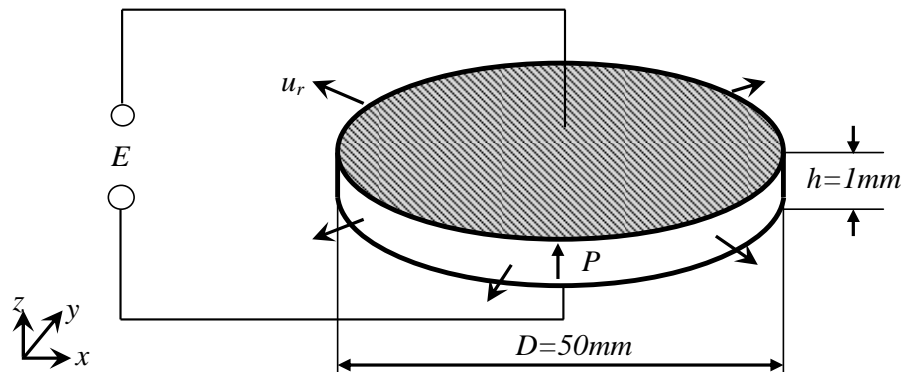


## Case Study 2: Piezoelectric Circular Plate

**PROBLEM - 3D Circular Plate, kp Mode, PZT4, D=50mm x h=1mm**

### GOAL

Evaluate the operation of a piezoelectric circular plate having electrodes in the top and bottom surfaces and polarized in the thickness mode. The operation is evaluated in the transversal (Radial) direction, i.e. kp mode. The size of the circular plate is 50mm diameter x 1mm thickness. The piezoelectric plate is made of PZT4 material.



**Figure C2.1.** Menu sequence for creating a circle

**D = 50 mm**

**H = 1 mm**

**Material: PZT4**

**Top and Bottom Electrodes**

**Thickness Polarization**

# GEOMETRY/DRAWING

## 1. Start GiD

Start GiD using the desktop icon.

*Note:* This example will incorporate both the menu and icon options to begin and transition to the most straightforward method as the example progresses. Later examples will outline the process and allow the user to select the method they are most comfortable with. All new processes will explain both options when introduced.

## 2. Create the 3D Model

### 2.1 Create the circle/surface

Alternative 1: Using the Drop Down menus.

**Geometry → Create → Object → Circle → Enter center point → 0,0 → Enter → Enter Normal → Select then OK → Enter Radius → 25 → Enter**

This sequence is illustrated in Figure C2.2.

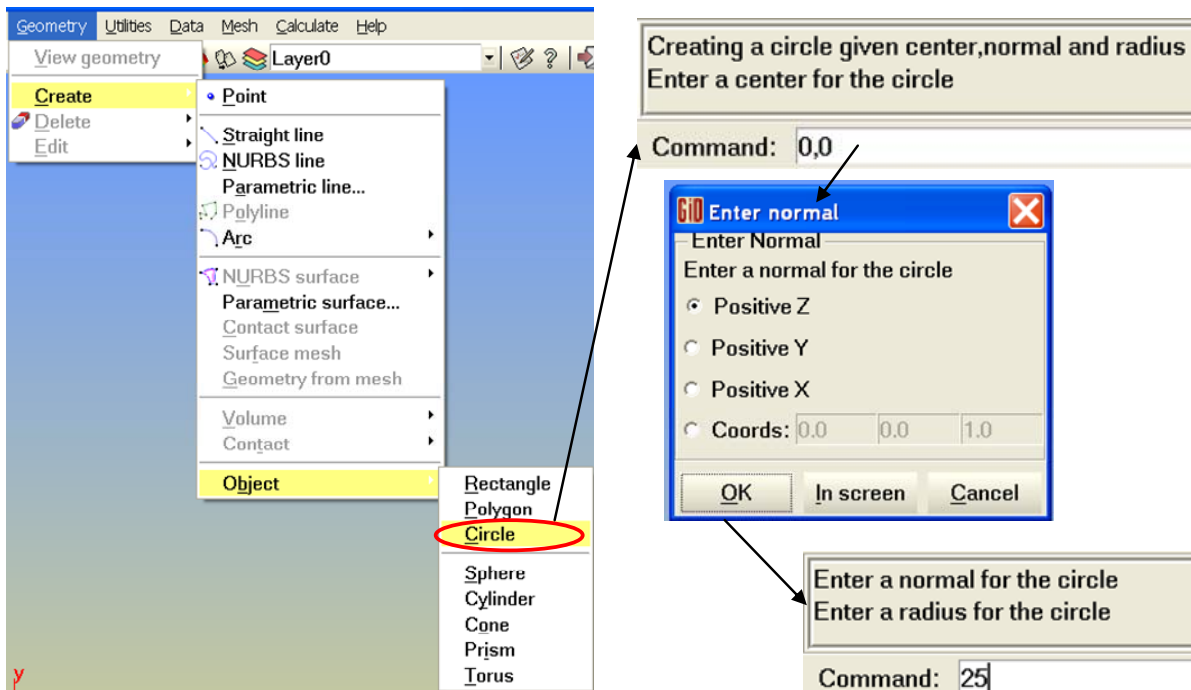




Figure C2.2. Menu sequence for creating a circle

## Alternative 2: Using the Icons

To create the circle using the icons bar, Click , the object icon drop menu will appear. Then click the circle icon . In the dialog box, enter the center for the circle (0,0) and press Enter. Then, enter a normal for the circle. In our case, choose **Positive Z**. Finally, enter the radius for the circle in the dialog box (25) and press Enter to finish the circle generation. This sequence is illustrated in Fig C2.3.

