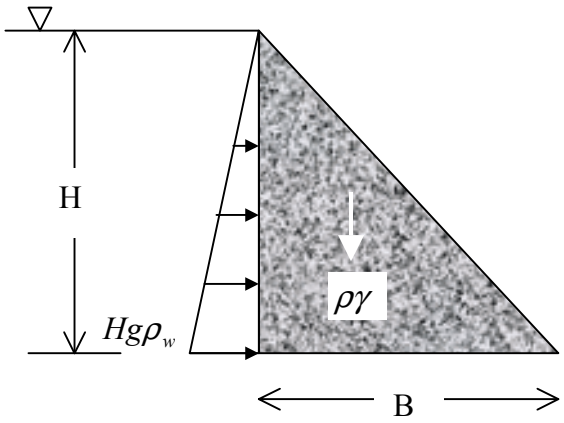
FE Modeling Example Using ADINA



H = B = 10 m
g = 9.81 m/s^2

$\rho = 2400$ kg/m^3

$\rho_w = 1000$ kg/m^3

Invoke the ADINA user interface (AUI) by typing:

> cd /tmp
> add adina
> aui7.3

and press the ADINA-IN button in the ADINA system window.

Defining model control data

Master degrees of freedom: From the ADINA-IN menu, choose Control->Degrees of Freedom and pull out the X-Translation, X-Rotation, Y-Rotation, and Z-Rotation buttons and press OK. Note that in the ADINA system, 2D problems are entered in the YZ-plane.

The Degrees of Freedom window should look like Figure 2:

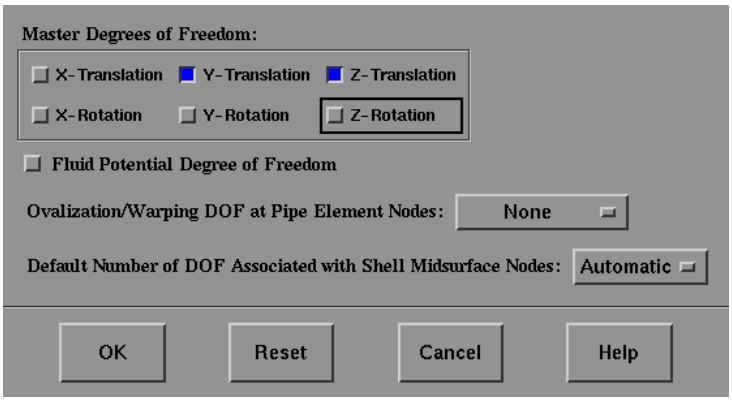


Figure 2

Defining Model Geometry

From the menu, choose Geometry->Points... and enter the values in Figure 3:

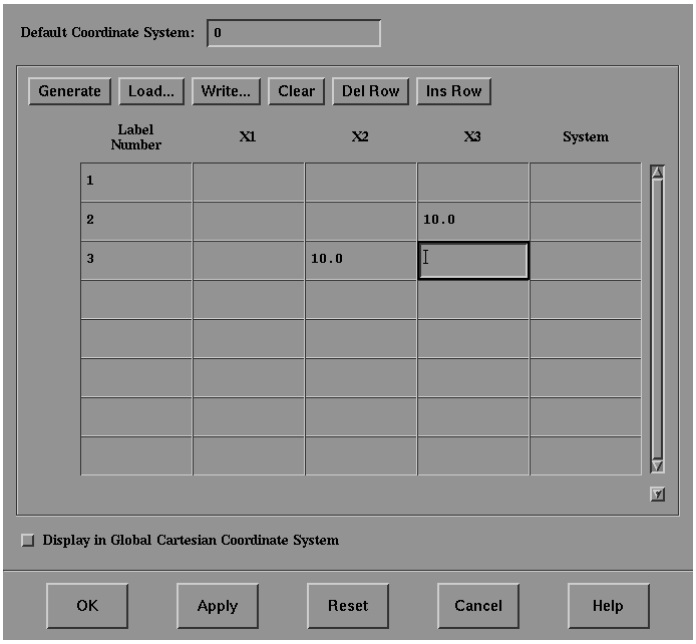


Figure 3

Empty fields are taken as zero by ADINA.

From the menu, choose Geometry->Surfaces->Define, press Add... and press OK in the popup box. Enter the following values in the Vertices table and press OK:

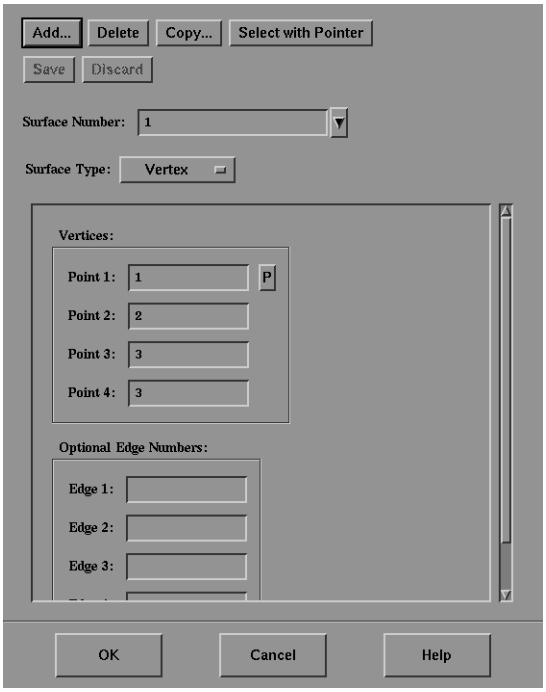


Figure 4

Note that we entered the same node for the third and fourth corners of the surface to create a triangle.

