temporal**

Active Independent Variables **Time**

Input Data...

Map Function to Table...

PCL Expression f('t) **cosr(10*'t)**

Start Time **0.0**

End Time **2.0**

Number of Points **30**

Apply

Cancel

OK

Apply

7. Verify the created fields using an XY-plot.

Verify the Created Field

Action: **Show**

Select Field to Show **pressure_temporal**

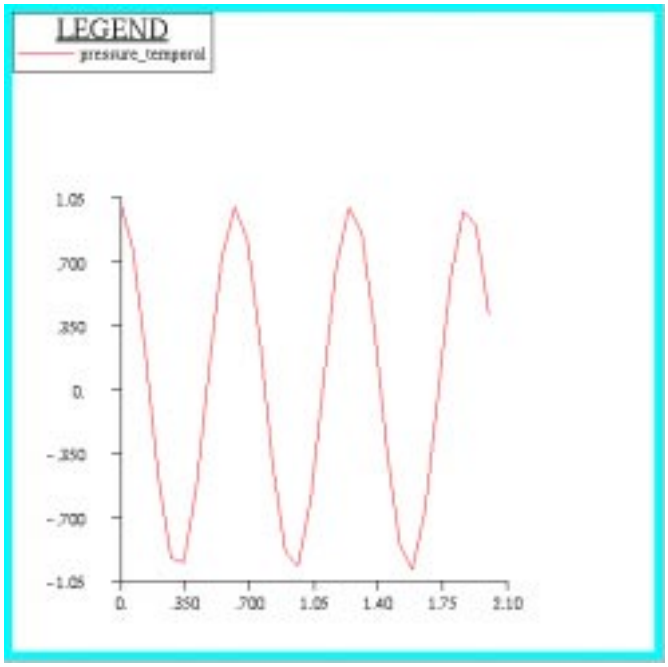
Specify Range...

Use Existing Points

OK

Apply

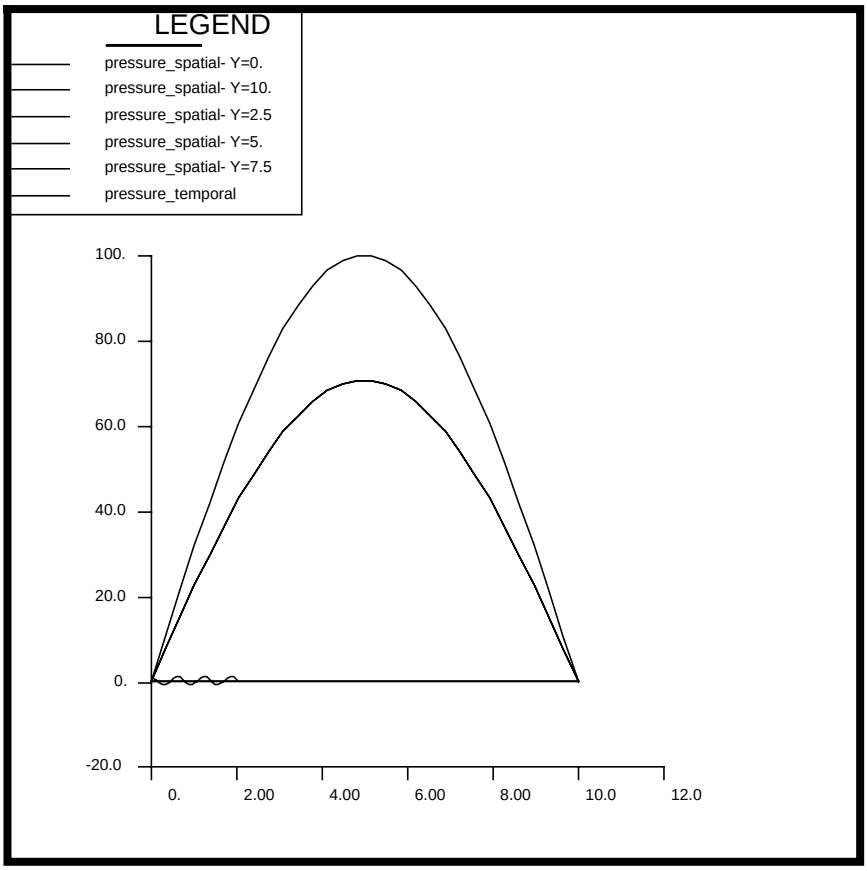
The XY plot is shown in the figure below.



A table called Plotted Curves will also be displayed, showing the actual data points plotted. Hit the **Cancel** button to close this form, or move it to the side.

To plot the **pressure_spatial** field, highlight it under *Select Fields to Show.* You may choose only one independent variable for the XY plots which means one of the variables will be held constant, while the other varies between user defined values.