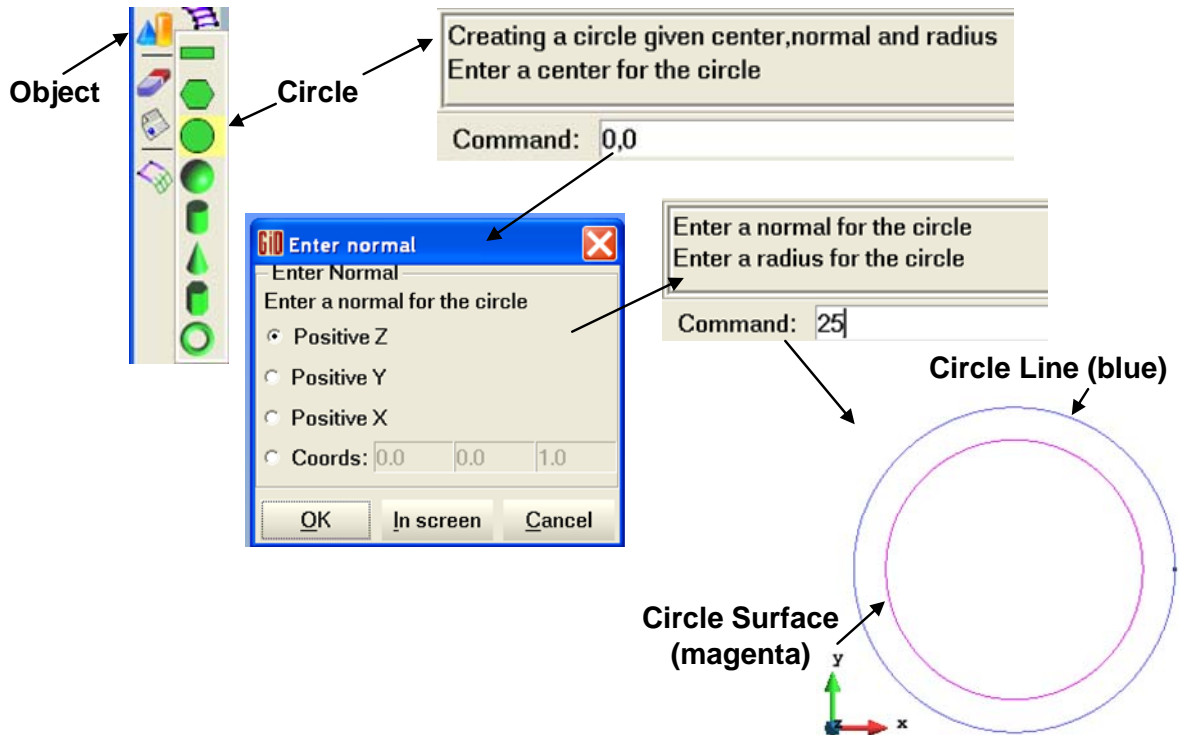


Figure C2.3. Icon sequence for creating a circle.

The circle line (in blue) has been created with the corresponding surface (in magenta color). In this case, to allow quadrilateral meshing in the center of the disc, it is required to create an additional surface in the center are of the disc. This will be done later on. Since two surfaces will be created, the first thing is to remove the surface created for the disc and then created the needed new surface in the center.

