

Case Study 1: Piezoelectric Rectangular Plate

PROBLEM - 3D Rectangular Plate, k31 Mode, PZT4, 40mm x 6mm x 1mm

GOAL

Evaluate the operation of a piezoelectric rectangular plate having electrodes in the top and bottom large surfaces and polarized in the thickness mode. The operation is evaluated in the transversal direction, i.e. k31-mode. The size of the plate is 40mm x 6mm x H=1mm. The piezoelectric plate is made of PZT4 material.

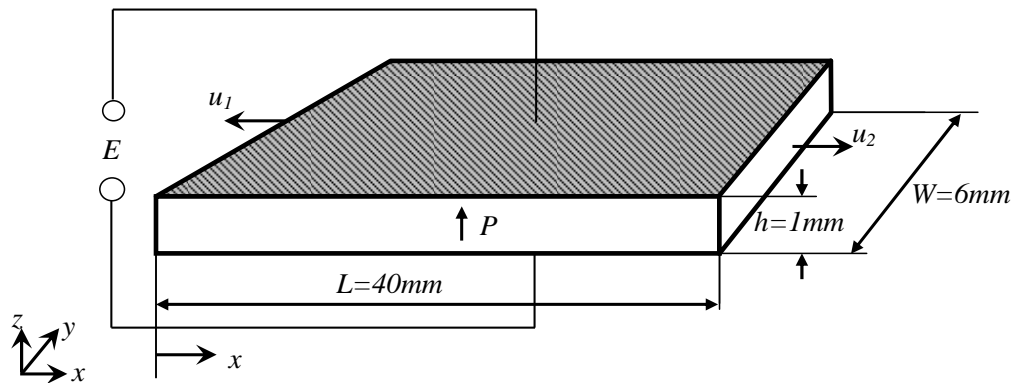


Figure C1.1. Piezoelectric Rectangular Plate.

$$L = 40 \text{ mm}$$

$$W = 6 \text{ mm}$$

$$H = 1 \text{ mm}$$

Material: PZT4

Top and Bottom Electrodes

Thickness Polarization

GEOMETRY/DRAWING

1. Start GiD

ATILA uses the GiD interface to create, modify, and setup simulations. This example will cover some GiD basics while introducing ATILA's powerful features.

Open GiD using the Desktop shortcut, you will see the work area, menus, and icons as shown in Figure C1.2. GiD/ATILA is ready to use when opened. The creation of file names etc occurs when the file is saved, anything not saved is discarded when GiD is closed.

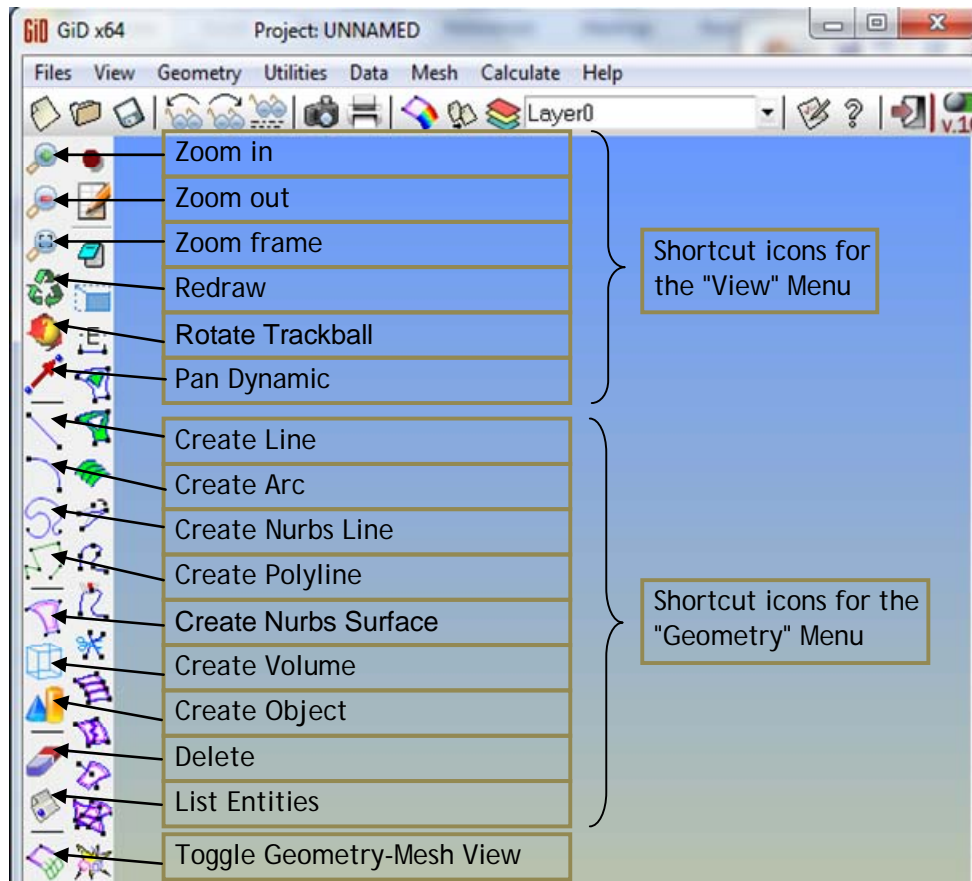


Figure C1.2. GiD work area with menus (top bar) and icons (vertical bars).

2. Create Surface

Alternative 1: Using the top drop menus

Geometry → **Create** → **Object** → **Rectangle** → *Enter first corner point* → **0,0** →
Enter second corner point → **40,6** → **Enter**

This example will be creating and simulating a Rectangular Plate Piezo. GiD works within a coordinate system, therefore, creating geometry requires entering coordinates. The example is a rectangle. Begin by clicking the top level menu **Geometry** → **Create** → **Object** → **Rectangle**. See Figure C1.3, left.

Note: The creation of geometry using the object menu simplifies the process, for example the creation of the rectangle in the example will require the input of 2 corners rather than the creation of 4 lines.

After selecting the Rectangle option, the GiD Command line will display the plane of creation and the request to *Enter first corner point* of the rectangle (see Figure C1.3, right-top). Selection of any point is possible, however, be aware that everything builds from here so choose wisely. For this example, the coordinates **(0,0)** are used. Then, press **Enter**.

GiD will request the coordinates for the second corner, *Enter second corner point*. For this example enter **(40,6)**, press **Enter**. This step can be seen in Figure C1.3, right-bottom.

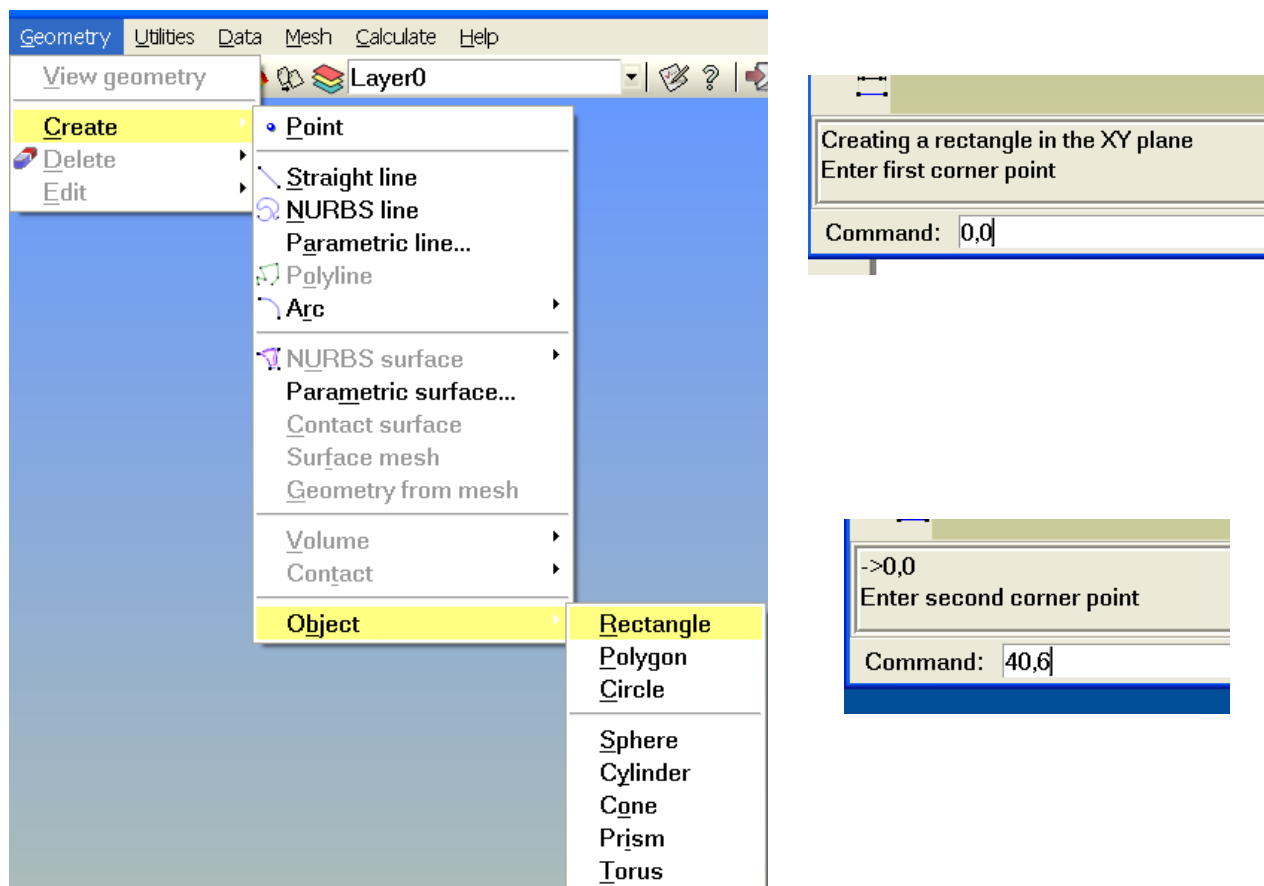




Figure C1.3. Create Rectangle using menu and command entry line.

Alternative 2: Using the vertical Shortcut Icons Bar

To create the rectangle using the icons, Click , the object icon drop menu will appear, click the rectangle icon , (the same requests appear in the Command line window) enter first corner coordinates (0,0), press enter, enter second corner coordinates (40,6), press enter. This sequence is shown in Figure C1.4.

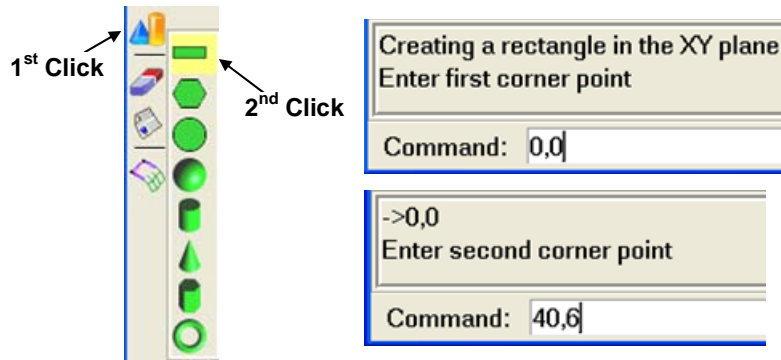


Figure C1.4. Create Rectangle using Icons and command entry line.

The Rectangle has been created as seen in Figure C1.5. Note that the object lines are shown in blue. GiD automatically creates surfaces, shown in magenta.

Note: GiD's default color scheme is Point = Black, Line = Blue, Surface = Magenta, Volume = Cyan

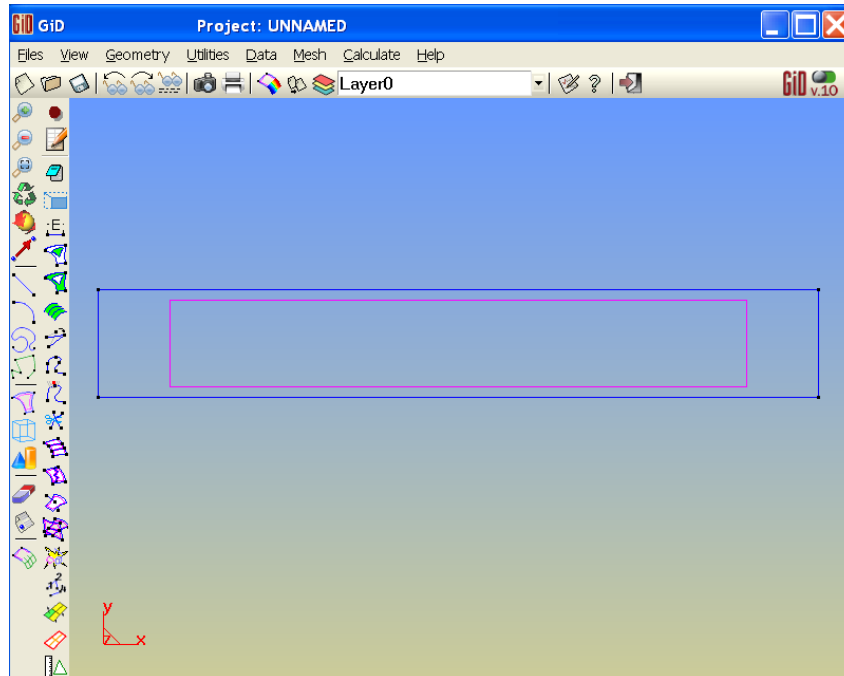


Figure C1.5. Rectangle created. The surface is in magenta color while the rectangle lines are in blue.