At this point, the ADINA-IN window should look like Figure 5:

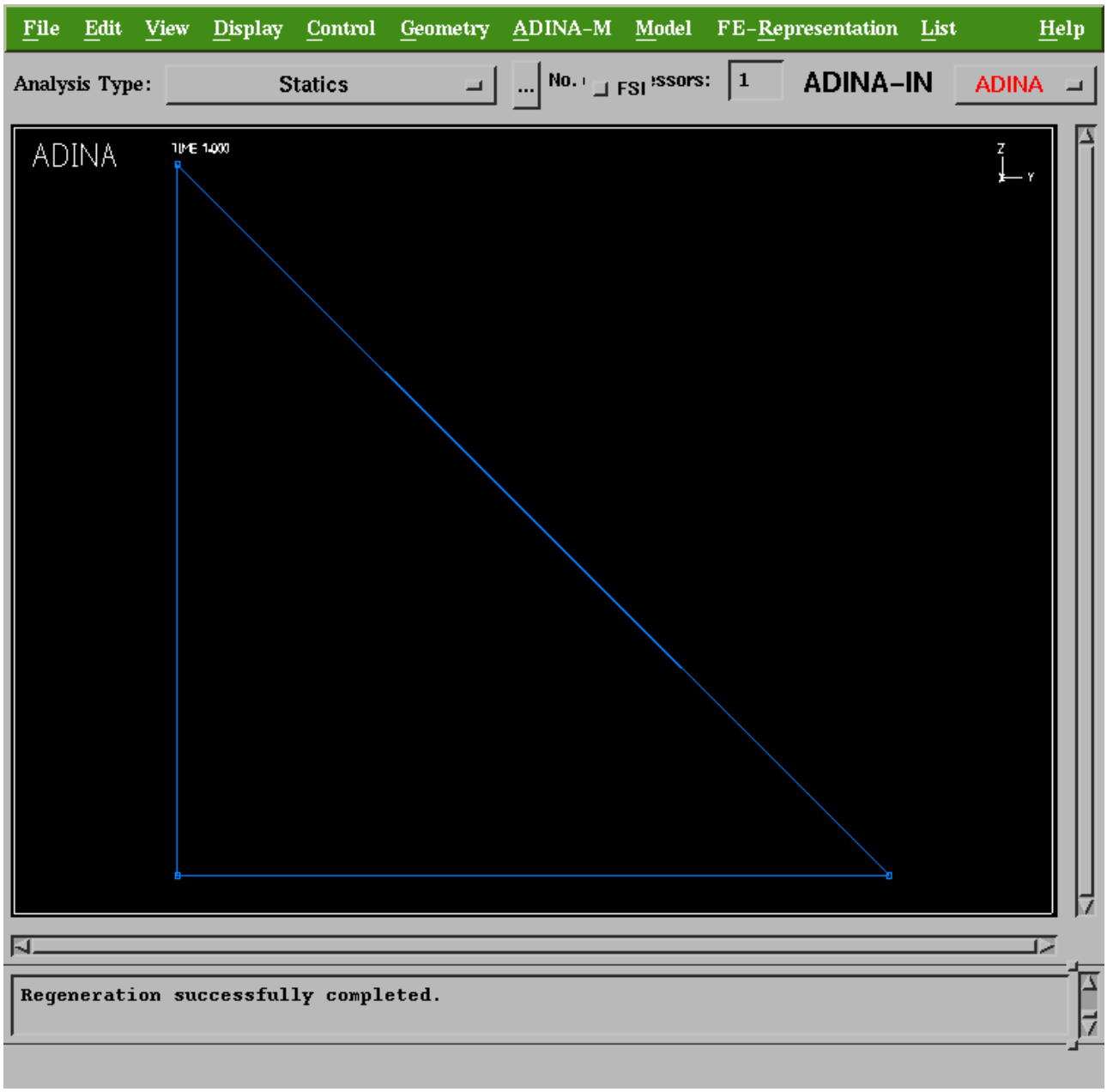


Figure 5

Defining and Applying Boundary Conditions

From menu, choose Model->Boundary Conditions->Fixity->Apply (or click on Apply Fixity button in the Modeling toolbox), and enter the following data and press OK:

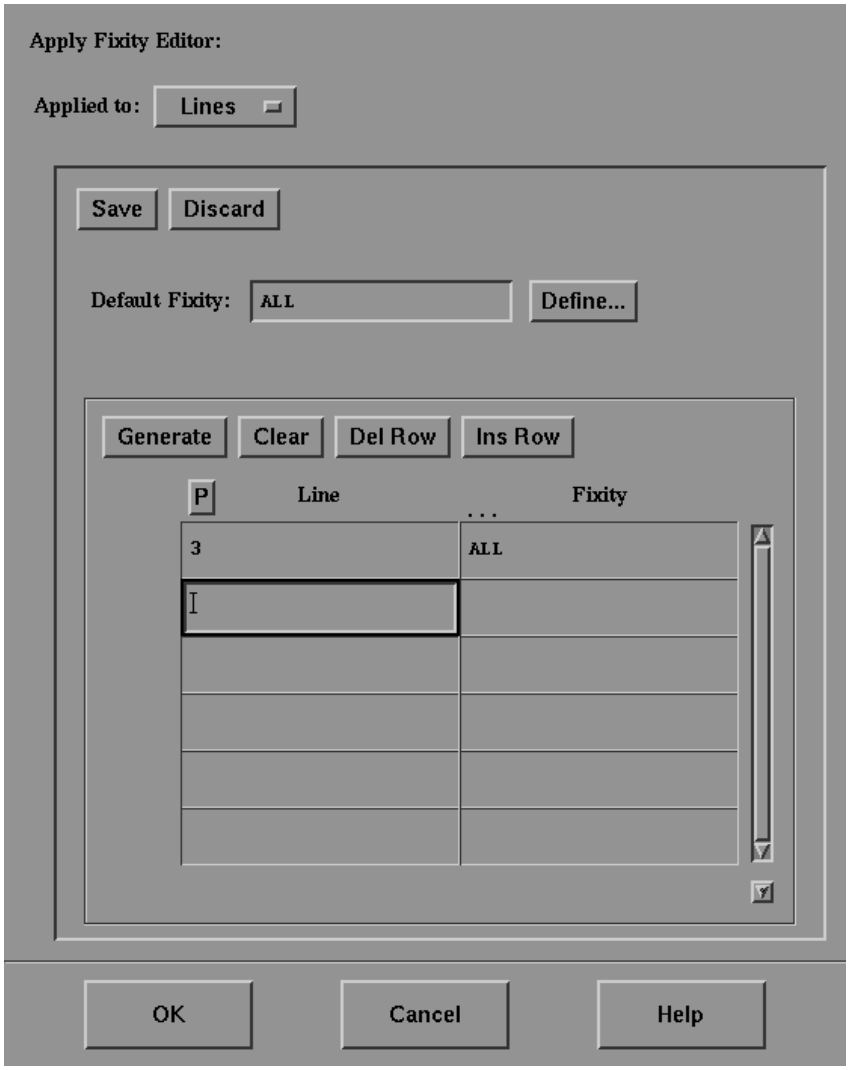


Figure 6

In cases where you don't know the line or node number, you can click on the P button and then click on the line or point of interest in the ADINA-IN window to find out its number.