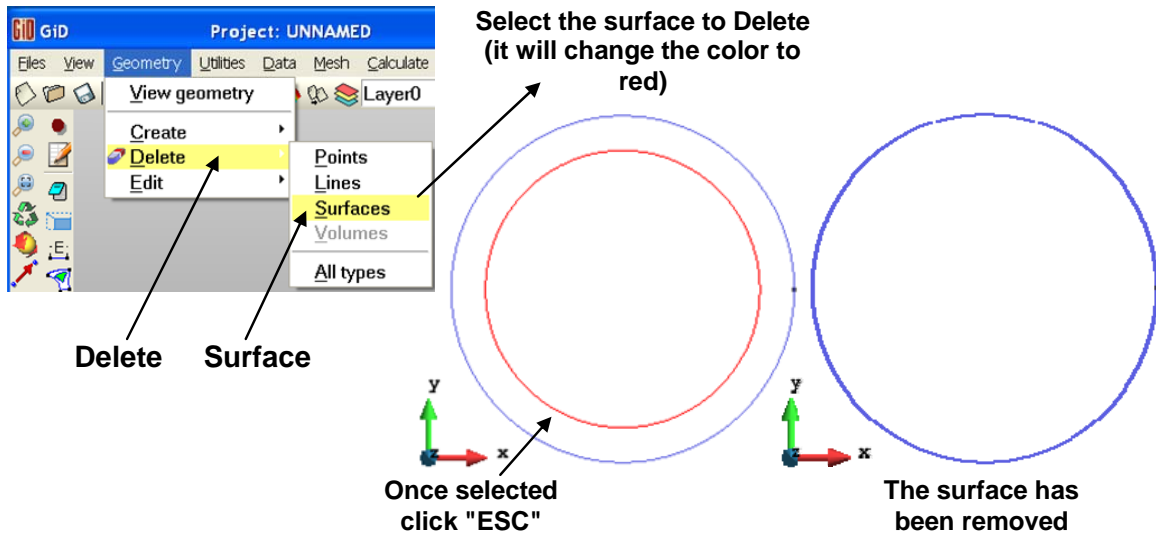
## 2.2 Deleting the Surface

**Note:** As indicated, this step is necessary before creating a quadrilateral element in the center for better meshing. Failure to delete this surface will cause issues later because of overlapping surfaces.

### Alternative 1: Delete Surface Using the Drop Down Menus



Geometry → Delete → Surfaces → Select Circle Surface → ESC

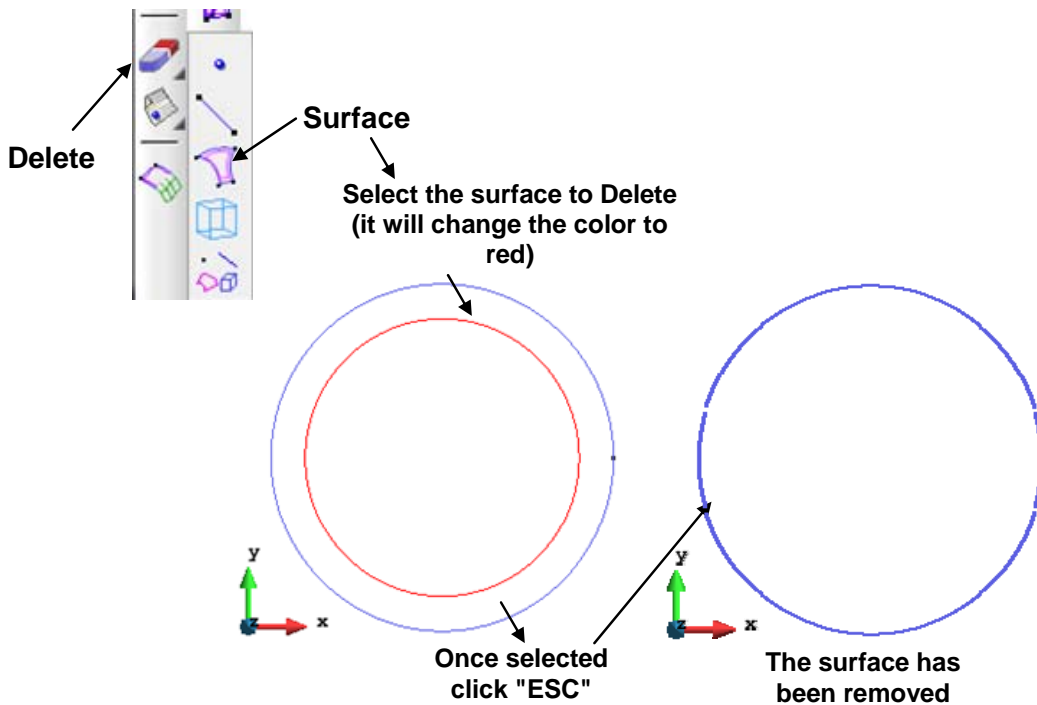
The process of deleting a surface using the drop down menu illustrated in Figure C2.4.



**Figure C2.4.** (Left) Menu sequence for deleting a surface; (Right) Result after removing the surface. Only the circle line remains.

### Alternative 2: Delete Surface Using the Icons

To remove the surface using the icons, Click . The delete drop menu will appear. Choose the surface icon, , to remove only the surface. Then, select Circle Surface to be removed. The surface will be highlighted in red. Press ESC to remove the surface. This sequence is shown in Figure C2.5.



**Figure C2.5.** Icon sequence for deleting a surface

## 2.3 Creating the center quadrilateral element for meshing of the center

As indicated, to obtain a better mesh, a quadrilateral element will be created at the center of the circle.

### Alternative 1: Using the Drop Down Menus

**Geometry → Create → Straight line → Points (0,5) → Enter → Points (5,0) → Enter → Points (0,-5) → Enter → Points (-5,0) → Enter → Points (0,5) → Enter → Join → ESC.**

This sequence is shown in Figure C2.6.