Note: GiD creates the rectangle and asks for additional inputs to continue creating additional rectangles. Press **ESC** (or the middle mouse button) to quit object creation.

3. Create Volumes Using Copy

Utilities → Copy → Surfaces → Translate → Enter first point → 0,0,0 → Enter second point → 0,0,1.0 → Duplicate Entities → Do Extrude Volumes → Select → Finish

There are several ways to create the volume. In this example, the **Copy** function is used. Copy allows the extrusion of the bottom surface and creates the full volume of the plate. An alternative option, not used in this example, is to create the individual lines at the corners and add the lines to complete the 3D model. This option also requires the creation of the surfaces and volumes.

Click **Utilities** → **Copy**. The Copy menu appears as illustrated in Figure C1.6, right.

In the **Copy** menu, select Entities type: **Surfaces** and **Translation** for the type of Transformation. For this example, the First Point coordinate is the default value, i.e. (0,0,0), the Second Point is the thickness of the plate, in this example 1mm, i.e. the Second Point should be (0,0,1.0). Check the box to **Do Extrude Volumes**, **Maintain Layers**, and **Multiple copies** is 1. This will copy the rectangle surface in the positive Z direction and extrude the sides at the same time. See the selections in Figure C1.6. center and right panels.

Note: Description of Copy Menu Options:

Duplicate entities: If this option is not set and after the copy operation an entity occupies the same position as an existing one that does not belong to a frozen layer, both entities are converted into one.

Extrude: This option can be set to either lines, surfaces or volumes. When a movement is selected, the copy is made and lines connecting the old and new points are created. These lines will either be straight lines or arcs depending on the movement type. If extrude surface is chosen, NURBS surfaces connecting old and new lines will also be created. If Volumes is chosen, the volume contained between old and new surfaces is also created. This option is not allowed when copying volumes.

Maintain layers: If this option is not set the new entities created will be placed in the layer to use; otherwise, the new entities are copied to the same layers as their originals.

Multiple copies: By selecting this option and giving the number of repetitions, the selected operation is performed this number of times. This option is not available for mirror.

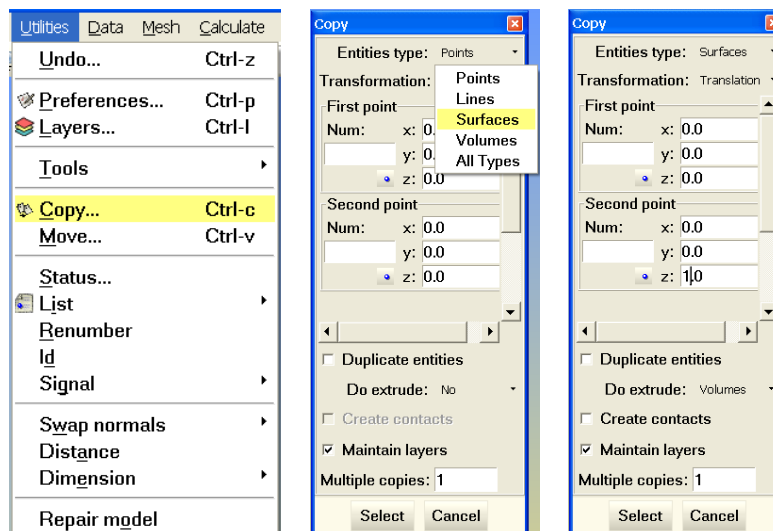


Figure C1.6. Settings for the Copy Control Panel

Click **Select** in the **Copy** menu. The cursor will change to a large square with a small square inside (☐). Select the surface of the rectangle. The color will change from magenta to red indicating that GiD has accepted the selection. Click **Finish** to complete the creation of the volume. GiD translated the surface and extruded the sides of the rectangle to create a solid model and the corresponding surfaces. The volume created is shown in cyan in Figure C1.7.

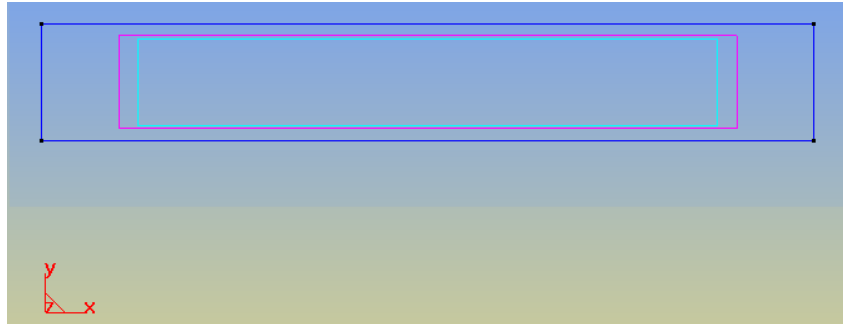


Figure C1.7. Volume Created. The volume is shown in cyan color.

4. Learning How To Use The "View" Functions: Pan, Zoom, Rotate

There are several options for changing the view of the model with GiD/ATILA; following is a quick look at each option. The process is generally the same for each of the viewing functions. This example will use the **Rotate** function.

Alternative 1: Using the top drop menus

To rotate the rectangle for clearer visibility click: **View → Rotate → Trackball**. Then, right click the mouse button and move the mouse to rotate. The **Pan** and **Zoom** functions are also located on this menu as shown in Figure C1.8.

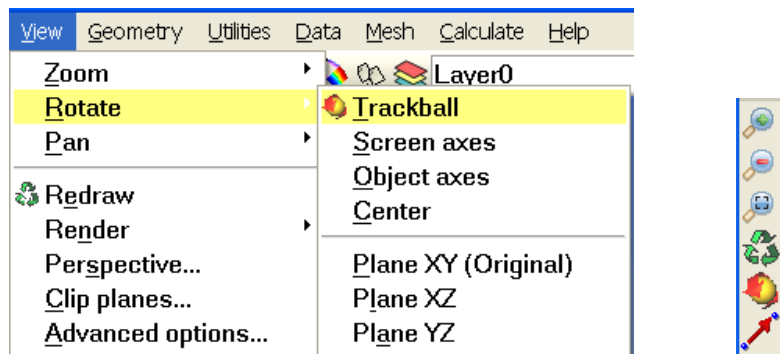







Figure C1.8. Changing the View using the Menu (left) or using the icon bar (right)

Alternative 2: Using the Icons

To rotate using the icons vertical menu shown in Figure C1.8-right, Click , Press and Hold Right Mouse Button, move the mouse. Press **ESC** or the Middle Mouse Button to quit. **Other View icons:** 

Zoom In,  Zoom Out,  Frame (Resize to fit screen),  Dynamic Pan

Alternative 3: Shift Key option

Press and hold the **Shift Key**, press the Right mouse button and move the mouse within the graphics window. The model will rotate. Release the Mouse Button to quit rotate.

Note: The **Shift Key** option includes the **Pan** function using the Left mouse button and the **Zoom** function using the Scroll Wheel (If your mouse is equipped with a scroll wheel).

To return to the 2D view: Click **View → Rotate→ Plane XY (Original)**.

Tip: Many functions are available within the graphics window by clicking the Left Mouse button, a menu appears and allows you to select from the functions available. See the menu in Figure C1.9.

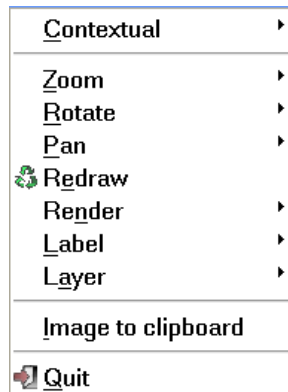



Figure C1.9. Onscreen menu

To better view the model just created fit it to the screen then rotate it.

To fit to the screen: Click **View → Zoom → Frame** or Click the "Frame" icon () to center the view in the graphics window. See the results in Figure C.1.10-left. Holding the **Shift key** and pressing the **Right** mouse button while moving the mouse will rotate the model to better see the surfaces and volume created. Release the **Shift key** to quit rotation. The result is shown in Figure C1.10-right.

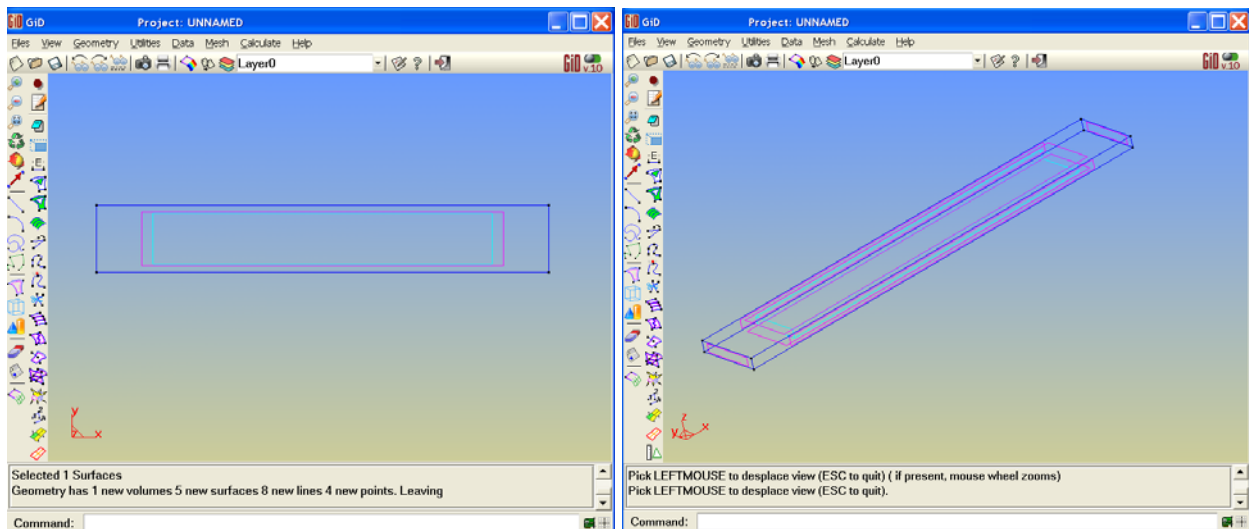


Figure C1.10. Resizing to fit the Window

MATERIALS AND CONDITIONS

5. Materials Assignment

The 3D model has been created, the next step is to assign the materials and the boundary conditions necessary for the simulation. The Menu and Icon options will be explained.

Begin by activating Atila. Click **Data → Problem type → Atila → 3.0.21**.

Note: Other compatible versions of ATILA may be present based, for this example we will use 3.0.21 as illustrated in Figure C.1.11.

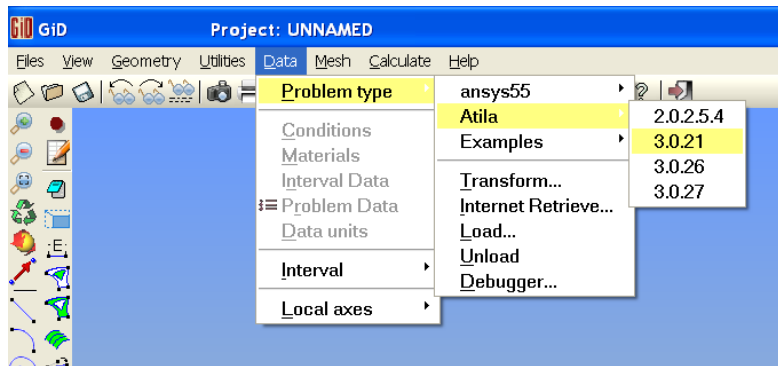


Figure C1.11. Starting ATILA 3.0.21 by selecting it under Problem type.

The header now shows the version of ATILA being used and the Menu and Icons associated with ATILA will be loaded and available for use, as shown in Figure C1.12.

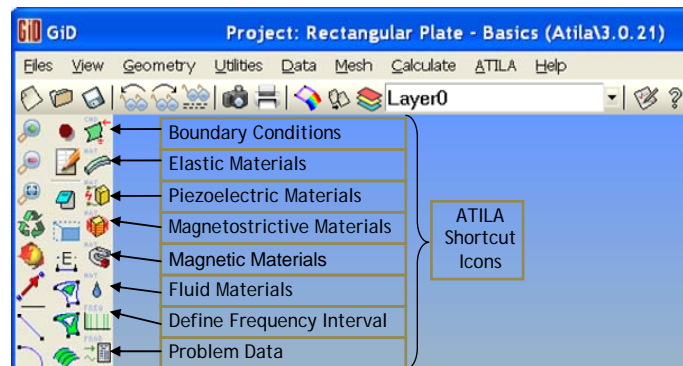


Figure C1.12. ATILA 3.0.21 menus and icons

Several options are available to assign material to the model:

Alternative 1: Using the top drop menus.