To check the boundary conditions, from the menu choose Display->Show Boundary Conditions->All. The ADINA-IN window should look like Figure 7:

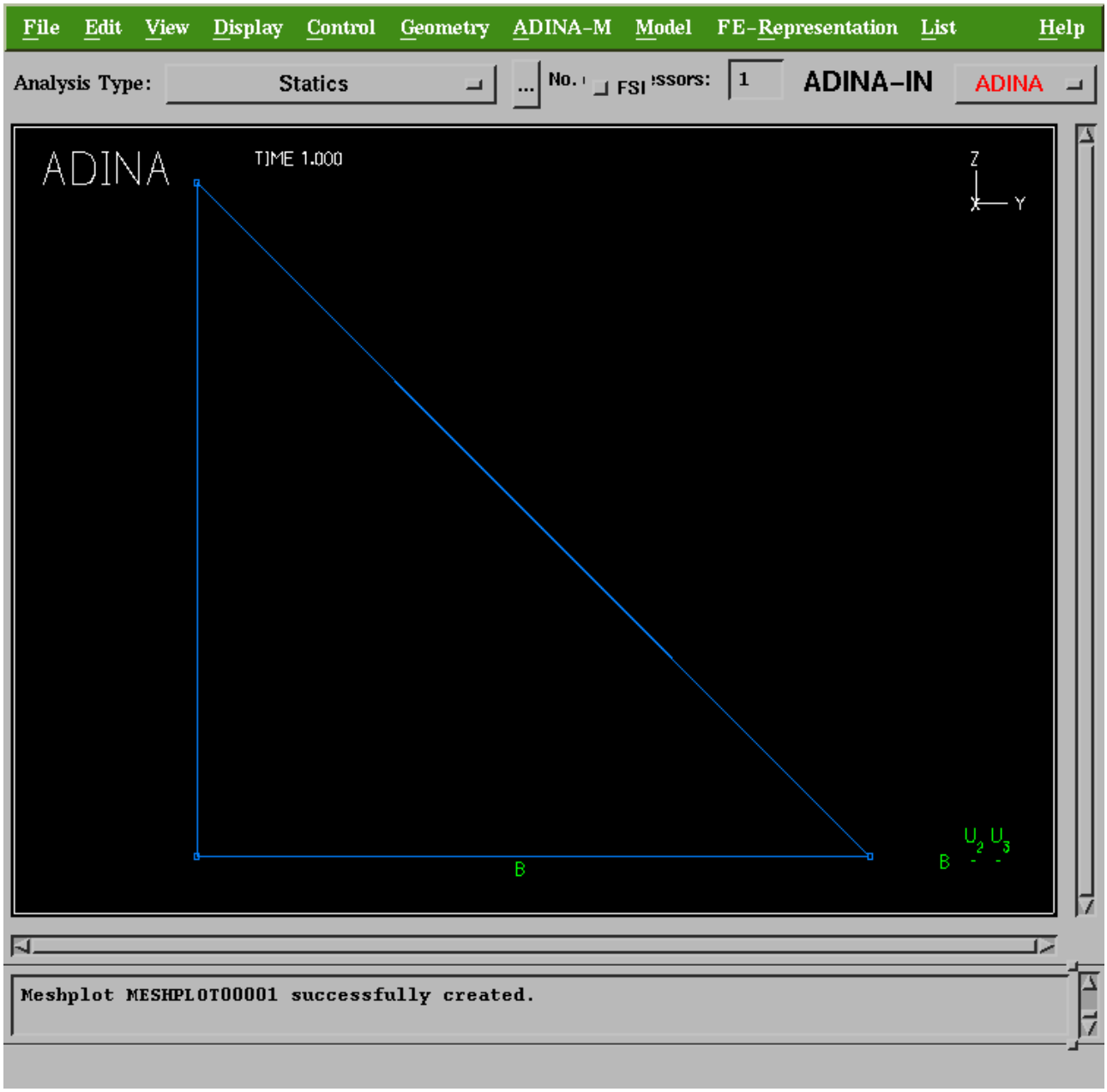


Figure 7

Defining and Applying Loads

To apply the gravitational body force, from the menu, choose Model->Loading->Apply (or click on the Apply Load button in the Modeling toolbox). Add load application number 1 as in Figure 8 (press Add... and and then press OK in the popup window). Choose Mass Proportional for the Load Type, define the Load Label as shown in Figure 9 and make sure that Site Type reads Model. Then press OK.

