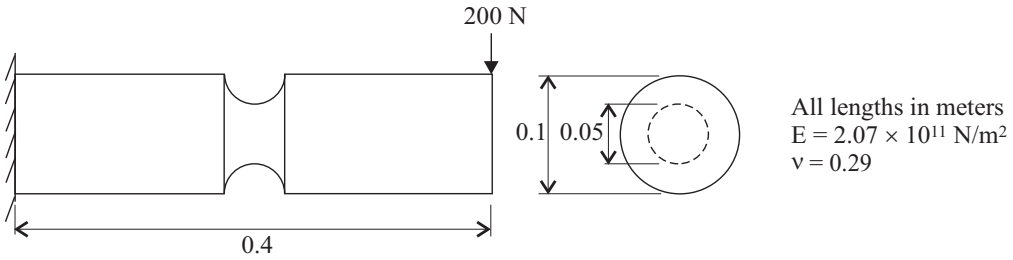
Problem description

A round bar with circumferential groove is subjected to a tip load as shown:



In this problem solution, we will demonstrate the following topics that have not been demonstrated in previous problems:

- Revolving a mesh of 2D elements to obtain a mesh of 3D elements
- Using an SCL line to examine the results

We assume that you have worked through problems 1 to 4, or have equivalent experience with the ADINA System.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem cannot be solved with the 900 nodes version of the ADINA System (there are 8483 nodes in this model).

Invoking the AUI and choosing the finite element program

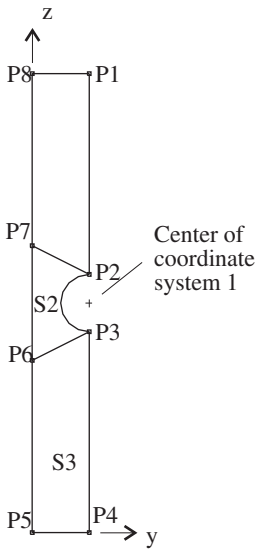
Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 5: Round bar with circumferential groove subjected to tip load" and click OK.

Defining model geometry

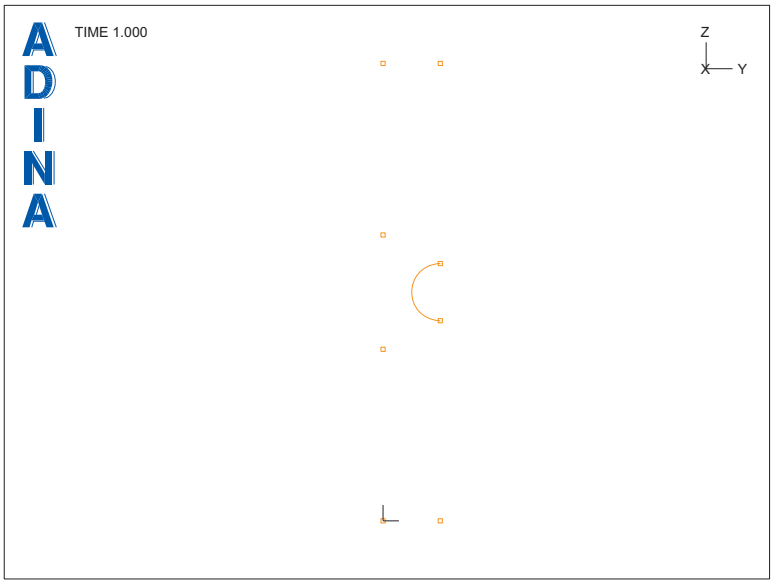
Here is a diagram showing the key geometry used in defining this model:



Geometry points: Click the Define Points icon and enter the following information into the table. You can leave the X1 column blank. Then click OK.

Point #	X2	X3
1	0.05	0.4
2	0.05	0.225
3	0.05	0.175
4	0.05	0.0
5	0.0	0.0
6	0.0	0.15
7	0.0	0.25
8	0.0	0.4

Geometry lines: Now we define the arc line. For the definition, it is convenient to place a coordinate system at the center of the arc. Click the Coordinate Systems icon , add coordinate system 1, set the Origin to (0.0, 0.05, 0.2) and click OK. Now click the Define Lines icon , add line 1, set the Type to Revolved, set the Initial Point to 2, the Angle of Rotation to 180, make sure that the Axis is set to X and click OK. The graphics window should look something like the figure on the next page.