Click **Data → Materials → Piezoelectric**. The material control panel will appear as illustrated in Figure C1.13. The PZT4 material will be used for this example. Select **PZT4 → Assign → Volumes → Select volumes**. Select the volume of the rectangle, it will change from cyan to red indicating GiD has accepted the selection. Then Click **Finish** or **Esc** to finish the material assignment, as shown in Figure C1.13..

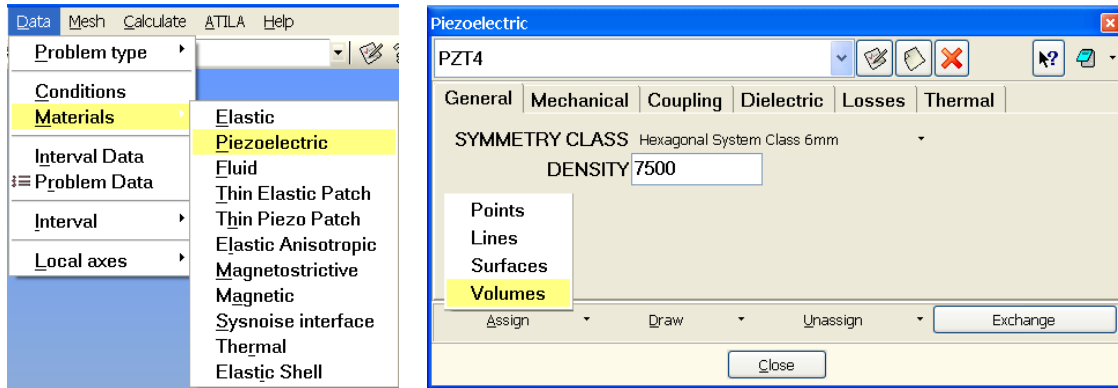


Figure C1.13. Material Selection

Alternative 2: Using the Icons Vertical Bar

Click () → PZT4 → Assign by volumes → Select volumes → Finish or Esc. See Figure C1.14.

Note: Using the icon presents the assignment panel directly.

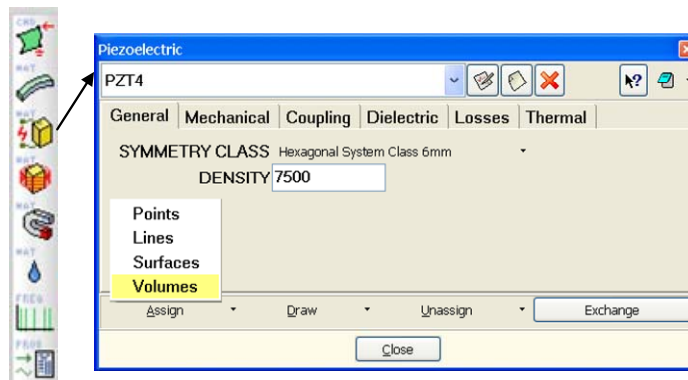


Figure C1.14. Piezoelectric Material Selection Panel

Note: The steps for both options are the same from this point forward.

Select the volume of the rectangle, it will change from cyan to red indicating GiD has accepted the selection. Click "**Finish**" or "**ESC**". to complete the operation. Click **Draw** → **PZT4** to confirm the material has been added. Click "**Finish**" or "**ESC**" to quit. This is illustrated in Figure C1.15.

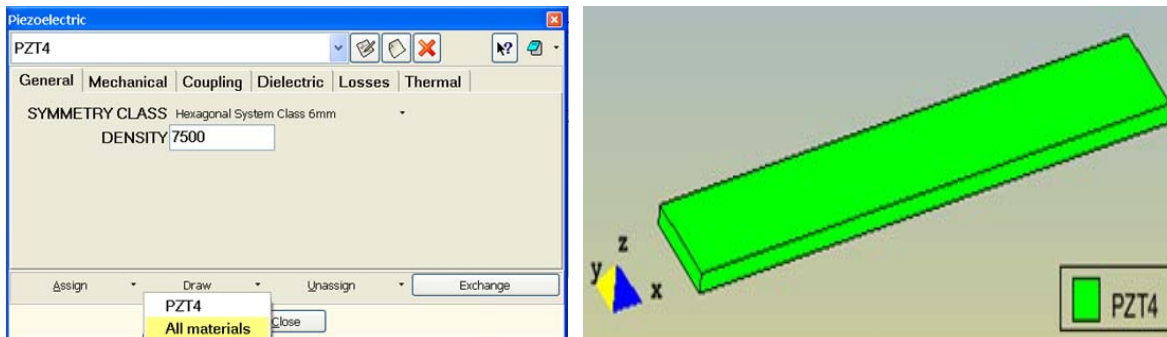


Figure C1.15. Piezoelectric Material Confirmation

6. Boundary Conditions Assignment

6.1 LOADING THE BOUNDARY CONDITION MENU

Assignment of Boundary conditions is necessary for the simulation. These are the constraints and parameters that will govern how ATILA processes the simulation. This example will cover using the menus and/or the icons.

Alternative 1: Using the top drop menus.

Click **Data** → **Conditions** as shown in Figure C1.16. The conditions panel appears as seen in Figure C1.16.

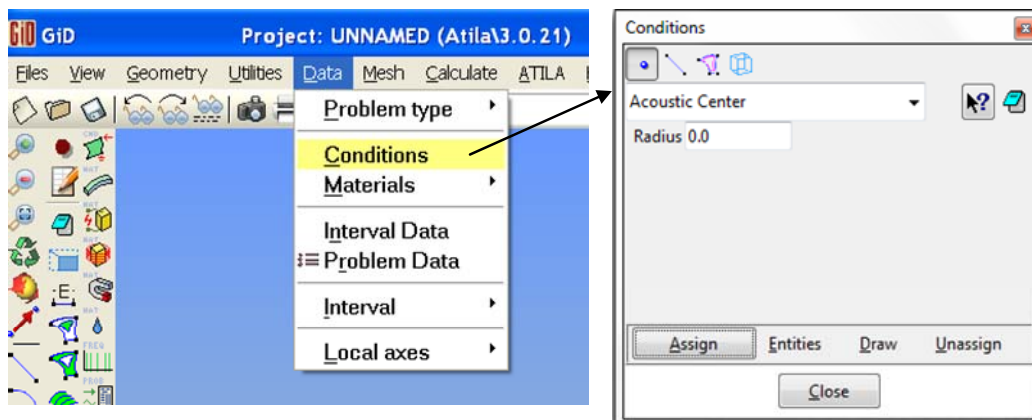



Figure C1.16. Boundary Conditions Assignment accessed through the top menu.

Alternative 2: Using the ATILA Icons Vertical Bar.

Click () on the ATILA vertical icon bar. The conditions panel appears as seen in Figure C1.17. All of the necessary boundary conditions for the simulation will be set using this Control Panel.

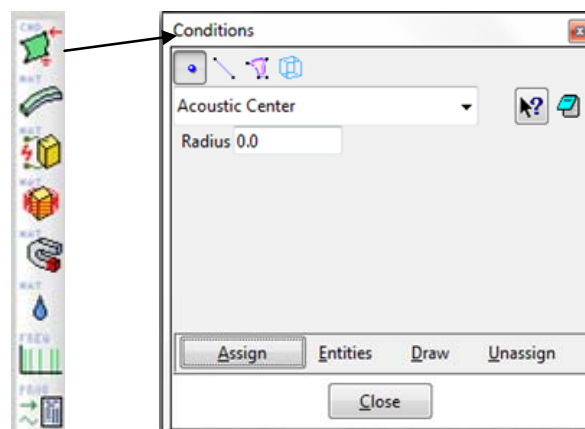



Figure C1.17. Boundary Conditions Assignment accessed through the vertical ATILA icons bar.

6.2 ASSIGN ELECTRICAL POTENTIAL BOUNDARY CONDITIONS

We need to assign electrical potential boundary condition to the top and bottom **surfaces** of the bar where the electrical electrodes are located. Two boundary conditions are required. Ground electrode and Forced (or positive electrode).

Assign Surface Forced Potential, i.e. Positive (+)

Click  → **Electrical Potential** → **Forced** → Enter **1.0** → **Assign** → Select **Surface** → **Finish** or **Esc**.

Select Surfaces () icon on the top panel and Electrical Potential for the condition. Enter (**1.0**) in the space provided for the value of the **Forced** potential. This value is in Volts. After setting the voltage, click **Assign** and select the top surface of the plate. It will turn red indicating acceptance of the selection. Click **Finish** to complete. See Figure C1.18.

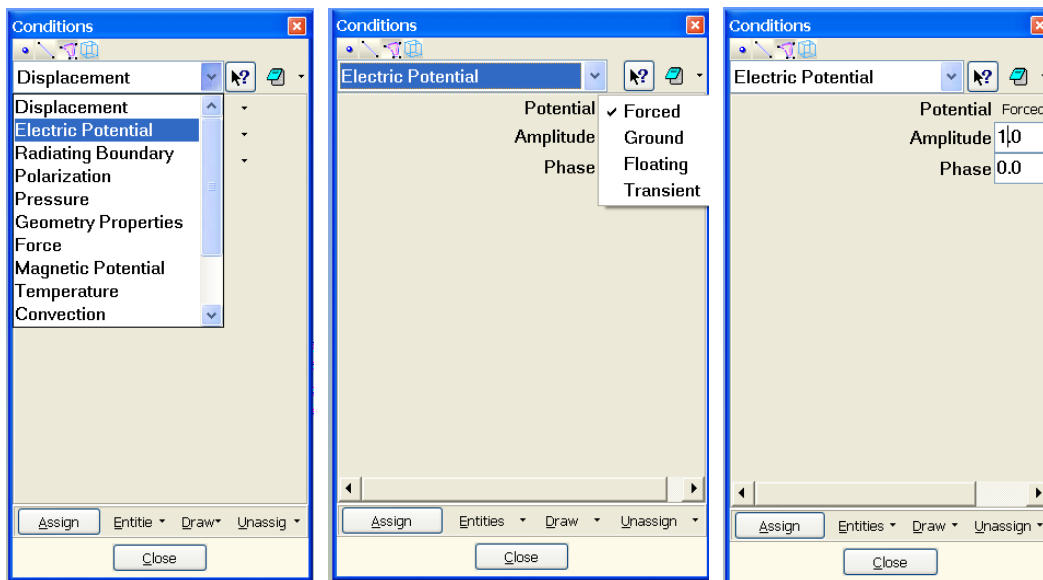



Figure C1.18. Boundary Conditions Control Panel

Assign Ground Potential (-)

Click  → **Electrical Potential** → **Ground** → Enter **0.0** → **Assign** → Select **Surface** → Select the bottom surface of the bar → **Finish** or **Esc**.

Repeat the same process to assign the Ground electrode. Click **Ground** → **Assign**, select the bottom surface of the plate. The surface will turn red to signify acceptance of the selection. Click **Finish** to complete. Confirm the Electrical Potential assignment by clicking **Draw** → **Colors**. See the result in Figure C1.19. Click **Finish** to quit.

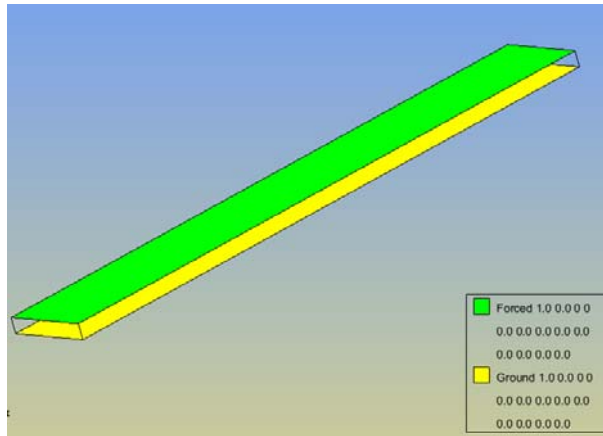



Figure C1.19. Boundary Conditions Control Panel

6.3 ASSIGN POLARIZATION BOUNDARY CONDITIONS

It is necessary to assign the volume polarization of the piezoelectric material. This will determine the direction of the movement of the material in response to the applied voltage.

Click the volume icon () on the top of the control panel, then select **Polarization** from the list. In this example the default, **Cartesian**, is selected. Click **Define**. The Local Axis naming panel will appear as shown in Figure C1.20.

It is necessary to name the “Local Axis system”, for this example use P1. (*Note: this can be any name that makes sense or best describes this axis for the particular application.*) Enter P1 and press **Enter** or click **OK**.

The orientation dialog window will appear giving two options for defining the axis location and direction. This example will use the X and Angle option. See Figure C1.20 right.