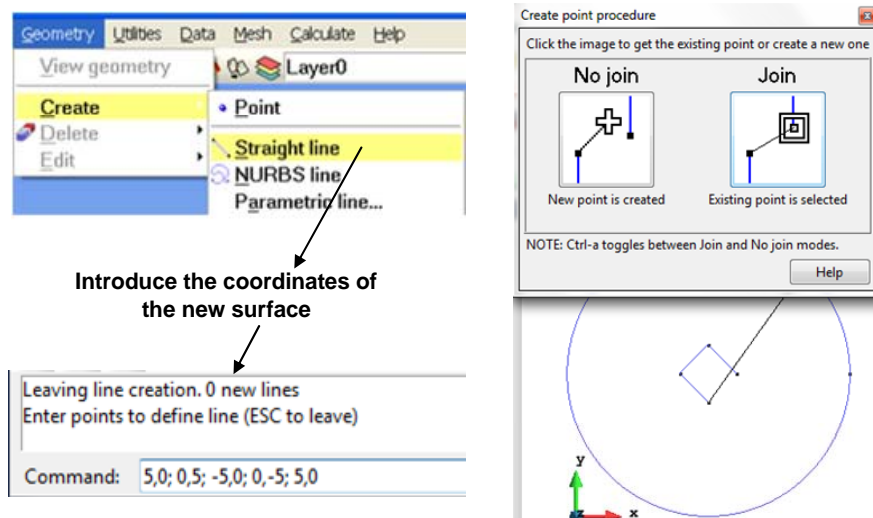



Figure C2.6. Menu sequence for creating the quadrilateral

### Alternative 2: Using the Icons

Click  → Points (0,5) → Enter → Points (5,0) → Enter → Points (0,-5) → Enter → Points (-5,0) → Enter → Points (0,5) → Enter → Join → ESC. This sequence is shown in Figure C2.7.

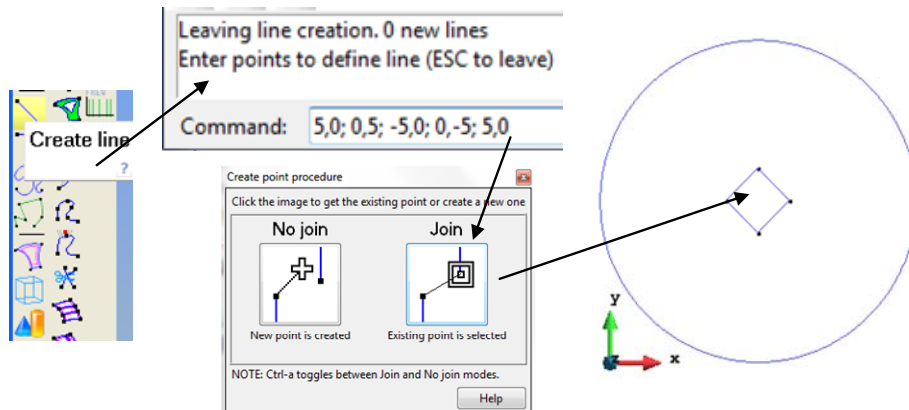


Figure C2.7. Icon sequence for creating the quadrilateral

## 2.4 Dividing the circle


The corners of the quadrilateral (square) need to be connected to the circle. The coordinates of the square were created to lie on the axis of the circle. Dividing the circle using the divide function will place a point on the each axis. This sequence is shown in Figure C2.8. The result is shown in Figure C2.9.

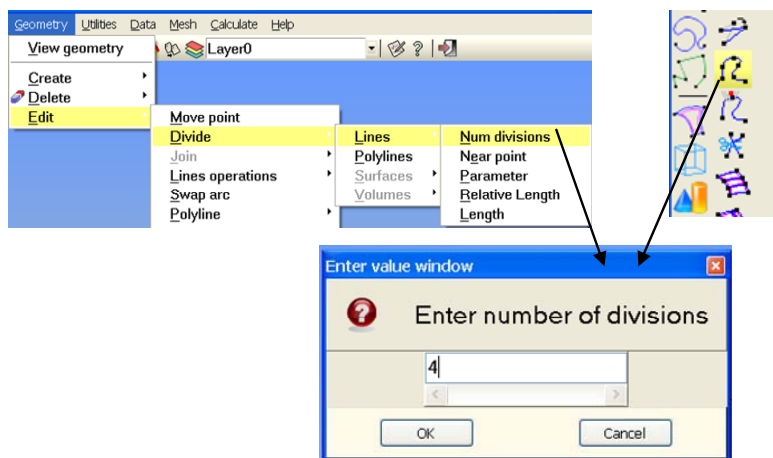
### Alternative 1: Using the Drop Down Menus

Click **Geometry** → **Edit** → **Divide** → **Lines** → **Num divisions** → **Enter (4)** → **OK**

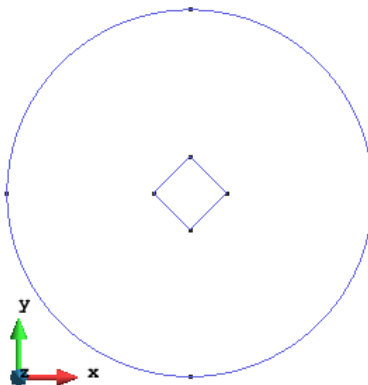
This sequence can be seen in Figure C2.8.