Since we are done with the coordinate system, let's reset the default coordinate system back to the original coordinate system. Click the Coordinate Systems icon, click the Set Global button and click OK.

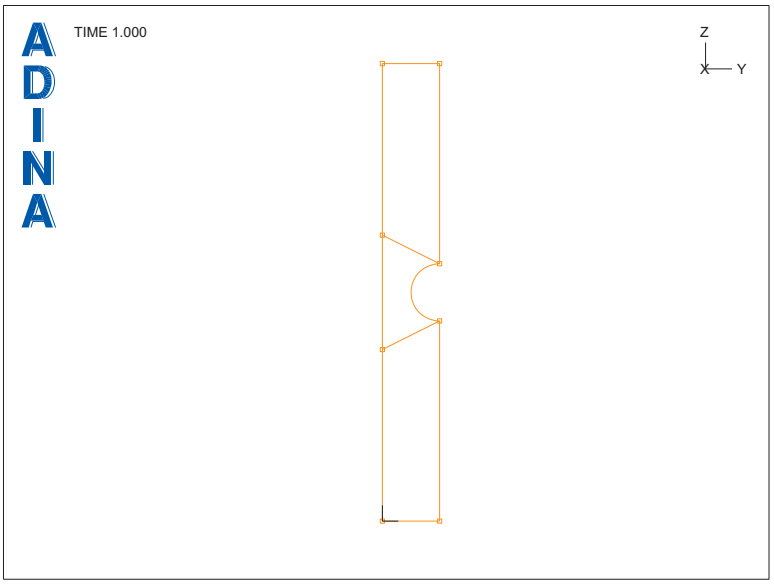
Geometry surfaces: Click the Define Surfaces icon, define the following surfaces and click OK.

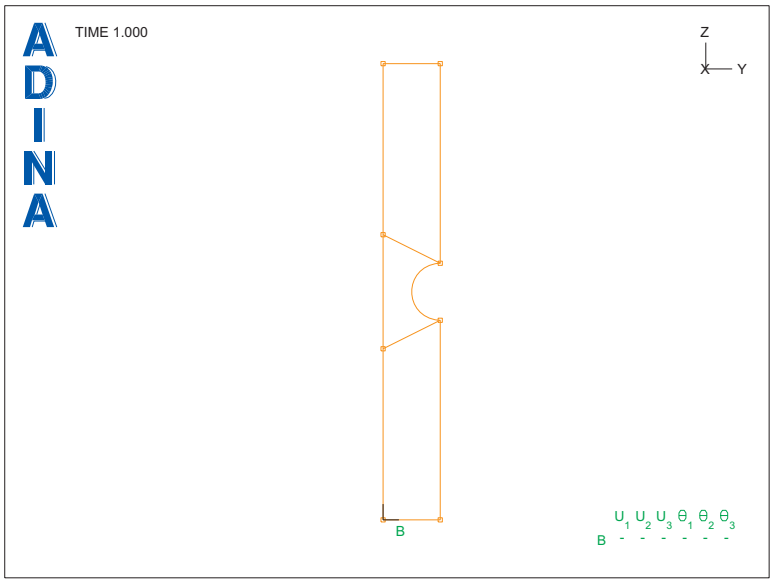
Surface Number	Type	Point 1	Point 2	Point3	Point 4
1	Vertex	1	8	7	2
2	Vertex	2	7	6	3
3	Vertex	3	6	5	4

The graphics window should look something like the top figure on the next page.

Applying boundary conditions

We will fix the line at z = 0. To determine this line number, click the Query icon and click on the lowest horizontal line until the line number appears in the message window (it should be 9). Now click the Apply Fixity icon, set the "Apply to" field to Edge/Line, enter 9 in the first row and column of the table and click OK. When you click the Boundary Plot icon, the graphics window should look something like the bottom figure on the next page.