For example, in the *Specify Range* form set X values between **0** and **10**, and the number of points to **30**. Set the range for Y values between **0** and **10**, and use **5** sets. The 5 sets for the Y scale represent the number of curves in the plot. Click on **OK** to close form and click on **Apply** to create and post the XY plot. The Y=0 and Y=10 curves are along the bottom axis and are difficult to see. Since the loading is symmetric, the Y=2.5 and Y =7.5 curves are identical and lie on top of each other. Only 1 color is plotted. A way to display the spatially varying pressure as a contour plot will be shown next.



When you are done viewing the xy plot, click on the **Unpost Current XY Plot** button.

Specify a Variable Load

8. Create a pressure load, named **pressure_1**, and include it in the time dependent load case, **my_load_case_1**. Use the spatial and temporal fields to define the pressure variation and apply the pressure to the top surface of all the elements.

◆ **Load/BCs**

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name	**pressure_1**
Current Load Case	**my_load_case_1**
Target Element Type:	**2D**

Input Data...

Top Surf Pressure	**f:pressure_spatial**
Time Dependence	**f:pressure_temporal**

OK

Select Application Region...

Geometry Filter ◆ **FEM**

Select 2D Elements or Edges Select All Elements

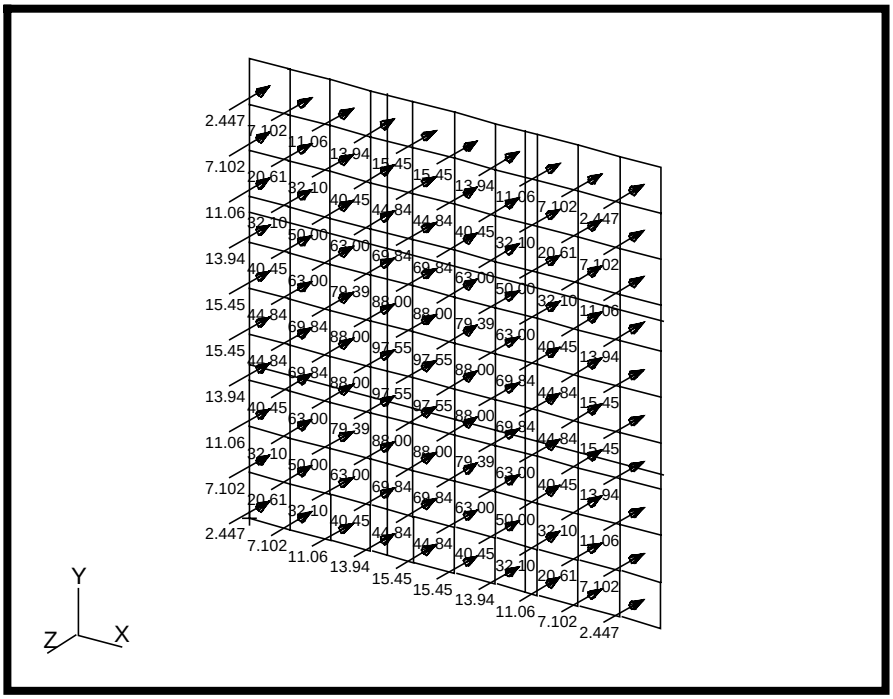
Add

OK

Apply

The pressure load set markers are drawn normal to the elements as shown in the figure below. Note that the view has been changed to

Iso 1 View so that the normal vectors can be seen clearly.



Attributes of the markers, such as color and display, may be changed in the **Display/Load/BC/Elem. Props…** menu accessed from the *Main Form*. Change the color of the pressure marker to another color.

9. Turn off the pressure labels so that only the pressure vectors are displayed.

Vector attributes, such as pressure labels, coloring method and vector size, may be modified in the **Display** menu**.**

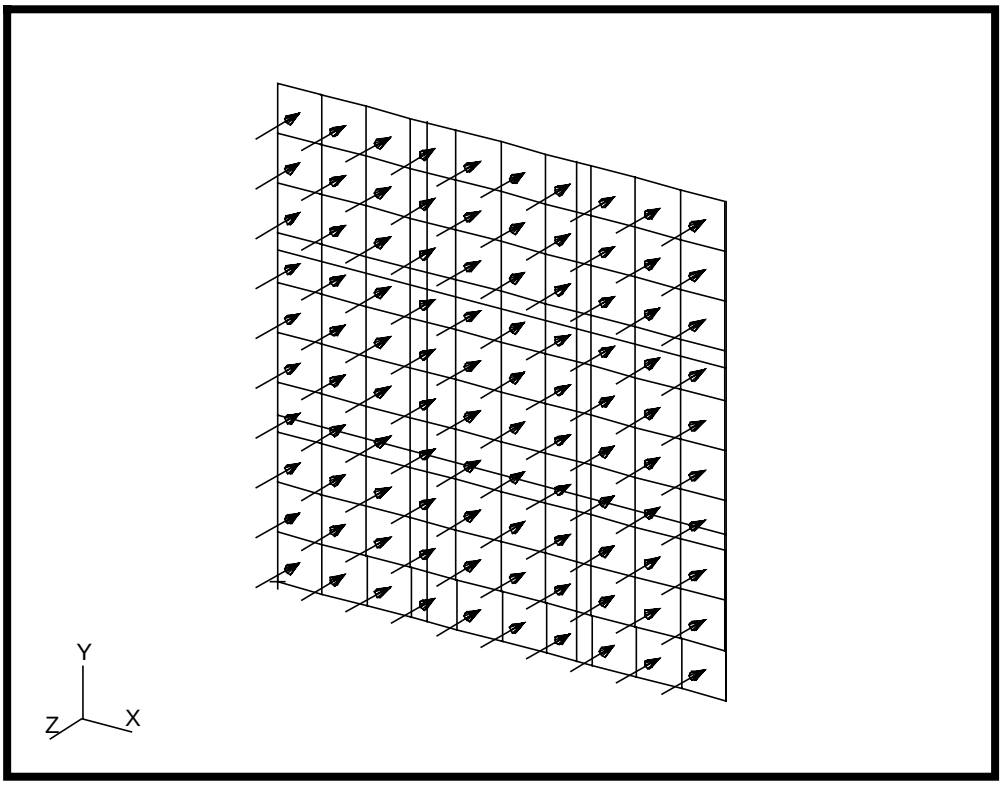
Display/Load/BC/Elem. Props...