### Alternative 2: Using the Icons

Click . Then enter the number of divisions. Enter (4). Press OK to finish. This sequence can be seen in Figure C2.8.



**Figure C2.8.** Sequence for dividing a line using drop down menus (left) and icons (right)



**Figure C2.9.** The divided circle with a quadrilateral at the center

## 2.5 Creating Connecting lines


The connecting lines need to be created from the corners of the quadrilateral to the points created on the circle. Both the menu and icon sequence are shown in the top panels of Figure C2.10. The first line and the final geometry are shown in the bottom panels of Figure C2.10.

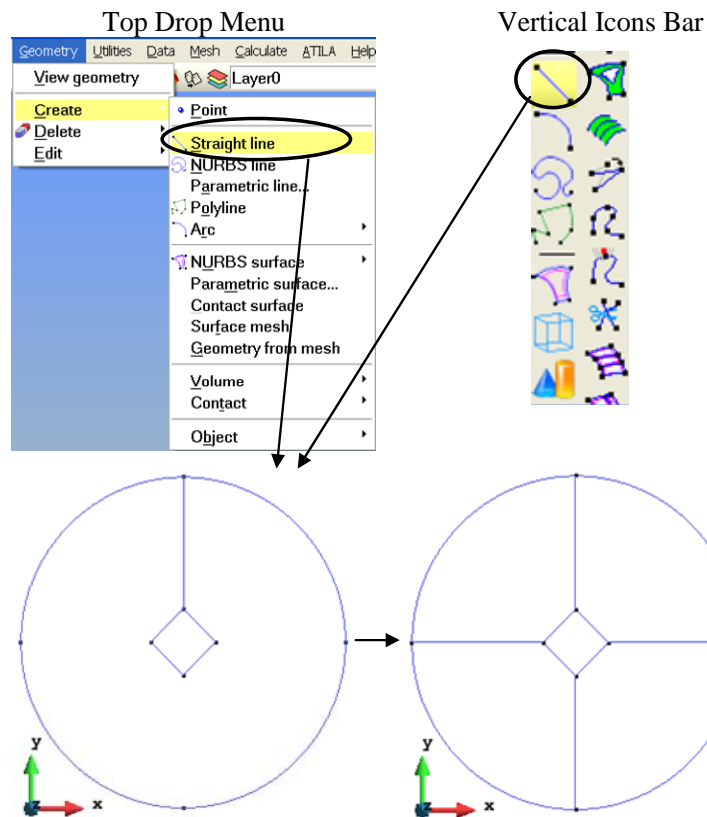
### Alternative 1: Using the Drop Down Menus

Click Geometry → Create → Straight line → Control-A → □ → Select 1<sup>st</sup> 2 - points → ESC → Select 2<sup>nd</sup> 2 - points → ESC → Select 3<sup>rd</sup> 2 - points → ESC → Select 4<sup>th</sup> 2 - points → ESC → ESC

This sequence can be seen in Figure C2.10 -top left.

### Alternative 2: Using the Icons

Click  → Control-A → □ → Select 1<sup>st</sup> 2 - points → ESC → Select 2<sup>nd</sup> 2 - points → ESC → Select 3<sup>rd</sup> 2 - points → ESC → Select 4<sup>th</sup> 2 - points → ESC → ESC. See Figure C2.10 top right.



**Figure C2.10.** Connecting the quadrilateral to the circle using the menus

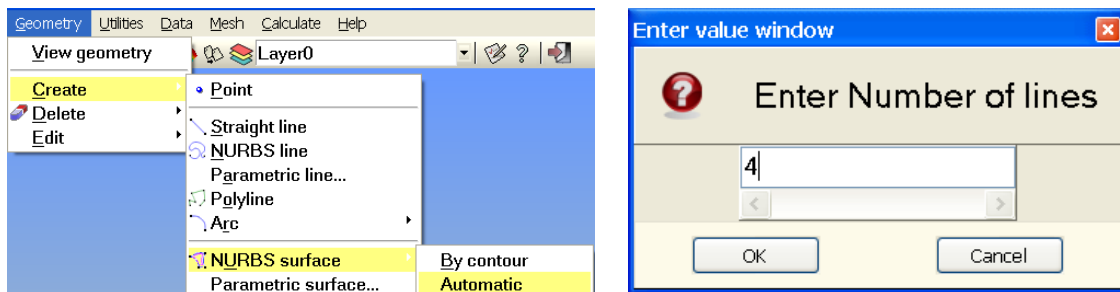
## 2.6 Create surface

### Alternative 1: Using the Drop Down Menus

GID will create surfaces automatically based on the information entered.