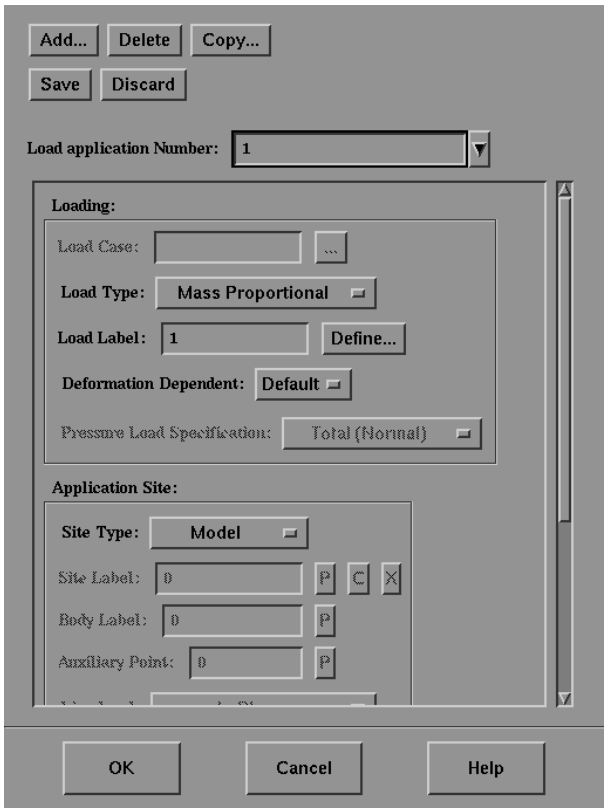


Figure 8

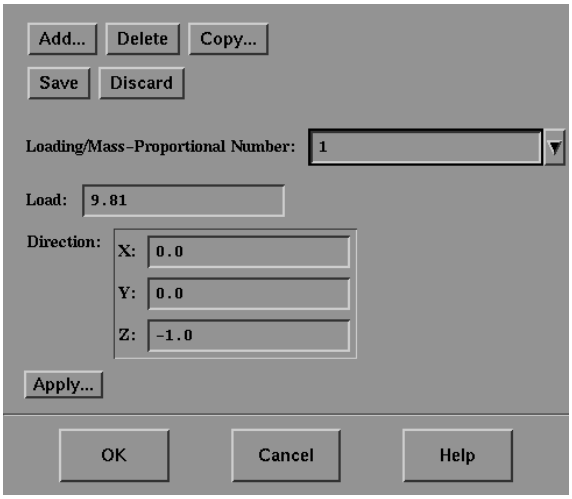


Figure 9

Since the water pressure changes with height, we first need to define a spatial function by choosing Geometry->Spatial Functions->Line, add Line Function Number 1, choose Type of data variation as Linear, and enter the values shown in Figure 10.

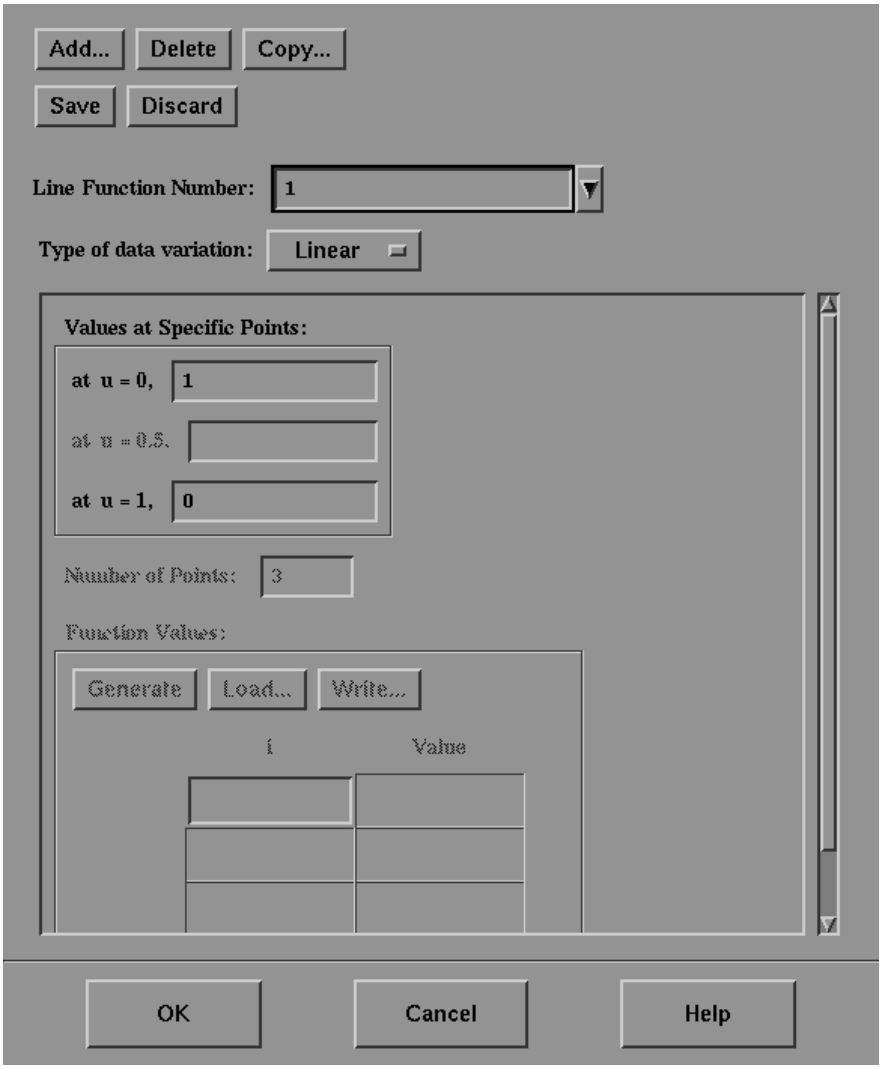


Figure 10

Now we can apply the water pressure by choosing Model->Loading->Apply (or click on the Apply Load button in the Modeling toolbox), add Load application number 2 as shown in Figure 11, choose Load Type as Pressure, define Load Label 1 as shown in Figure 12, define Site Type as Line, Site Label as 1, and Spatial Data Variation Label as 1. Then press OK.