Vectors/Fields...

☐ **Show LBC/El. Prop. Values**

Apply

Your model should look like the one shown below.



10. Turn off the pressure vectors and then verify the specified pressure loading by plotting contours of the pressure load.

Create an Element Fill Plot

Display/Load/BC/Elem. Props...

Pressure

Apply

Cancel

Load/BCs

Action:	**Plot Contours**
Object:	**Pressure**
Existing Sets	**pressure_1**
Select Data Variable	**Top Surf Pressure**

Time **0.0**

Select Groups **default_group**

Apply

You may need to reset the range to span the actual property range.

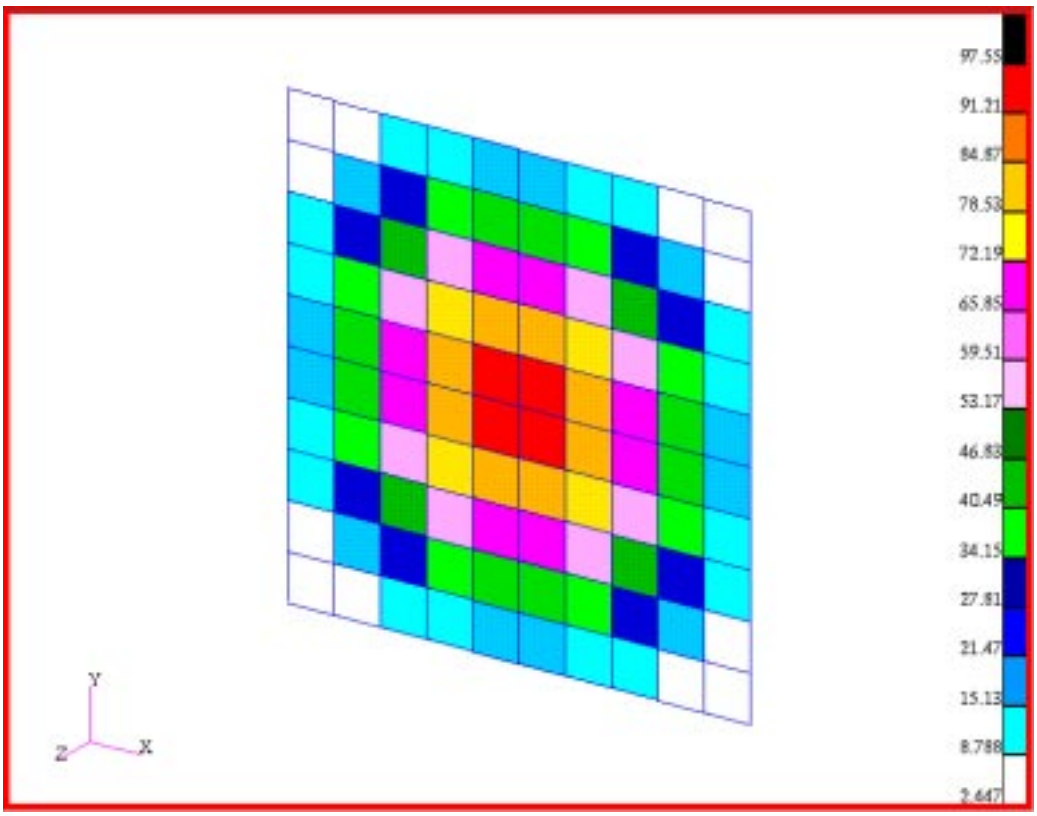
Display/Ranges...

Fit Results

Calculate

Apply

Your screen should appear as below



To complete the exercise, you need to close the database.

File/Quit