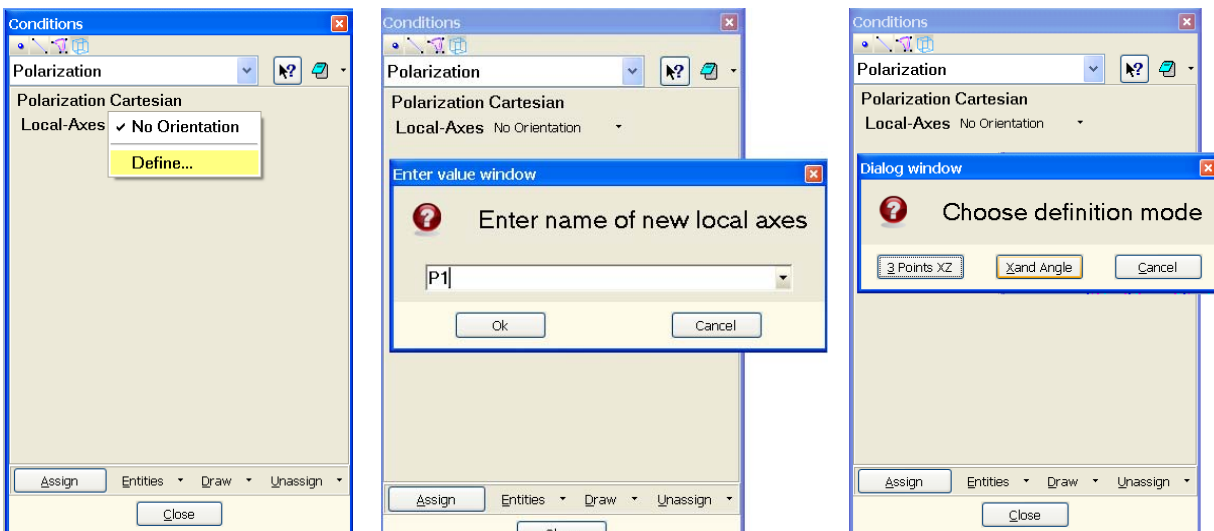


Figure C1.20. Boundary Conditions Control Panel

Note: ATILA polarization follows the “Z” Local direction.

There are two options to define the local axis:

Alternative 1: Graphical Option

Click the **X and Angle** button and the cursor will change to a plus (+). You will need to select 2 points along the “X” axis to set the “X” direction. Right click in the graphics window and click **Contextual → Join Ctrl-A** as shown in Figure C1.21. The cursor will change to a square with the smaller square inside (◻). This change can also be accomplished by holding the **Control** key and pressing **A**.

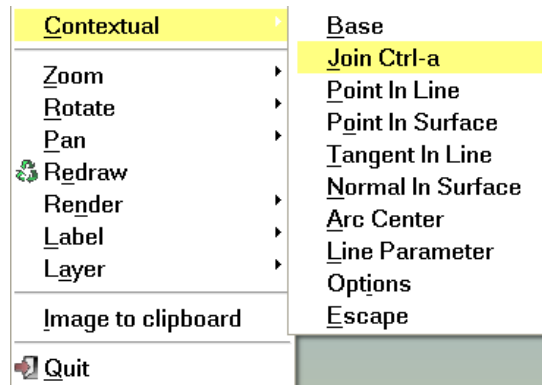


Figure C1.21. Boundary Conditions Control Panel

The Join function allows you to select points already created and snap to them. Select the first corner, reselect **Join Ctrl-A**, select the top corner. See Figure C1.22 left side.

A moveable axis will appear, move the mouse to orient it in the desired direction, for this example match the graphic “Z” axis. The angle will appear next to the cursor, when it is at “0” click the **Right Mouse Button** to select. See illustration in Figure C1.22 right.

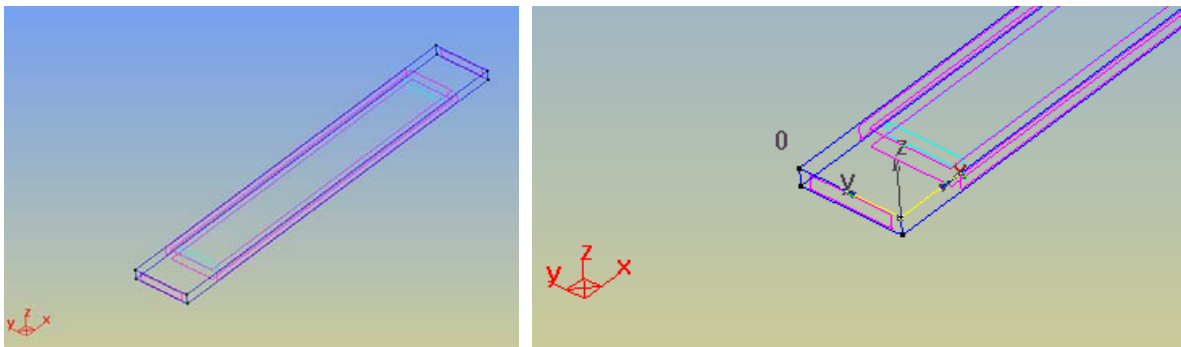


Figure C1.22. Creating Local Axis graphical method

Alternative 2: Coordinate Entry Option

Click the **X and Angle** button and the cursor will change to a plus (+). Move the cursor to the **Command Line**, enter the coordinates for the **Center** of the local axis, **Enter**. Enter the coordinates for a point in the positive **X** direction, **Enter**. Enter the **Angle** to define the **Y** axis (This is relative to the graphic axis). These steps are illustrated below in Figure C1.23.

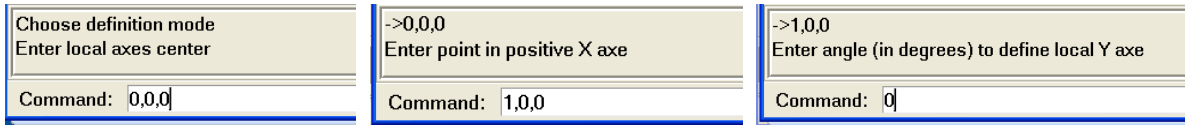


Figure C1.23. Creating Local Axis Coordinate Entry method

To confirm the Boundary Conditions click **Draw** → **All conditions** → **Include local axis** → **Finish** or **ESC** to quit. The process and results are shown in Figure C1.24.

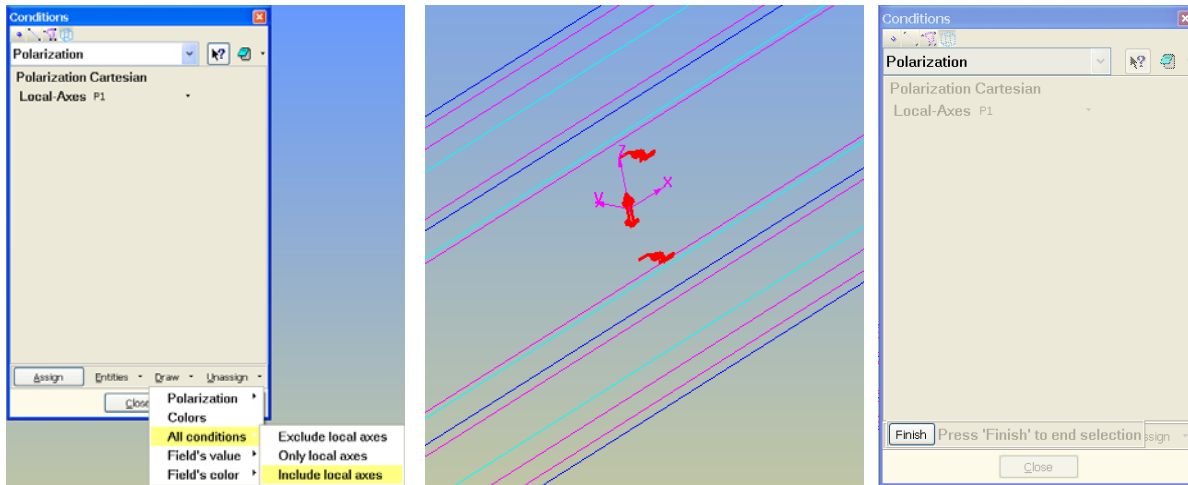


Figure C1.24. Confirmation of Polarization

Note: All of the conditions will be shown in red. See center panel of Figure C1.24.

Select **Finish** or **ESC** to complete the operation.

MESH

7. Meshing the model

The geometry is created, the material assigned, and the conditions assigned, the next step is to mesh the model. Meshing divides the part into geometric components that the software uses to calculate the stress, deformation, etc to create the outputs used for analysis. There are several options to generate the mesh. Here we provide two examples. More details can be found in GiD help.

7.1 UNSTRUCTURED MESH

Alternative 1: Using the top drop menus.

Click **Mesh** → **Generate Mesh** → **Mesh generation panel** → **OK** See Figure C1.25.

Alternative 2: Using the Icons.

Click () → **Mesh generation panel** → **OK** See Figure C1.25 right side.

Alternative 3: Using Hot keys.

Press and hold **Control** + press **G** → **Mesh generation panel** → **OK**. See Figure C1.25 right side.

Note: The Mesh Generation popup asks for a size, this is only applied to elements that have not been assigned a size. GiD offers two default possibilities:

- The default size automatically calculated by the program to define a coarse mesh.
- The last size given by the user in a previous meshing.

You can choose one of these or enter a new one.

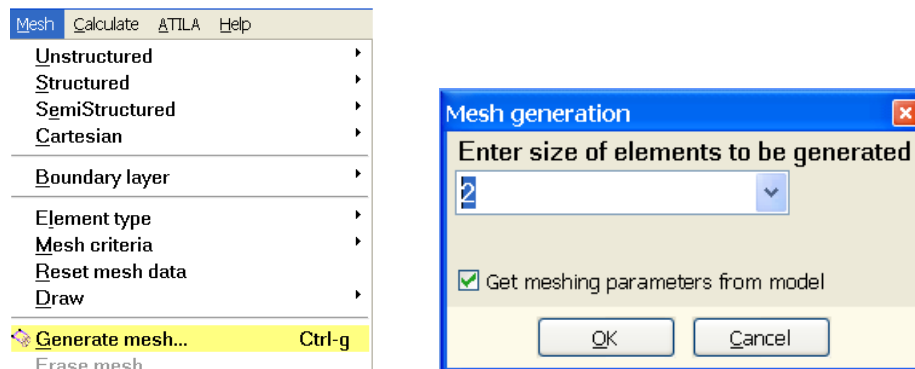


Figure C1.25. Creating a Random Mesh

The Dialog panel appears indicating the mesh generation is complete. Click **View Mesh** to view the mesh created as illustrated in Figure C1.26.

Note: In this example the mesh is uniform, however, depending on the shape of the geometry this is not always the case.

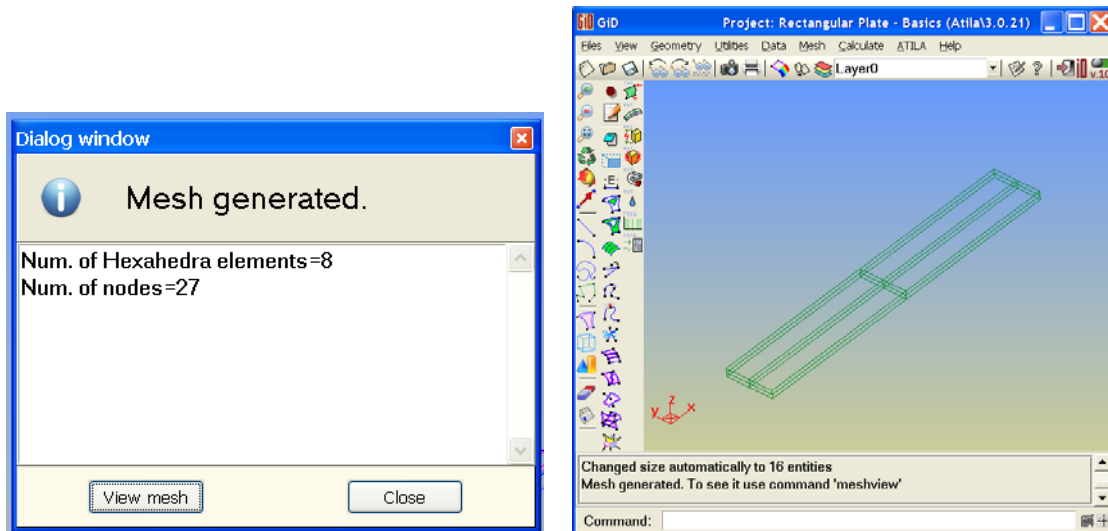


Figure C1.26. Viewing the Mesh

7.2 STRUCTURED MESH

The meshing of the geometry is the same for a structured mesh as for the Unstructured or Random. the difference is in the settings applied before generating the mesh.

First of all, erase the old mesh.

Click **Mesh → Erase Mesh → OK → OK**. See Figure C1.27.