Click **Geometry** → **Create** → **NURBS surface** → **Automatic** → **Enter (4)** → **OK**


This sequence is shown in Figure C2.11. The resultant surfaces are shown in Figure C2.12.

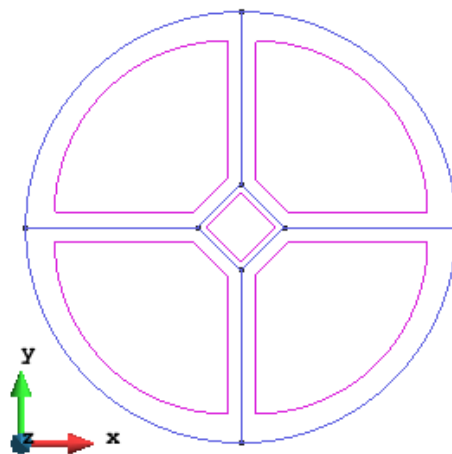


**Figure C2.11.** Menu sequence for creating surfaces

### Alternative 2: Using the Icons

The icon option requires that the bounding lines of the surface be selected to create the surface. This allows the user control over how the surfaces are created. Select the bounding entities that will form the edges of the surface, when finished with that surface click **ESC**. The cursor will remain for the creation of additional surfaces. The resultant surfaces are shown in Figure C2.12.

Click  → (□) → Select the bounding lines of the first surface → **ESC** → Select the 2<sup>nd</sup> set of bounding lines for the next surface → **ESC** → Repeat the same for the rest of surfaces → **ESC** to quit.



**Figure C2.12.** Geometry after the creation of surfaces



## 2.7 Create Volumes Using Copy and Create

### Alternative 1: Using the Drop Down Menus

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

In Case 1 the volume was created directly when the 2D geometry was extruded into 3D. This is usually the fastest way. This example will use a second method that creates the 3D geometry then the volumes to show an alternative method of volume creation.

Click **Utilities** → **Copy**.

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 - **Do Extrude Surfaces, Maintain Layers, and Multiple copies is 1**. This will copy the circular surface in the positive Z direction and extrude the cylinder at the same time. It will **not** create the volume. This sequence is illustrated in Figure C2.13.

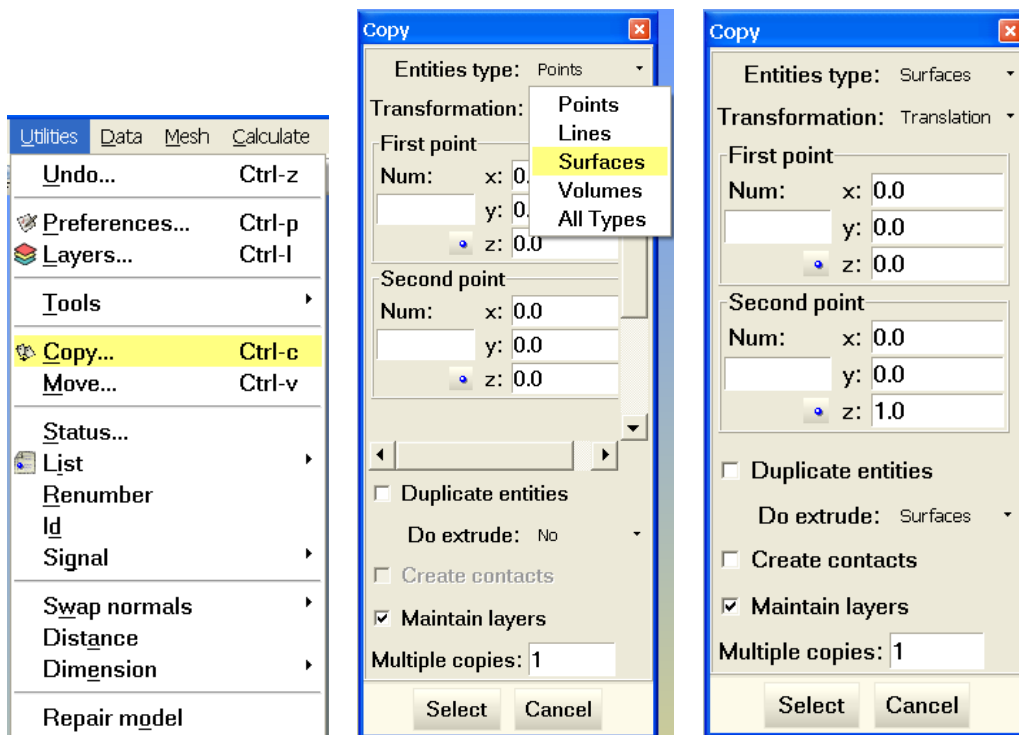
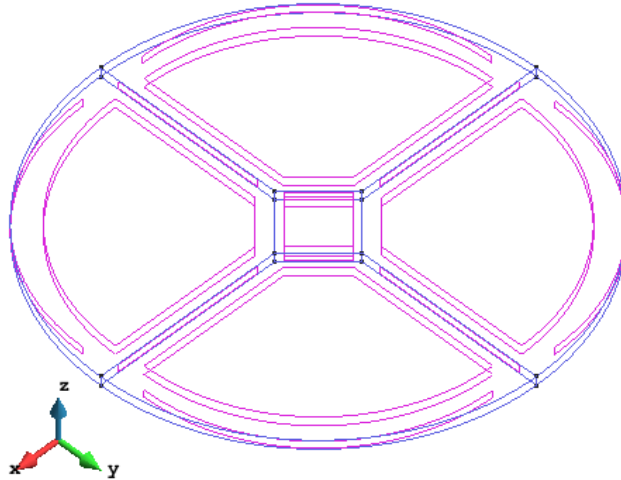