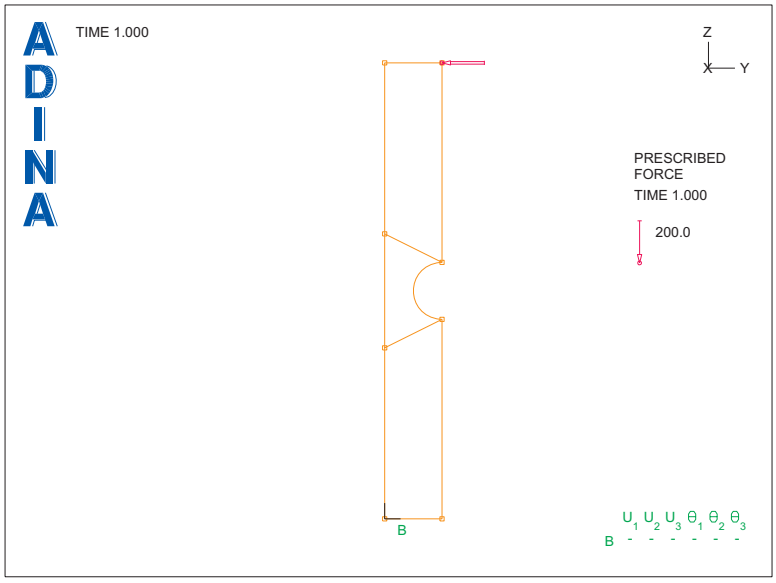

ADINA
TIME 1.000
Z
X
Y


ADINA
TIME 1.000
Z
X
Y
B
U1 U2 U3 Θ1 Θ2 Θ3
B - - - - - -

Defining and applying loads

Click the Apply Load icon , make sure that the Load Type is Force and click the Define... button to the right of the Load Number field. In the Define Concentrated Force dialog box, add load 1, set the Magnitude to 200, the Direction to (0, -1, 0) and click OK. In the Apply Load dialog box, set the Point # to 1 and click OK.

When you click the Load Plot icon , the graphics window should look something like this:



Defining the material

Click the Manage Materials icon and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 2.07E11, the Poisson's ratio to 0.29 and click OK. Click Close to close the Manage Material Definitions dialog box.

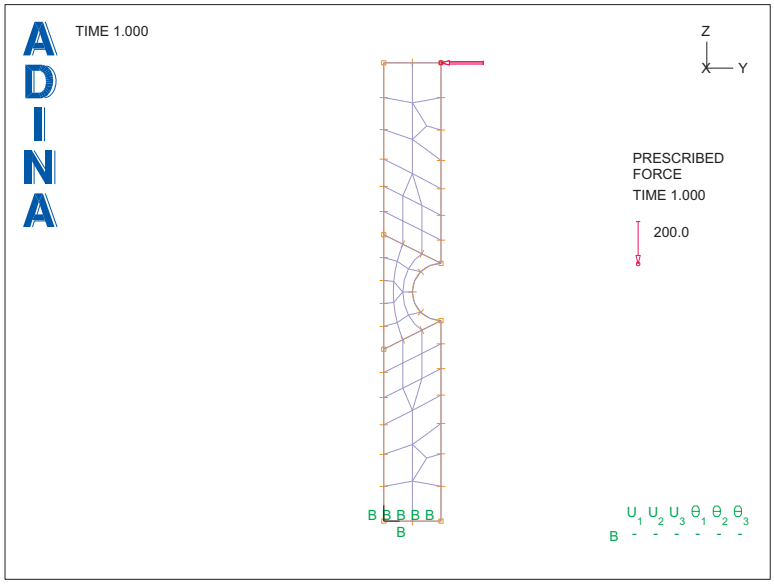
Defining the elements

Element group: Click the Element Groups icon , add element group number 1, set the Type to 2-D Solid and click OK.

Subdivision data: In this mesh, we will assign a uniform point size to the points of the model, then assign a smaller point size to the points near the groove. Choose Meshing→ Mesh Density→Complete Model, make sure that the "Subdivision Mode" is set to "Use End-Point Sizes" and click OK. Now choose Meshing→Mesh Density→Point Size, set the "Points Defined from" field to "All Geometry Points", set the Maximum to 0.03 and click Apply. Then, in the table in the Define Point Size dialog box, set the Mesh Size for Point Labels 2, 3, 6, 7 to 0.02 and click OK.