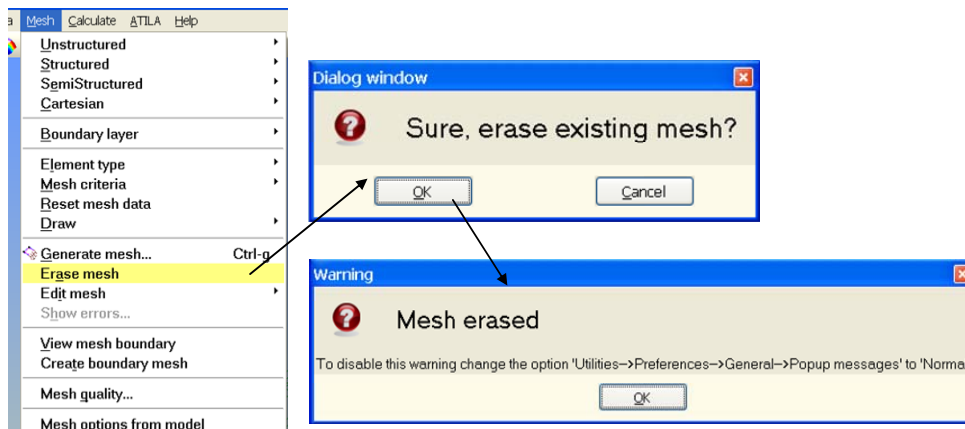


Figure C1.27. Erasing the Mesh

To setup the structured mesh Click **Mesh → Structured → Volumes → Assign number of cells**. This is illustrated in Figure C1.28.

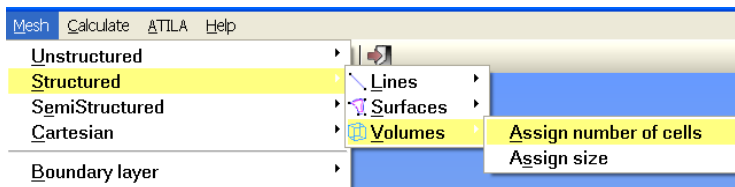


Figure C1.28. Creating a Structured Mesh

The cursor will change to a square with the small square in the center (☐), select the volume you want to mesh. Press **ESC**. (Note: This process will be followed for each volume to be meshed if there is more than one. This example only has one volume to be meshed.) The control panel shown will appear. Enter the number of divisions for a line. Click **Assign**. See Figure C.1.29.

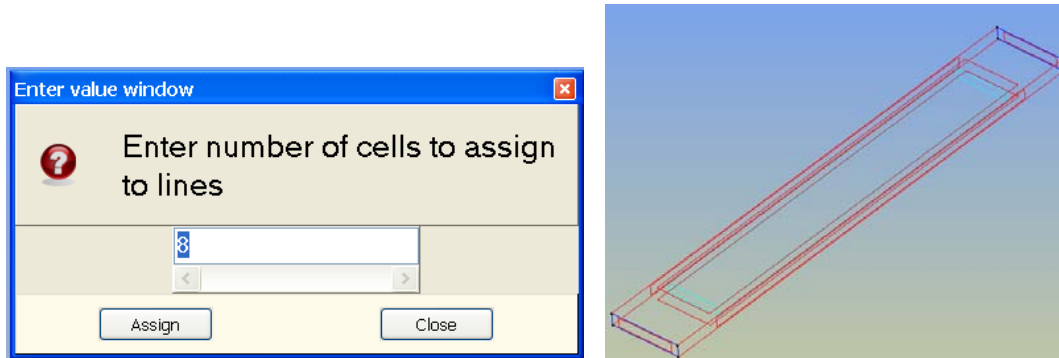


Figure C1.29. Assigning number of divisions

The cursor will change again (☐) to allow the selection of the lines for that number of divisions. When the lines are selected they will turn red indicating acceptance.

Note: In a structured mesh the lines opposite one another are selected when any one is chosen to maintain symmetry in the mesh. See Figure C1.29 right panel.

When finished selecting the lines for that number of divisions press **ESC**. The Control Panel will reappear with the option to enter a new value, the selection process is the same as used in the previous step. After all of the lines have been assigned a number of divisions press **ESC** twice.

Press the key board **CTRL + G**, the Mesh Generation control panel below will appear. Click **Get meshing parameters from model** and click **OK**. The mesh will be generated, select **View mesh** to see the mesh as illustrated in Figure C1.30.

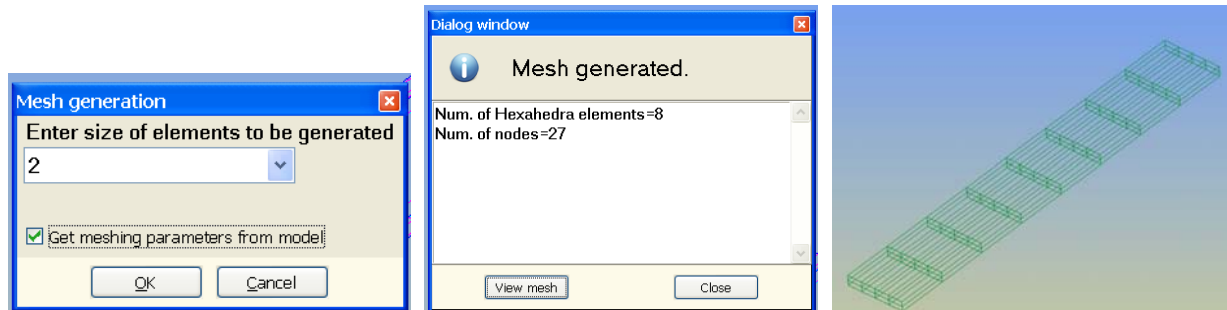


Figure C1.30. Confirm Mesh Element Size

CALCULATE

8. Calculations

The model has been created, the material and the boundary conditions assigned, and the geometry Meshed, the next step is to assign the Problem Data. The Problem Data are the rules for the simulation, they determine the type of simulation, the number of steps, etc that are to be run to generate the data for analysis.

8.1 HARMONIC CALCULATION

STEP 1: ELECT HARMONIC ANALYSIS FROM PROBLEM DATA

Alternative 1: Using the top drop menus.

Click **Data** → **Problem Data**, the Control Panel will appear.

Click **Geometry** → **3D**.

Click **Analysis** → **Harmonic**. Click **Include Losses** → **Write strain** → **Write stress** → **Accept**. The lower left corner of the control panel will flash the message “*Problem Data modified*”. See these steps in Figure C1.31.

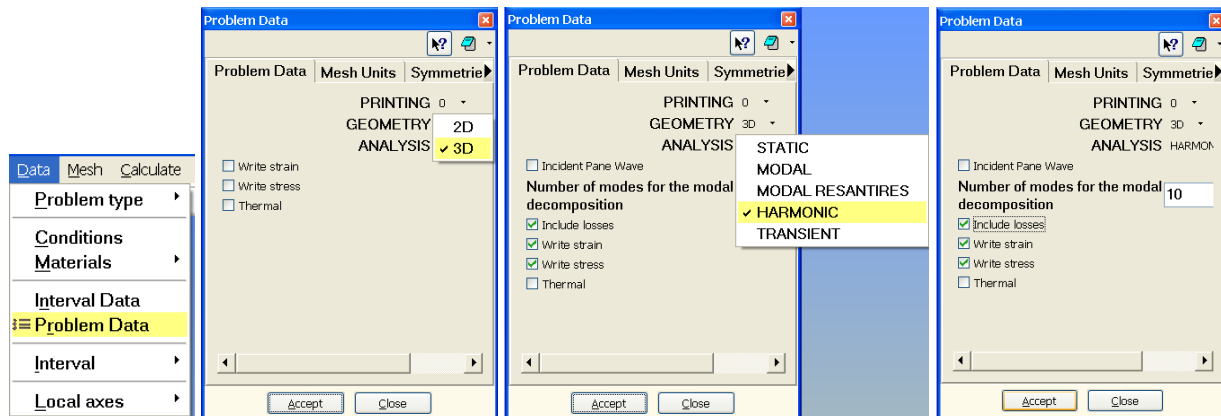



Figure C1.31. Assign Problem Data

Alternative 2: Using the Icons.

Click () → the Control Panel will appear then follow the steps listed above in Alternative 1, Figure C1.31.

STEP 2: ENTER THE FREQUENCY RANGE TO PERFORM THE HARMONIC ANALYSIS

Alternative 1: Using the top drop menus.

Click **Data** → **Interval Data**. The Interval Data Control Panel will appear.

Click **TYPE** → **LINEAR DISTRIBUTION**.

Enter the Minimum and Maximum frequencies and the number of points to be tested. (Note: More points tested will give a smoother Admittance/Impedance graph and more accurate results, however the

calculation time will increase correspondingly.) Click **Accept** when finished, the lower left corner of the control panel will flash the message *Interval Data modified*. Sequence is shown in Figure C1.32.

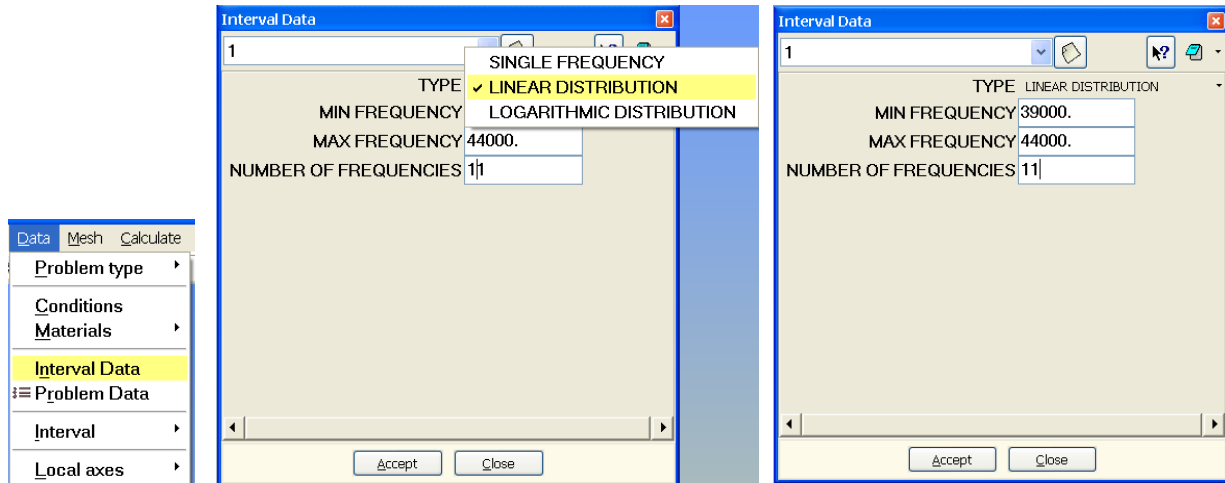



Figure C1.32. Assign Interval Data

Alternative 2: Using the Icons.

Click (). The Interval Data Control Panel will appear then follow the steps listed above in Alternative 1, Figure C1.32.

At this point, all of the necessary information to perform an harmonic calculation has been entered.

STEP 3: START THE CALCULATION

Click **Calculate** → **Calculate** as shown in Figure C1.33 left panel. The calculation dialog box will appear briefly letting you know the calculation has started. See Figure C1.33 center panel.

Note: The progress can be monitored by clicking **Calculate**, then **View Process info....** This will open the window displaying the calculation processes as they are proceeding. When the calculations are complete the **Process Info** panel will appear. Click **Post-process** to open the postprocessor and view the results of the simulation. See Figure C1.33 right panel.

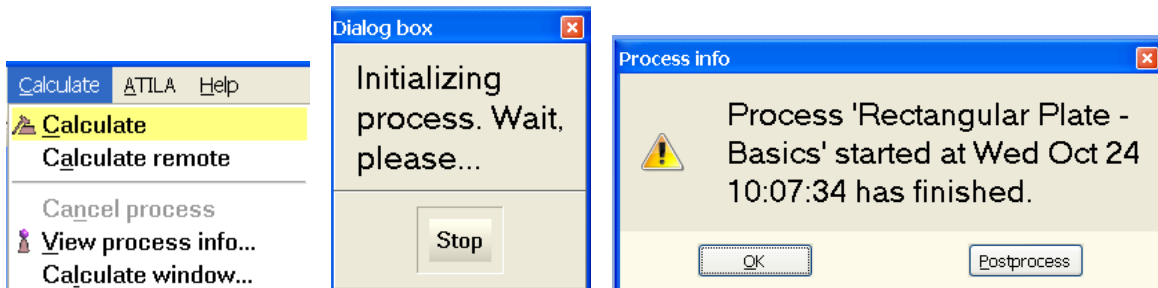


Figure C1.33. Calculation Panels

RESULTS

9. Post-Process/View Results

The post-process mode will appear similar to the preprocess mode with the addition of several new commands and icons. See window in Figure C1.34.