Figure C2.13. Settings for the Copy Control Panel

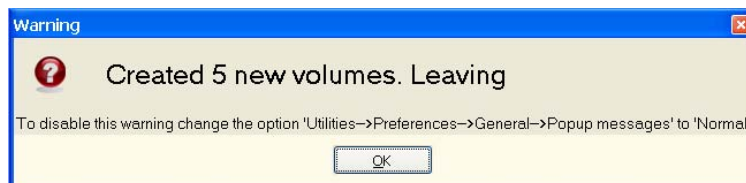
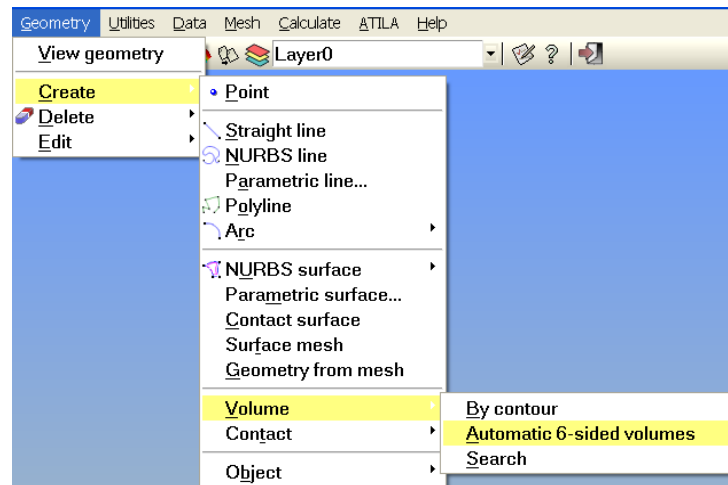
To complete the copy operation Click **Select** in the **Copy** menu. The cursor will change to a large square with a small square inside (□). Select the surfaces of the Circle and quadrilateral. The color will change from magenta to red indicating that GiD has accepted the selection. Click **Finish** to complete the copy. GiD translated the surface and extruded the cylinder to create a solid model and the corresponding surfaces. This resulting geometry is illustrated Figure C2.14.



**Figure C2.14.** Circle geometry and surfaces copie/extruded

The volumes can be added quickly using the menus. The Warning panel confirms the creation of the volumes. This sequence is illustrated in Figure C2.15. Figure C2.17 illustrates the disk after all of the volumes have been created.


**Click Geometry → Create → Volume → Automatic 6-sided volumes → Warning panel with confirmation appears → OK**



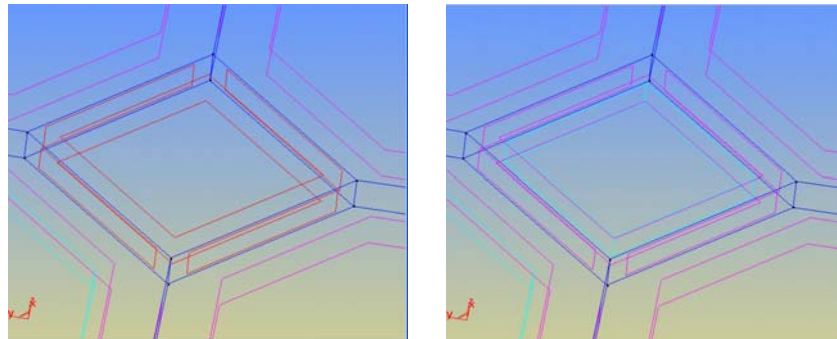
**Figure C2.15.** Settings for the Copy Control Panel

### Alternative 2: Using the Icons

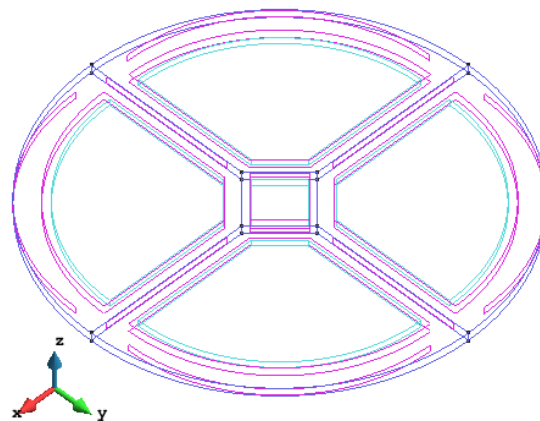
The same process can be accomplished with the icons, however, it requires that all of the bounding surfaces for each volume be selected.