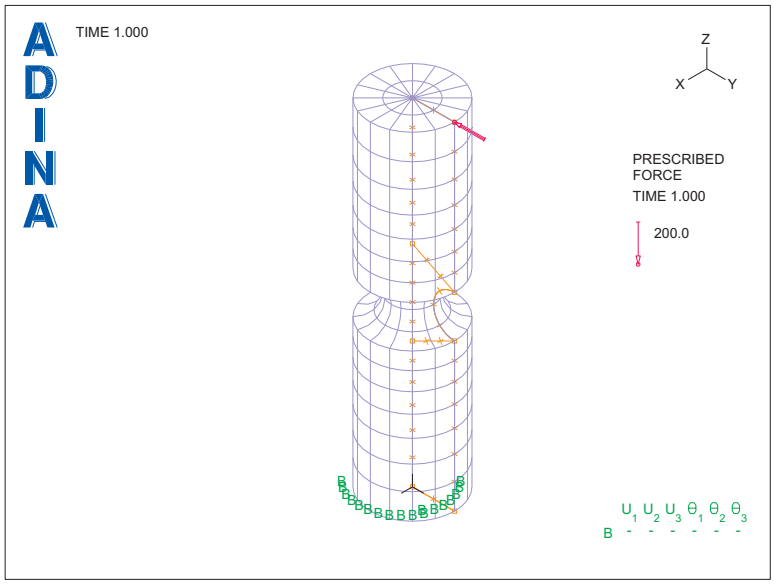
2D element generation: Click the Mesh Surfaces icon  , enter surfaces 1, 2, 3 in the first three rows of the table and click OK. The graphics window should look something like this:

.



3D element generation: Choose Meshing→Mesh Revolve, set the Angle of Revolution to 360, the "No. of Elements in Revolve Direction" to 16, the Z Direction of Axis to 1.0, and, in the Options tab, check the "Check Coincidence" button, and click OK.

When you click the Iso View 1 icon  , the graphics window should look something like the figure on the next page. You can rotate the mesh plot with the mouse to make sure that all of the nodes at z=0 are fixed.



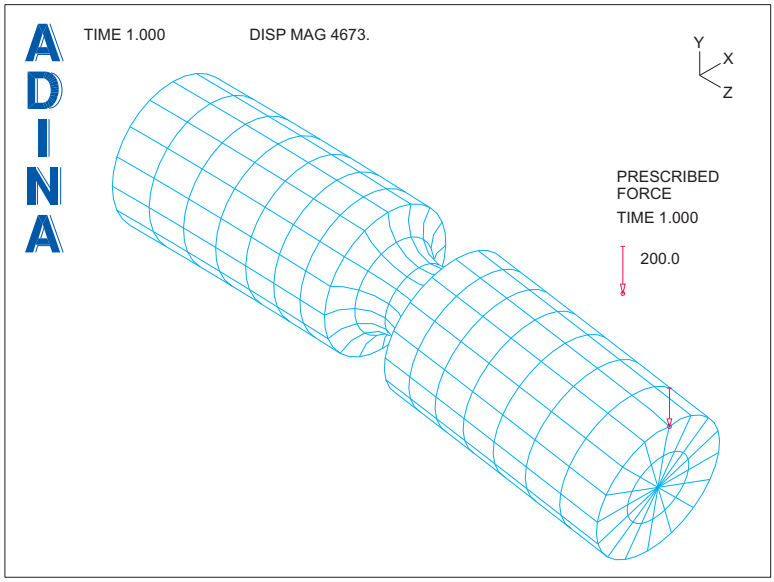
Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