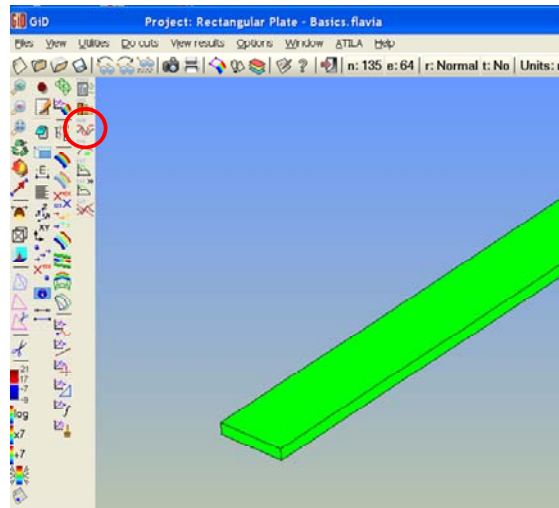




Figure C1.34. Post-Processor Window

TIP: To enter to the postprocessor mode after exiting or for a calculation that has been run previously, open the file and click (). This will load the data and open the postprocessor window.

9.1 ADMITTANCE GRAPH

GID/ATILA will generate a graph of the Admittance/Impedance of the Rectangular Plate Piezo. To view the graph, click the **Magnitude and phase of the admittance** icon () along the left side of the graphics window as indicated in Figure C1.34. in the red circle. The graph control panel will appear as shown in Figure C1.35, click **View Graph**.

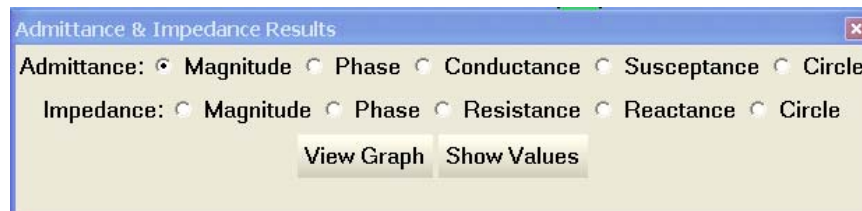


Figure C1.35. Admittance/Impedance Control Panel

The Admittance magnitude graph will appear, see Figure C1.36. left panel. Other options available include Phase, Conductance, Susceptance and Circle admittance and associated values for the Impedance as illustrated in Figure C1.35.

The values of the points graphed can be displayed by clicking Show Values. They will be listed in a column format as seen in Figure C1.36 right panel. These can be exported to an Excel file for later analysis. That will be discussed in a later example.

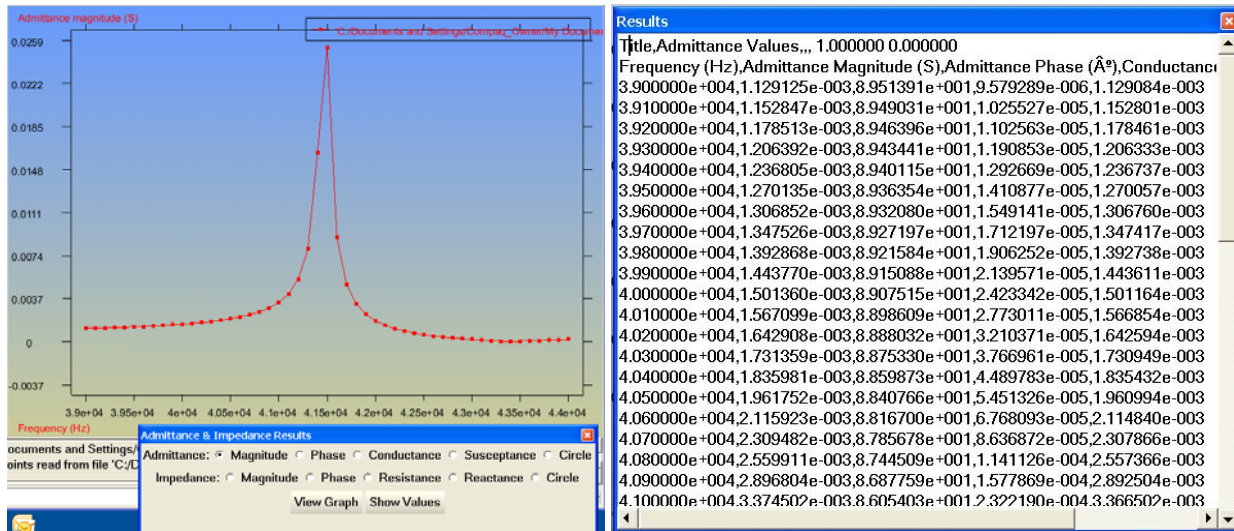


Figure C1.36. Admittance Graph

Details

To get the exact coordinates of a point on the graph right click in the graphics window. The contextual menu will appear, select **Label** → **Select** → **Res.** As seen in Figure C1.xx. left panel. The cursor will change to a square with the solid center (Ⓚ), select the point you want to display the values of, then press **ESC** (More than one point may be selected). The values will be displayed as seen in Figure C1.37. right panel

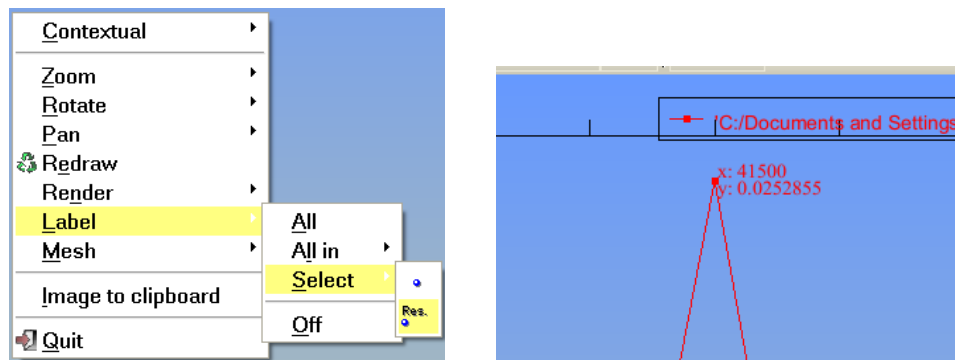


Figure C1.37. Labeling a Point

A clearer picture of the movement of the piezo can be seen by transforming the “Y” scale of the graph to a log scale. To make this change click **Options** → **Graphs** → **Y axis** → **Set Logarithmic Scale**. The confirmation window will appear, click **Yes**. See Figure C1.38. for this process.

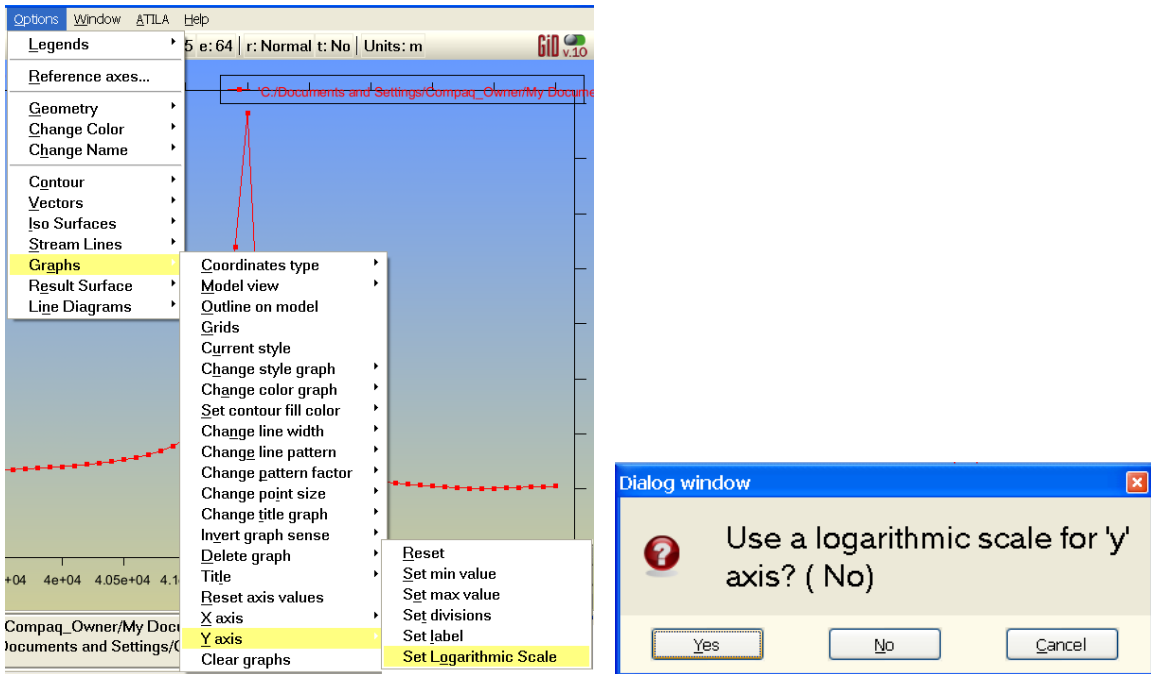


Figure C1.38. Converting Y axis to Log10 Scale

The graph now displays the result with a log scale. The resonant peaks can be seen more clearly as shown in Figure C1.39.

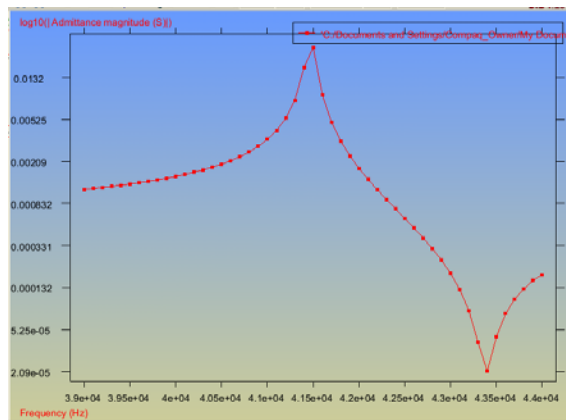



Figure C1.39. Admittance Graph

To remove the graphs select the clear graphs icon (). See the red circle in Figure C1.40.

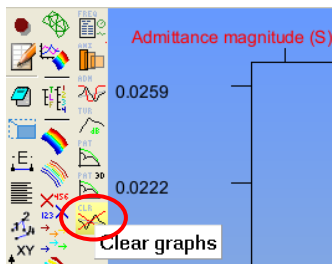


Figure C1.40. Clear Graphs

9.2 HARMONIC ANIMATION

ATILA produces an animation to visually display several parameters and their changes as the material is cycled. This example will look at displacement, though others results are available.

Click **View Results** → **Default Analysis/Step** → **Harmonic-Magnitude** → **Frequency** to display as shown in Figure C1.41. Select the resonance frequency of 41500 Hz as seen in the admittance graph in Figure C1.37.

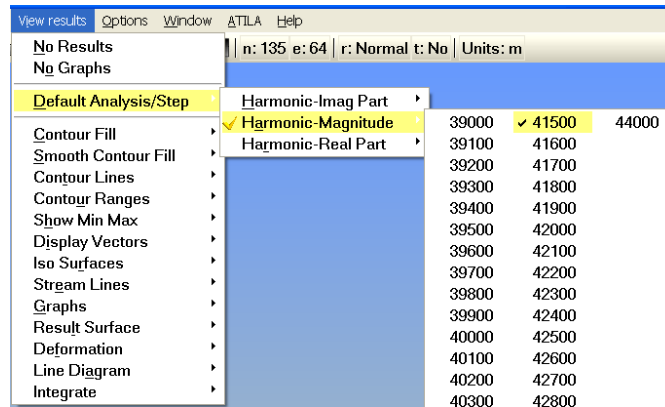


Figure C1.41. Frequency Selection for Animation

Click **ATILA** → **Animate** or Click (), as shown in Figure C1.42.

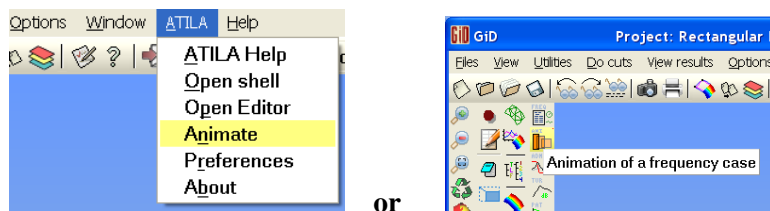


Figure C1.42. ATILA Animation Start

The Animation panel will appear, See Figure C1.43, click **OK**.

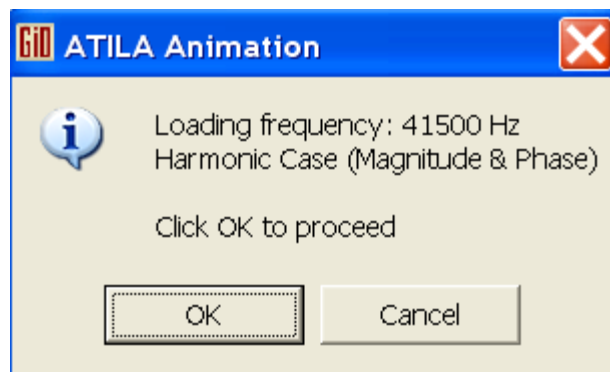



Figure C1.43. ATILA Animation Start

The **ATILA Animate** control panel will appear. Click **Results View** or **Deformation** and the particular result or deformation magnitude to view with the simulation. If no selection is made and the run arrow () is clicked the Warning Panel appears as shown in Figure C1.44.