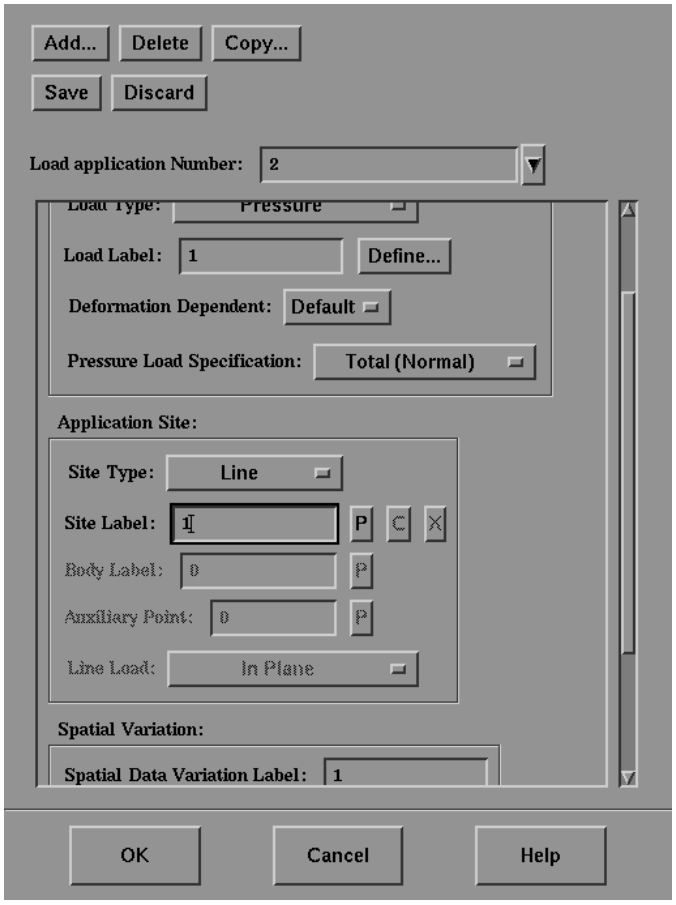


Figure 11

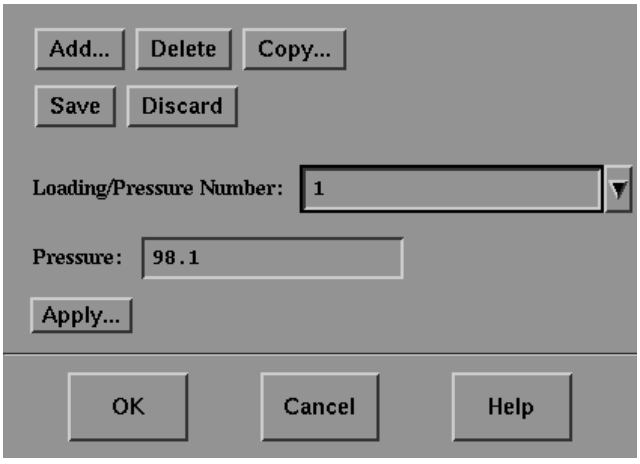


Figure 12

To display the water pressure, from the menu choose Display->Load Plot->Use Default (or click on the Load Plot button in the General toolbox). The ADINA-IN plot window should look like Figure 13.

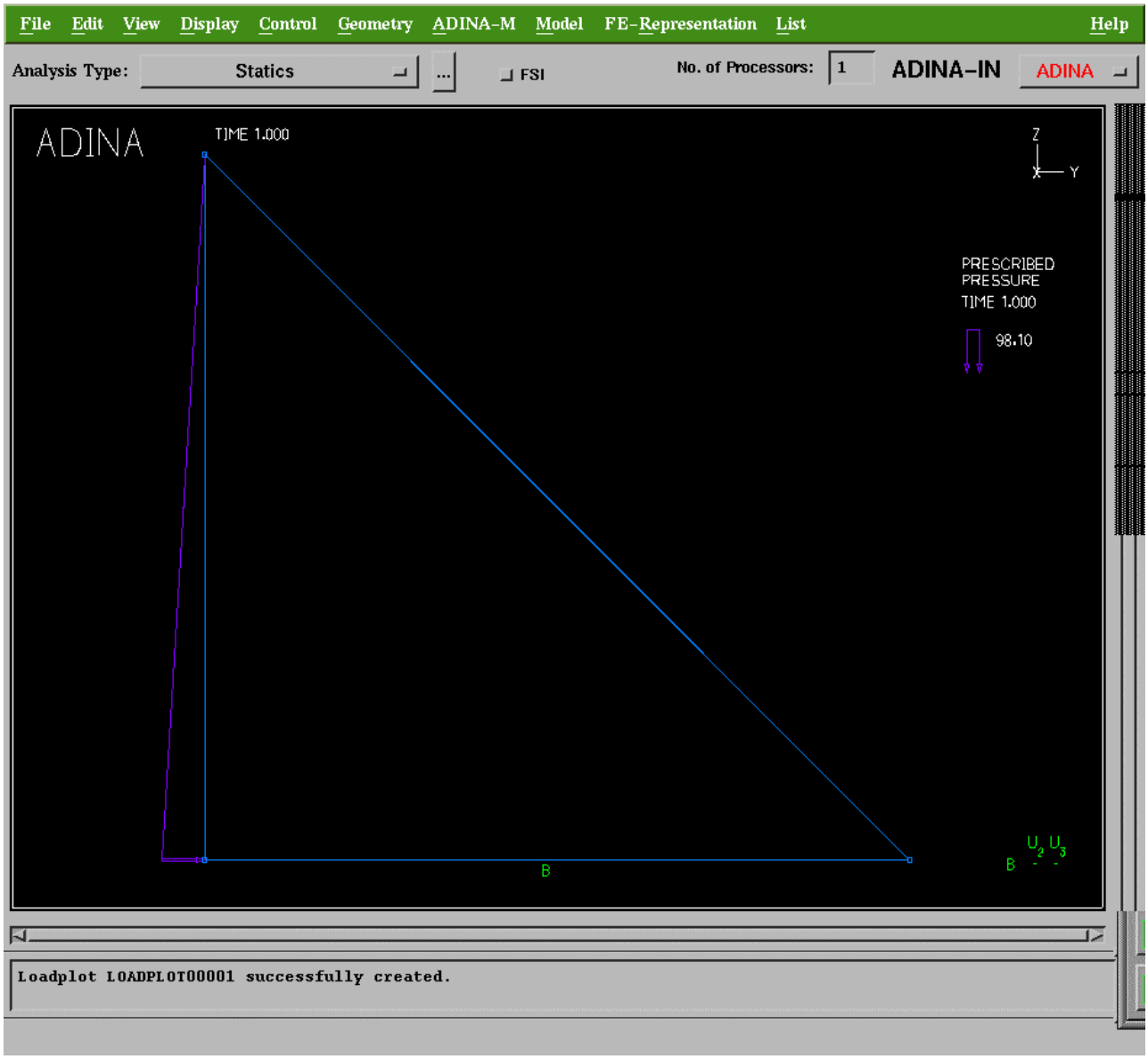


Figure 13

Defining the Material

Choose Model->Materials->Elastic->Isotropic, add Elastic Isotropic Material Number 1 and enter the Elastic modulus, Poisson's ratio, and density of the material as shown in Fig.14 and press OK.

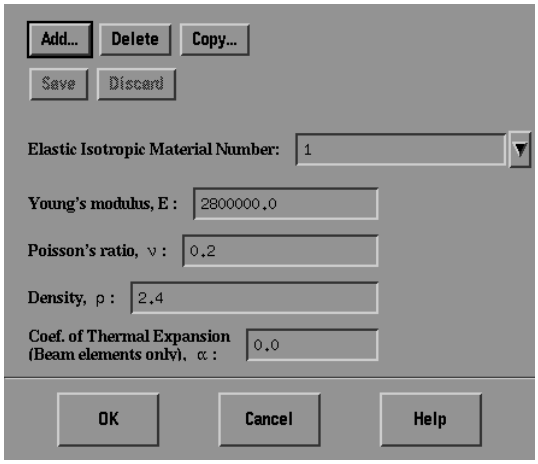


Figure 14

Defining the Elements

Choose FE-Representation->Element Groups->Define, add Element Group Number 1, select Element Type as 2-D Solid, Type of 2D Solid Element as Plane Strain, and press OK (Figure 15).