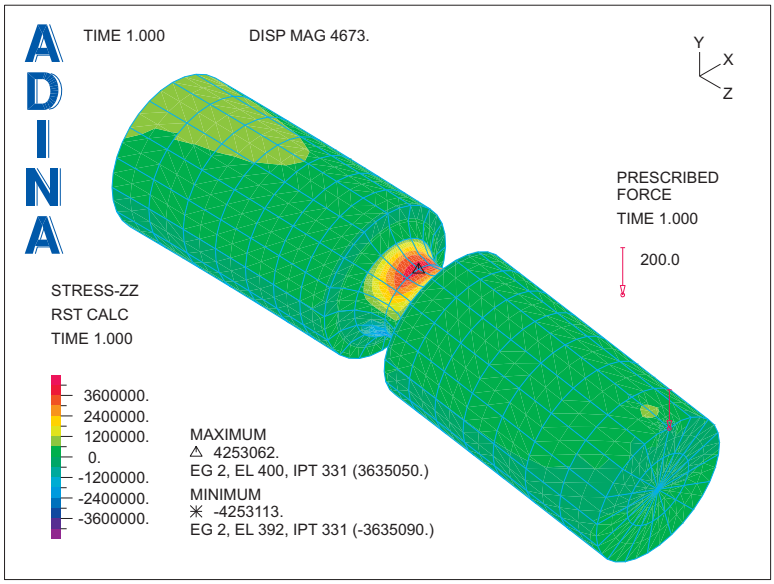
First click the Save icon and save the database to file `prob05`. To generate the ADINA Structures data file and run ADINA Structures, click the Data File/Solution icon, set the file name to `prob05`, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob05`.

Examining the solution

Click the Iso View 2 icon, then click the Load Plot icon to display the finite element mesh and loads. To magnify the displacements, click the Scale Displacements icon. The graphics window should look something like the top figure on the next page.

Stress band plot: Click the Create Band Plot icon, set the Band Plot Variable to (Stress:STRESS-ZZ) and click OK. Move the objects in the graphics window until the graphics window looks something like the bottom figure on the next page.


ADINA
TIME 1.000
DISP MAG 4673.
Y
X
Z
PRESCRIBED
FORCE
TIME 1.000
200.0

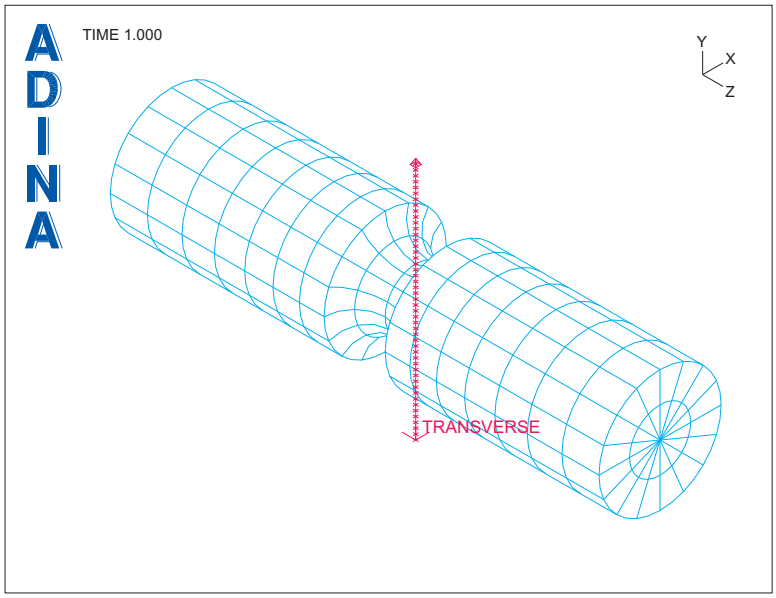

ADINA
TIME 1.000
DISP MAG 4673.
Y
X
Z
PRESCRIBED
FORCE
TIME 1.000
200.0
STRESS-ZZ
RST CALC
TIME 1.000
3600000.
2400000.
1200000.
0.
-1200000.
-2400000.
-3600000.
MAXIMUM
△ 4253062.
EG 2, EL 400, IPT 331 (3635050.)
MINIMUM
✳ -4253113.
EG 2, EL 392, IPT 331 (-3635090.)