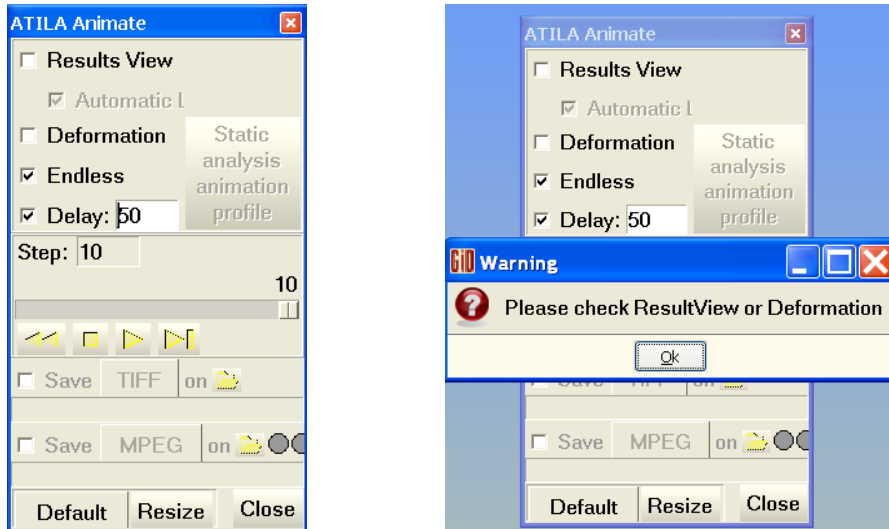



Figure C1.44. Atila Animation Control Panel

Click **View Results** → **Deformation** → **Displacement**. The selections can be done with the icons as well. Click the Deformation icon () then set the magnitude. This is done using the slider bar, see Figure C1.45, or the left and right arrows on the keyboard.

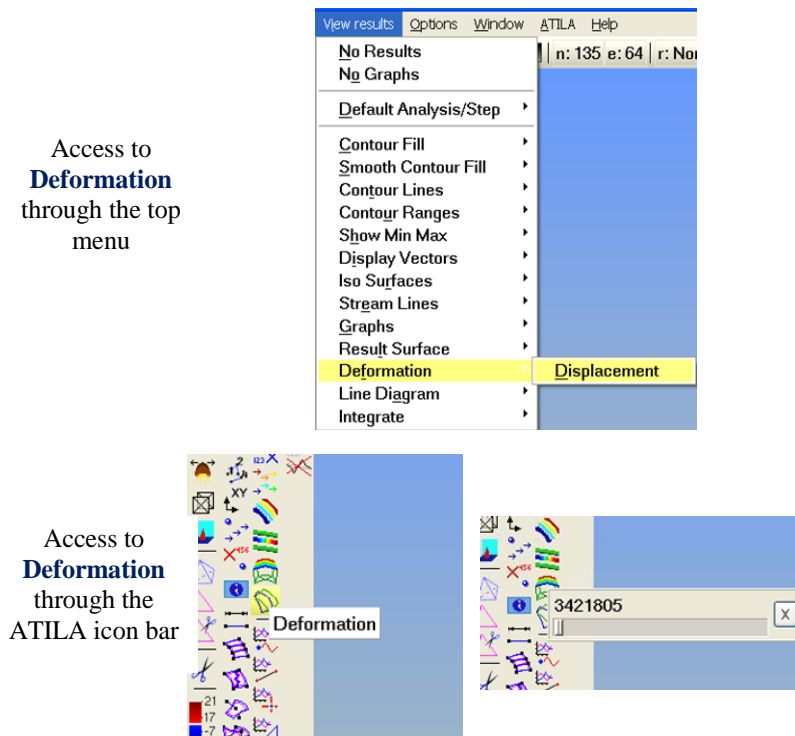






Figure C1.45. ATILA Animation Set-up

When the **Deformation** and/or **Results** have been selected the animation can be run. Click the **run arrow** () to begin. The animation can be paused by clicking the stop button () and stepped through the animation by clicking the step button () or moving the slider bar (). See Control Panel in Figure C1.44.

The start, minimum displacement, and maximum displacement are shown in Figure C1.46 for reference.

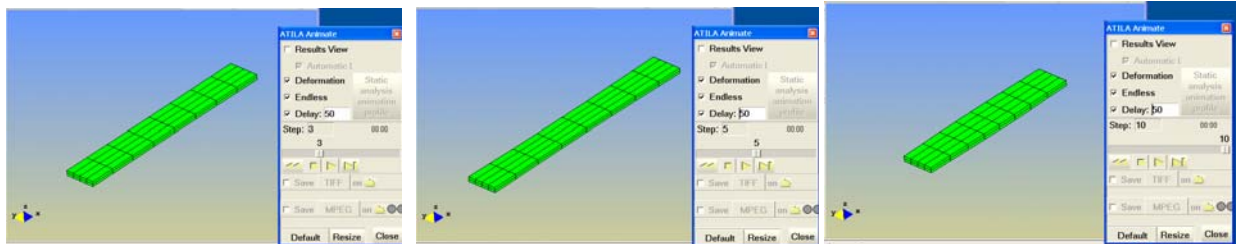


Figure C1.46. ATILA Animation Running

DISPLACEMENT

Various results can viewed during the animation, such as *Displacement*.

Click **View Results** → **Contour Fill** → **Displacement** → **X-displacement**. The **Contour Fill** icon () can be used in place of the menu, see Figure C1.47.

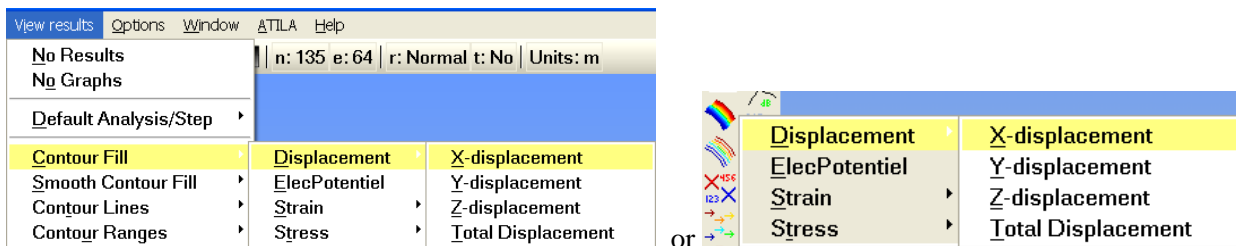


Figure C1.47. Adding Displacement to Animation

The model will change color to show the movement of the material at various sections of the plate. This view is dynamic and changes as the animation is run. See Figure C1.48.

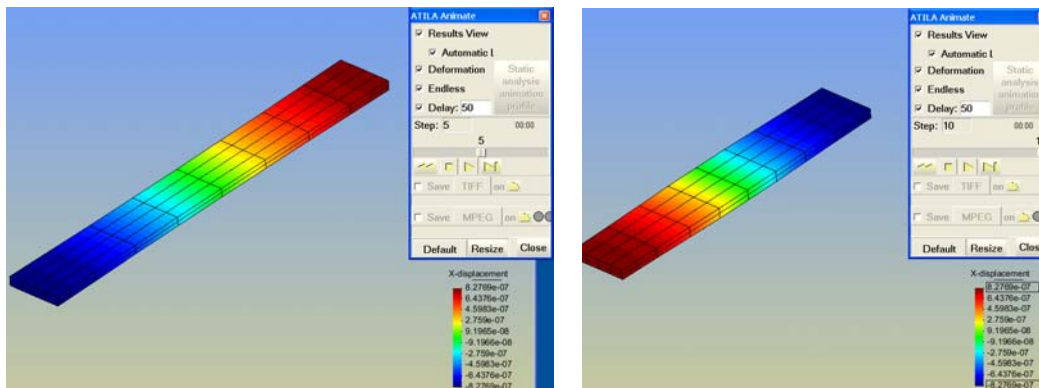





Figure C1.48. Running Animation with the Displacement

To exit the animation mode click **stop** () → **close** () on the animation panel as shown in Figure C1.49. left panel

Click **View Results** → **No Results** to get back to the normal view as shown in Figure C1.51. center panel

To finish exiting the animation mode click () as shown in Figure C1.51. right panel

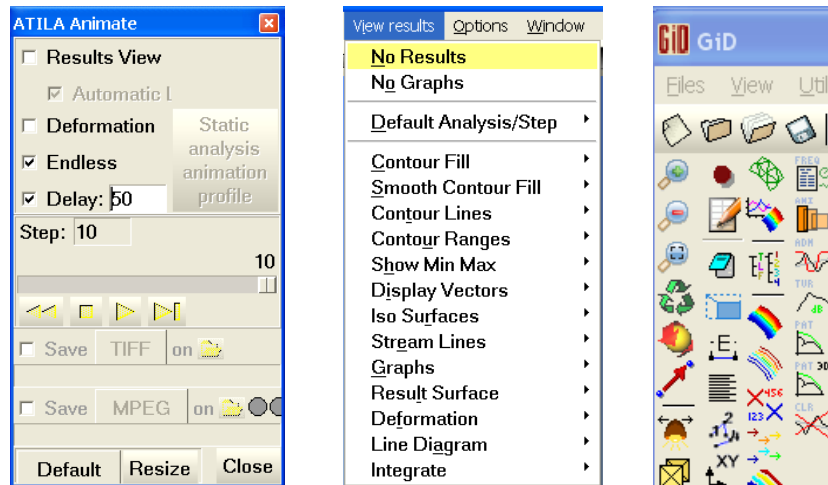


Figure C1.49. Exiting Animation Mode

9.3 ADDITIONAL VISUALIZATIONS

POINT EVOLUTION GRAPH

The point evolution graph will show the changes at a given point selected on the part throughout the steps of the simulation, for this example the X-displacement will be graphed.

Alternative 1: Using the top drop menus.

Click **View results** → **Graphs** → **Point evolution** → **Displacement** → **X-displacement** as shown in Figure C1.55.

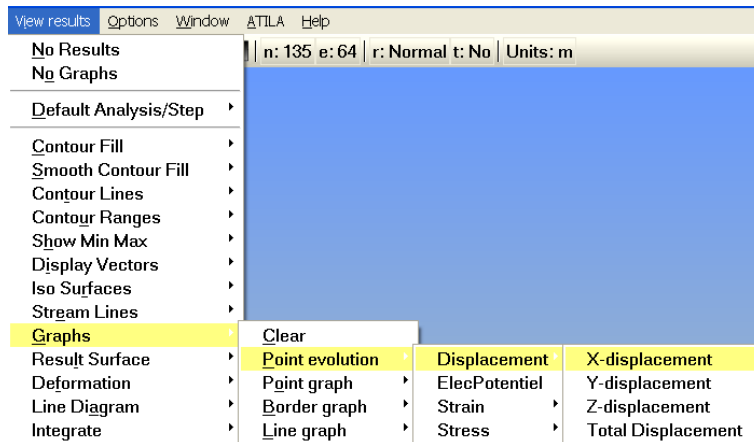


Figure C1.55. Point evolution graph creation using Menu option

Alternative 2: Using the Icons.

Click () → **Displacement** → **X-displacement** as shown in Figure C1.56.

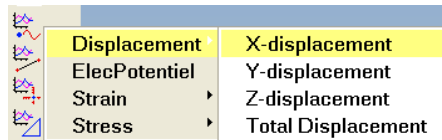



Figure C1.56. Point evolution graph creation using icon option

The next step is to select the point that will be graphed. Upon selection of the **X-Displacement** the cursor will change to (+) for the selection. Click **CTRL + A** in the keyboard to change the cursor to () and select the center point at one end of the rectangle, see the red circle in Figure C1.57. for clarity.

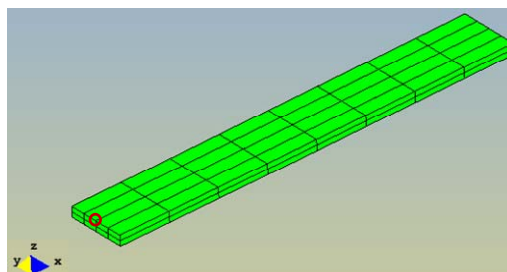


Figure C1.57. Point Selection

Upon the selection of the point the graph will appear as shown in Figure C1.58 bottom. On Figure C1.58 top, the admittance graph is also included. The maximum displacement is obtained at the maximum of the admittance. In this example, the maximum occurs at 41500 Hz.