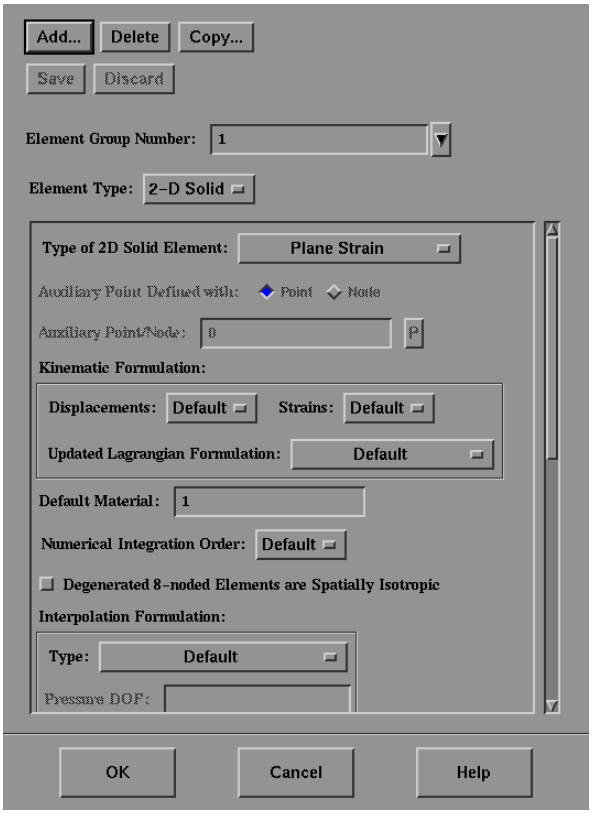


Figure 15

To generate meshes, choose FE-Representation->Mesh->Density->Surface, and select Mode of Subdivision on Lines as Use Number of Divisions. For this example, a 2x2 mesh density is used. Finally enter Surface Label as 1 and press OK (Figure 16).

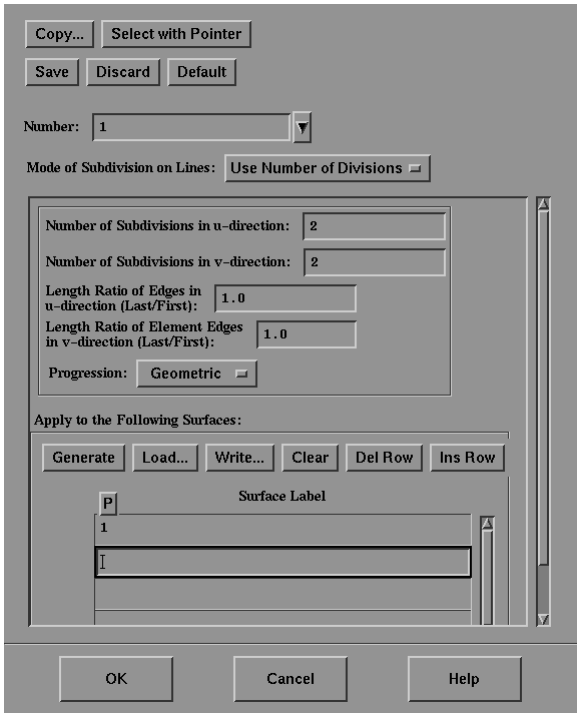


Figure 16

To plot the mesh, choose FE-Representation->Mesh->Create->Surface, choose Number of Nodes per Element and Preferred Cell Shape (3 node triangular element is selected for this example), enter Surface Label as 1, and press OK (Figure 17).

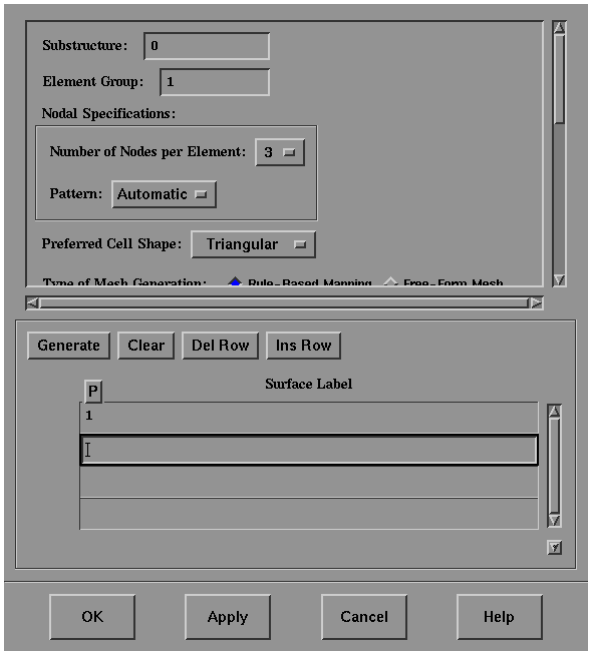


Figure 17

After mesh generation and plotting, the ADINA-IN window looks like in Figure 18.