Results along a line:

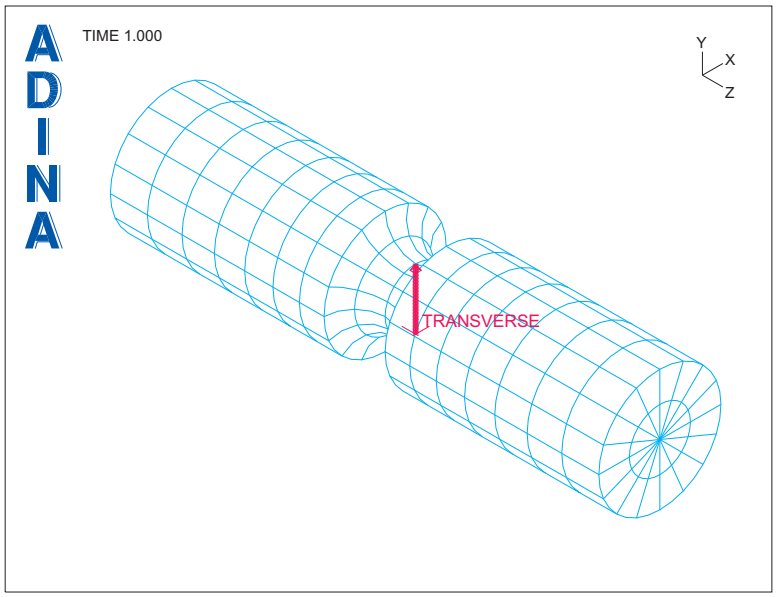
We would like to examine the results along a straight line through the groove in the y direction.

Click the Clear icon and the Iso View 2 icon .

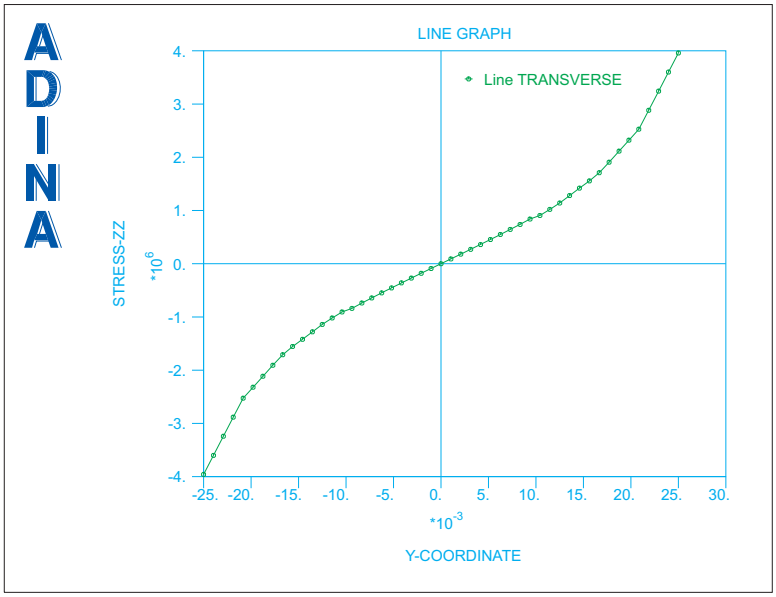
Choose Definitions→Model Line→Stress Classification Line, add line TRANSVERSE, set (X1, Y1, Z1) to (0, -0.1, 0.201), (X2, Y2, Z2) to (0, 0.1, 0.201) and click OK. Then choose Display→Result Line Plot→Create and click OK. The graphics window should look something like this:



We would like to trim the line so that it does not extend outside the mesh. Choose Definitions→Model Line→Stress Classification Line, set the Trimming Tolerance to 0.001 and click OK. When you click the Redraw icon , the graphics window should look something like the top figure on the next page.