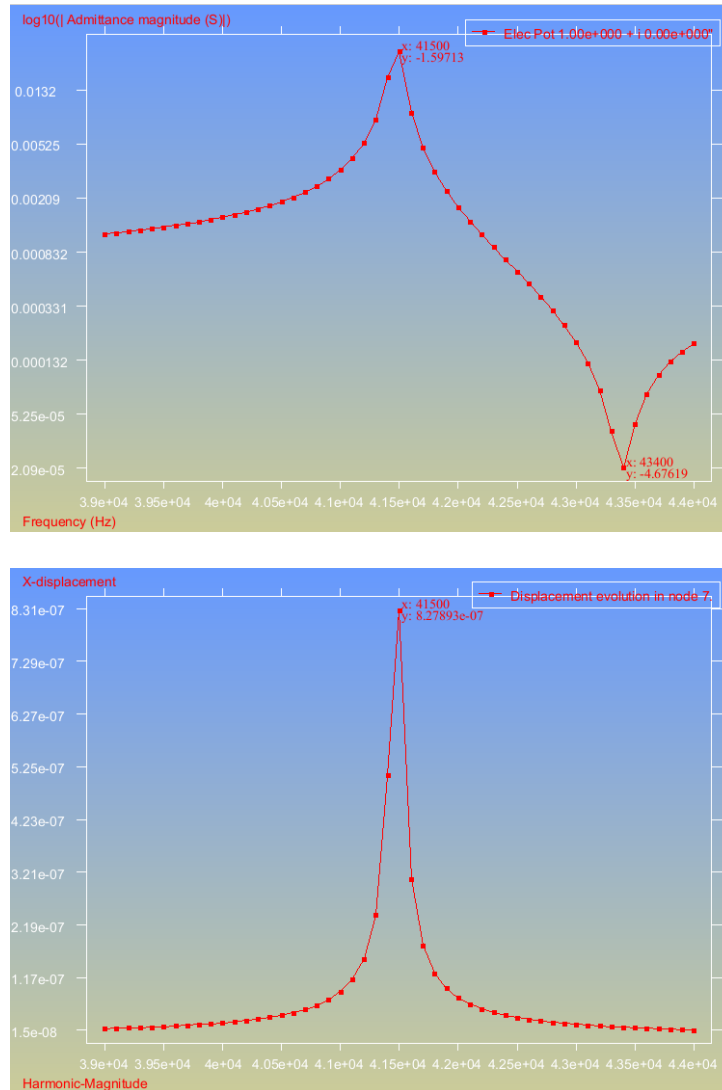



Figure C1.58. Point Evolution Graph

Any of the graphs in this group can be cleared using the following steps: Click **Options** → **Graphs** → **Clear graphs**. Alternatively, you can use the vertical icon bar: Click () as shown in Figure C1.59.

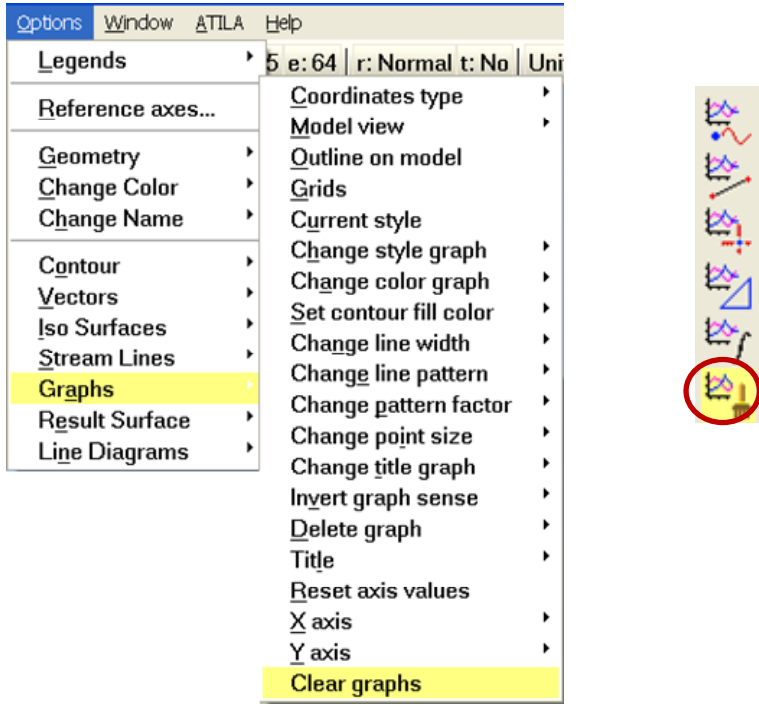


Figure C1.59. Clearing Graphs through the top drop menu (left) and through the vertical icon bar (right).

POINT LINE GRAPH

The point line graph will show the changes along a line drawn on the part throughout the steps of the simulation, for this example the X-displacement will be graphed.

First of all, select the type of Results to be show. In general, there are three options: Real Part, Imaginary Part and Magnitude. The Magnitude is the absolute value of a specific result, for instance the displacement. All the results will shown in the positive Y-axis. The Real value provides positive and negative visualization. For this example, we will use Real Part. This is shown in Figure C1.60.

Also, the selection of the frequency were the point line graph is needed is selected. In this example, we are selecting 41500Hz, which is the resonance frequency.

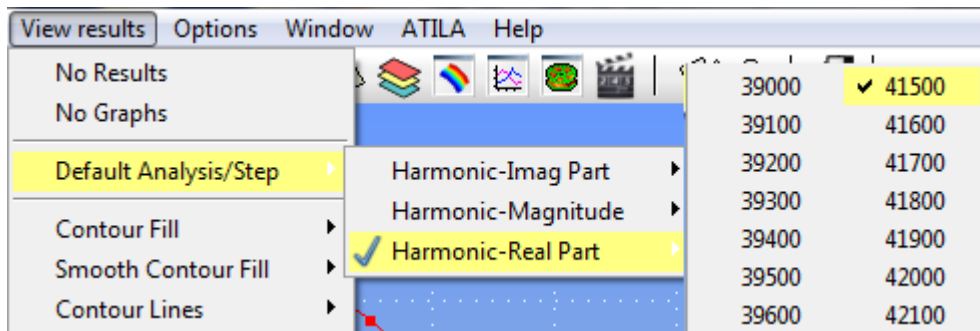


Figure C1.60. Selection of Harmonic-Real Part result for the point line graph.

Alternative 1: Using the top drop menus.

Click **View results** → **Graphs** → **Line graph** → **Displacement** → **X-displacement** as shown in Figure C1.61.

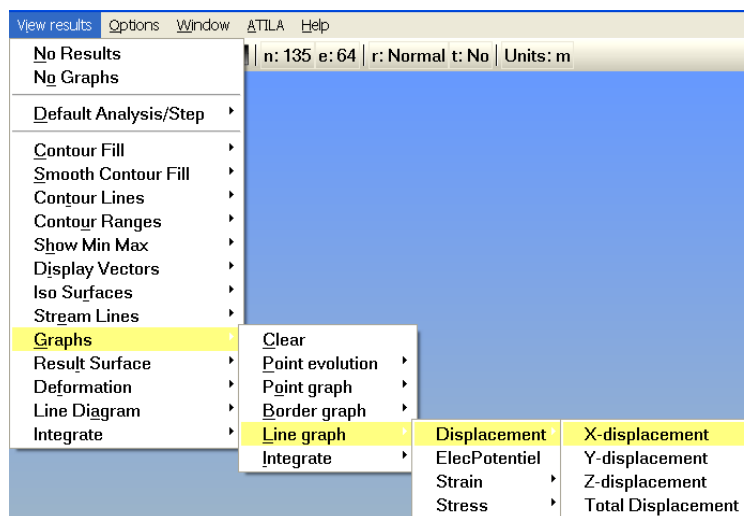


Figure C1.61. Line graph creation using menu option

Alternative 2: Using the Icons.

Click () → **Displacement** → **X-displacement** as shown in Figure C1.62.

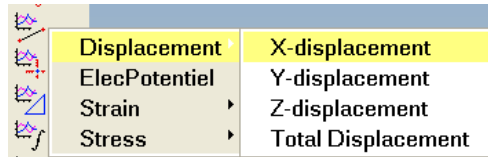


Figure C1.62. Line graph creation using icon option

The next step is to select the two points that will define the line on the part to be graphed. Upon selection of the **X-Displacement** the cursor will change to (+) for the selection. Click **Control + A** to change the cursor to (□) and select the center point at one end of the rectangle then select the center point at the other end of the rectangle, a line will appear until the selection is made, see Figure C1.63. for clarity.

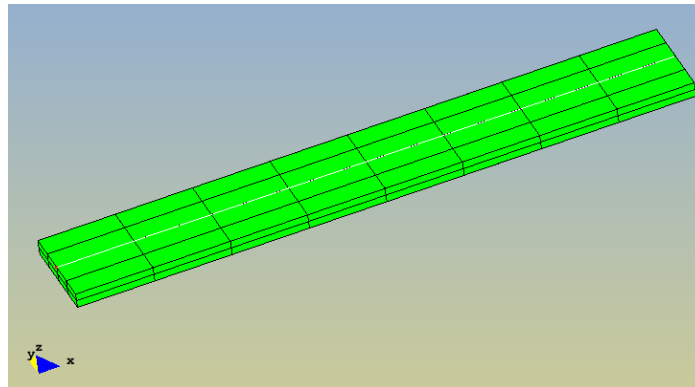


Figure C1.63. Line placement on the part

Upon the selection of the second point the line will be defined and the graph will appear as shown in Figure C1.64.

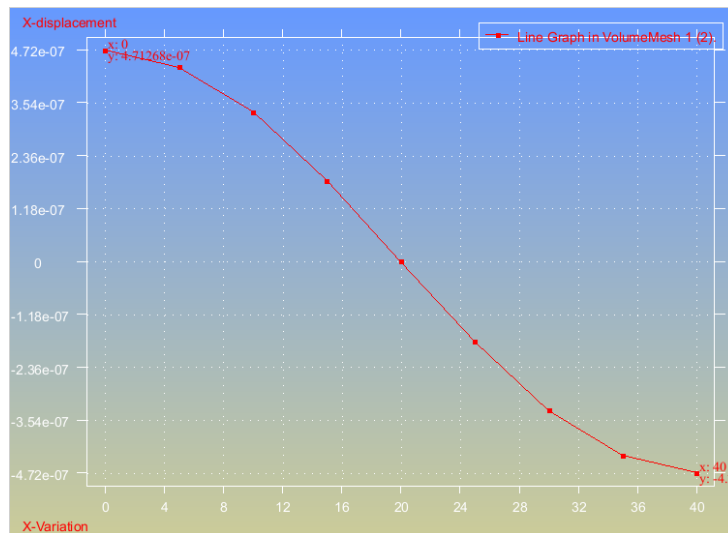



Figure C1.64. Line Graph representing the x-deformation across the length of the rectangular bar.

SAVING DATA

10. Exiting Post processor and saving the file

To exit the post processor mode click (), to confirm this action click **OK** as shown in Figure C1.61.

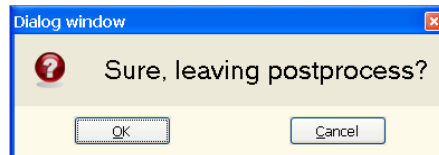



Figure C1.61. Exiting Post Processor

To save the project which includes the graphics, conditions, parameters, and all of the simulations Click () or **Files** → **Save** as shown in Figure C1.62.

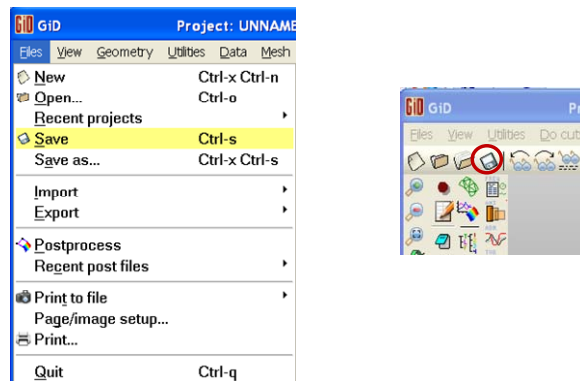


Figure C1.62. Saving a Project

The Save Project panel; will appear. Enter the desired name for the file and click **Save**. Be sure to set the desired destination for the file to save to, the is the same as any file in a windows enviroment. Figure C1.65.

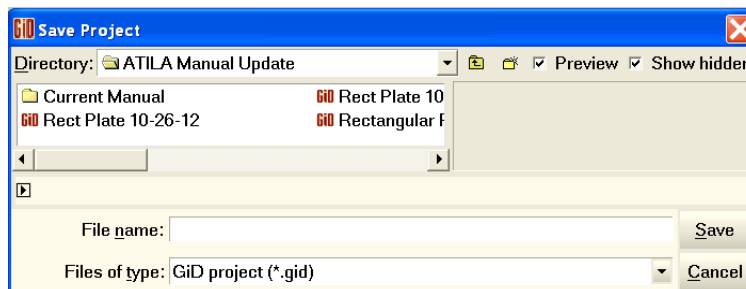


Figure C1.65. Naming the File and Selecting a Destination

Other useful data that can be obtained from the simulation.

1. The Displacement through a given frequency.
2. The displacement of a given point through the range of frequencies.
3. The Electrical potential

These will be discussed in future examples.