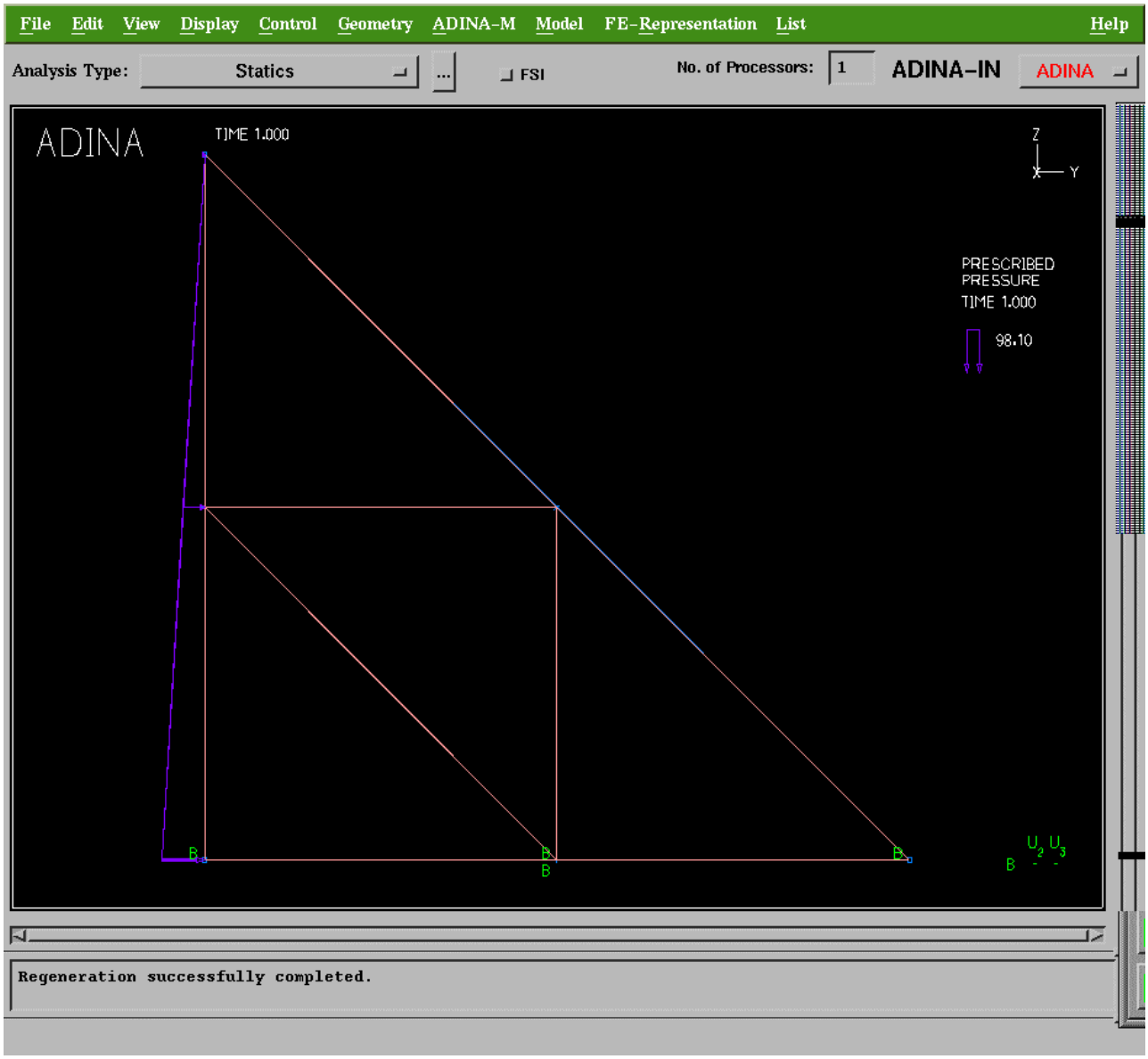


Figure 18

Generating the ADINA-IN Data File

From the menu, choose File->Datafile, enter example.dat for the data file name, and press OK.

Saving the ADINA-IN Database

From the menu, choose File->Close, and enter example.idb for the file name and press OK.

Running ADINA

When ADINA-IN window is closed, chose ADINA from the ADINA system window. From the menu, choose Job->Start and select example.dat data file. When the analysis is complete, choose File->Close and return to ADINA system window.

Plotting the Deformed Shape

Choose ADINA-PLOT from the ADINA system window. From menu, choose File->Load Porthole, and select example.port file. To plot the deformed shape choose Display->Geometry/Mesh Plot->Use Default. To adjust the magnification and to plot the original shape, choose Display->Geometry/Mesh Plot->Modify. Press the Model Depiction button, press the "Display the Original Mesh" button, press the Length button in the Displacement Display Options box and set the Max. Displacement Length to 5,

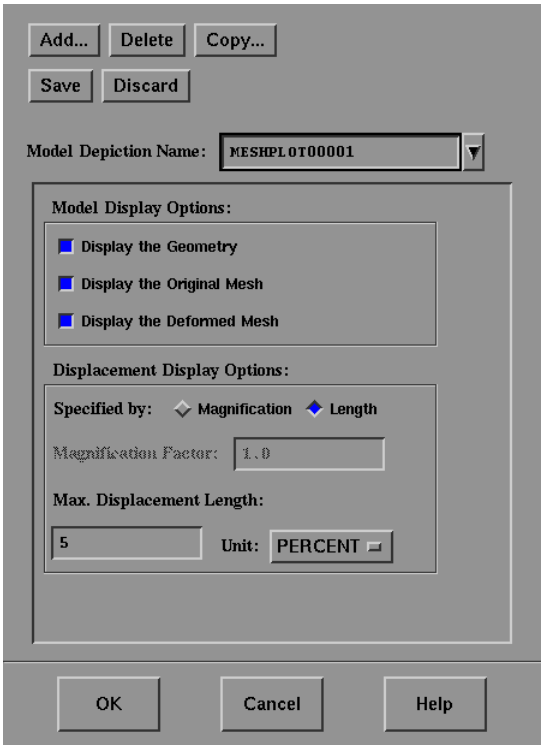


Figure 19

then press OK twice to close both dialog boxes.

The graphics window should now look like Figure 20.