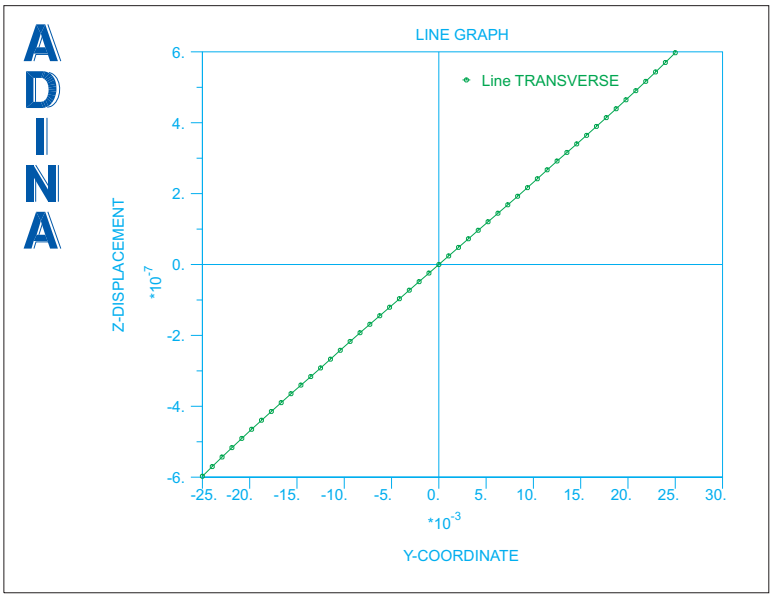


Now click the Clear icon CLEAR, choose Graph→Response Curve (Model Line), set the X Coordinate Variable to (Coordinate: Y-COORDINATE), set the Y Coordinate Variable to (Stress: STRESS-ZZ) and click OK. The graphics window should look something like this:



Choose Graph→List. The value of STRESS-ZZ at coordinate -2.50000E-02 should be -3.95895E+06 (Pa). Click Close to close the dialog box.

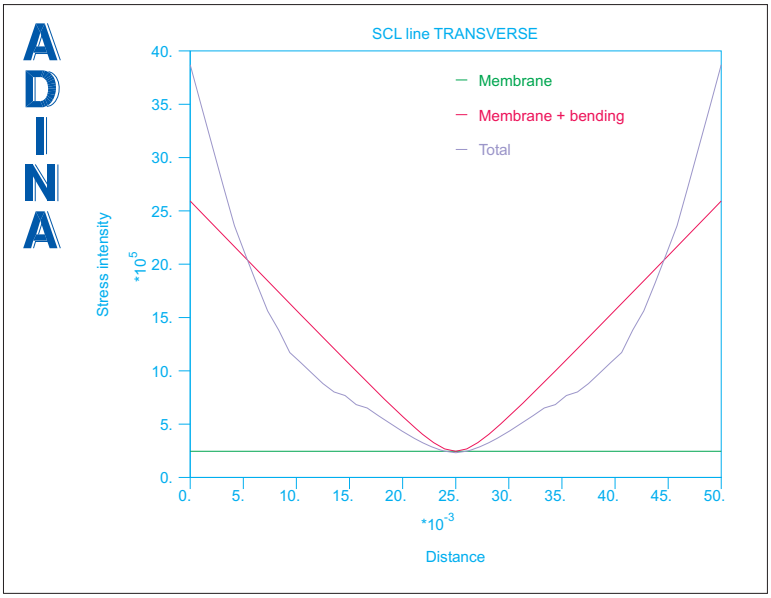
Just to show that the SCL line can be used for results other than stresses, we now graph the displacements along the line. Click the Clear icon  , choose Graph→Response Curve (Model Line), set the X Coordinate Variable to (Coordinate: Y-COORDINATE), set the Y Coordinate Variable to (Displacement: Z-DISPLACEMENT) and click OK. The graphics window should look something like this:



When you choose Graph→List, the value of Z-DISPLACEMENT at coordinate -2.50000E-02 should be -5.97415E-07 (m). Click Close to close the dialog box.

ASME NB-3200 stress linearization calculations: One important use of the SCL line is for stress linearization according to ASME NB-3200 (for nuclear power plant stress analysis). Click the Clear icon  , choose Graph→Stress Linearization on SCL Line and click OK. The graphics window should look something like the top figure on the next page.

Note, the stress intensity is defined as the difference between the maximum and minimum principal stresses. The stress intensity is not the same as the effective stress.



Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.