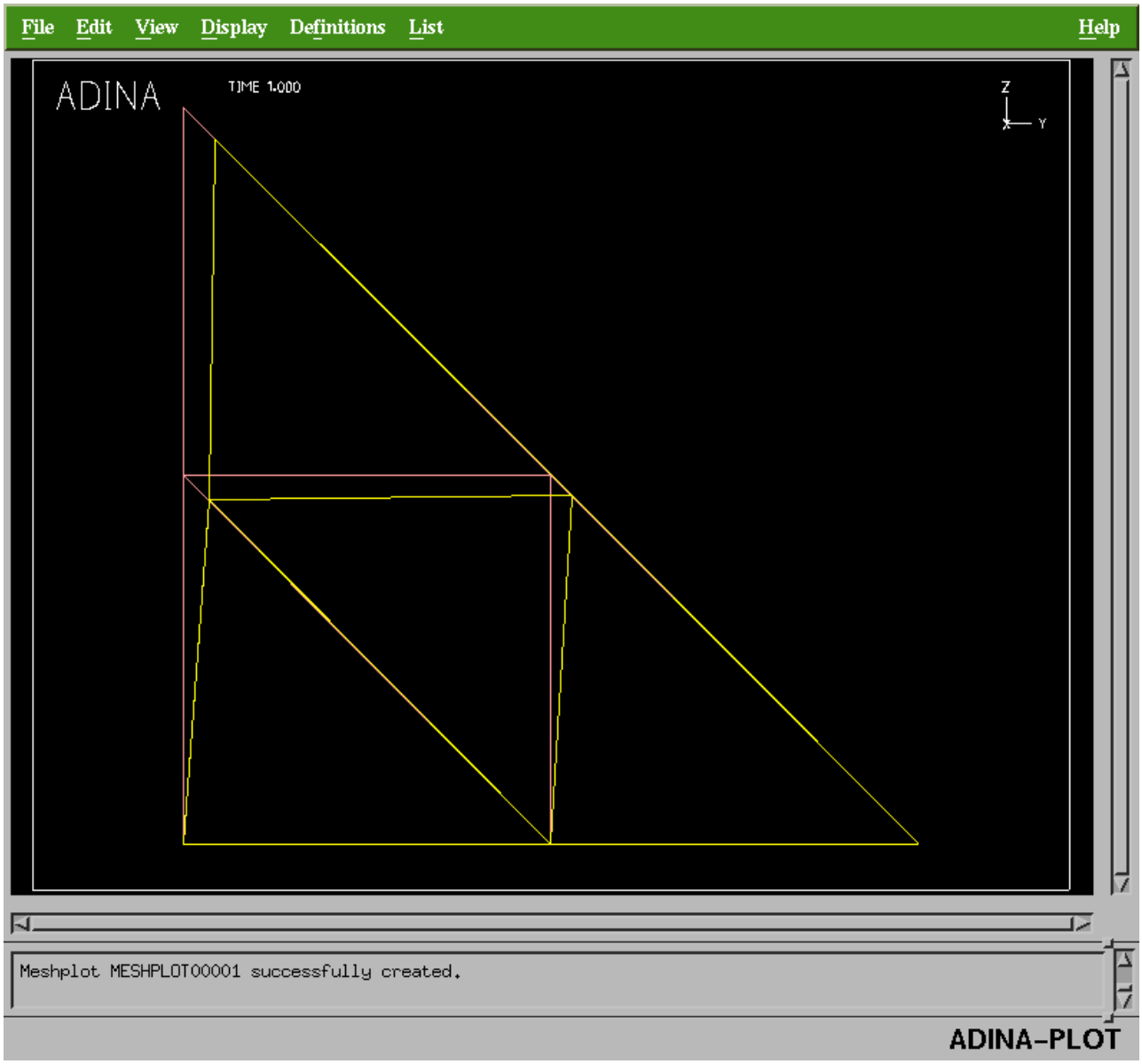


Figure 20

Listing the tip deflection

Choose List-Extreme Values->Zone. In the lower half of the dialog box, use the scroll bar to display the Variables box. For Variable 1, choose Y-Displacement from the right hand drop-down window list. Then press Apply. Displacement of the tip will be listed in the top window as shown in Figure 21.

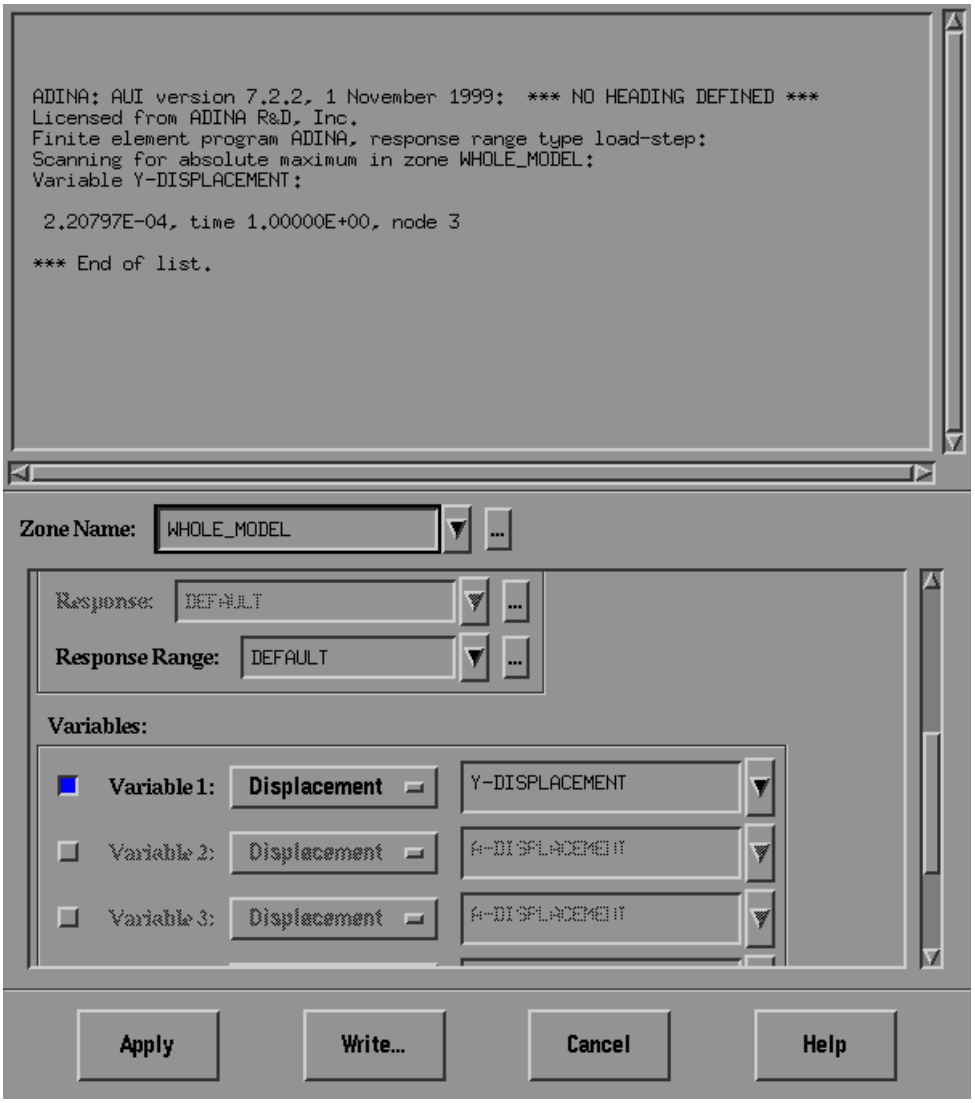


Figure 21