Click  → (□) → Select surfaces that bound the proposed volume → Esc to create the volume → Esc to quit (When all volumes have been created)

See selection of the quadrilateral and the creation of the volume in Figure C2.16. Figure C2.17 illustrates the disk after all of the volumes have been created.

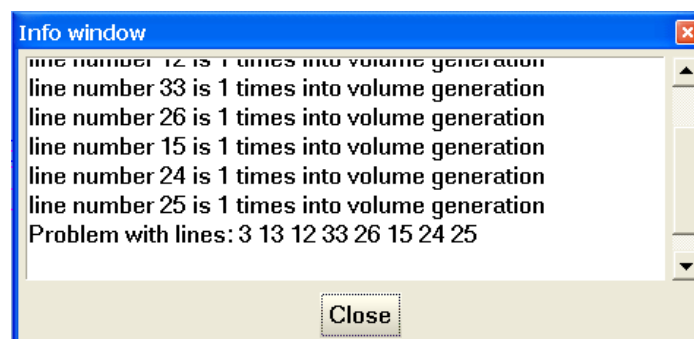


**Figure C2.16.** Selection and creation of Volumes



**Figure C2.17.** Complete Disk

**Note:** If all bounding surfaces are not selected a warning panel appears and the volume is not created. See Figure C2.18. for an example of the warning panel.



**Figure C2.18.** Volume creation error Information window

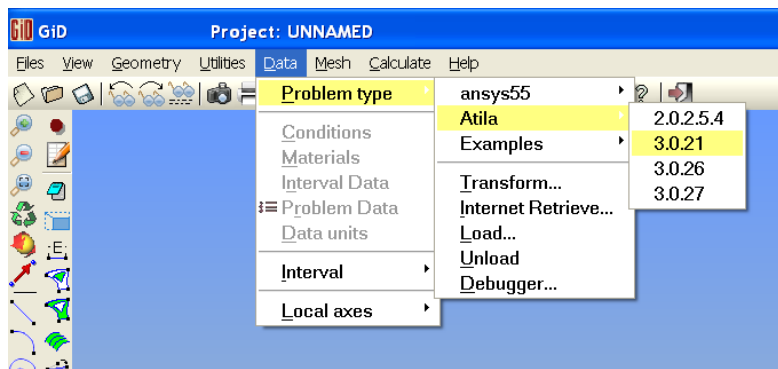
# MATERIALS AND CONDITIONS

## 3. Materials Assignment

The next step is to assign the material and boundary conditions necessary for the simulation. Begin by activating ATILA in the Problem type.

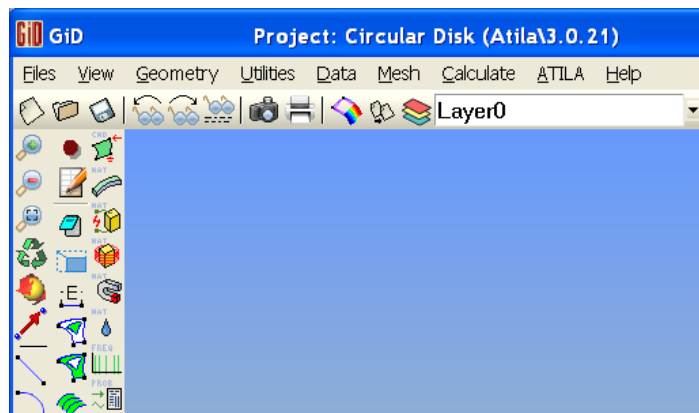
**Click Data → Problem type → Atila → 3.0.21.**

*Note:* Other compatible versions of Atila may be present, for this example we will use 3.0.21. See Figure C2.19.



**Figure C2.19.** Starting ATILA 3.0.21

The header will now show the version of Atila being used and the menu and Icons will be available for use, as shown in Figure C2.20.




**Figure C2.20.** ATILA 3.0.21 Menus and Icons

### Alternative 1: Using the drop down menu

**Click Data → Materials → Piezoelectric → Control Panel appears → Select PZT4  
→ Assign by Volumes → Select Volumes → Finish**

This sequence is illustrated in Figure C2.21 at the top left panel.

### Alternative 2: Using the Icons

Click  → Control Panel appears → Select PZT4 → Assign by Volumes → Select Volumes → Finish. This is illustrated in Figure C2.21 at the top right panel.

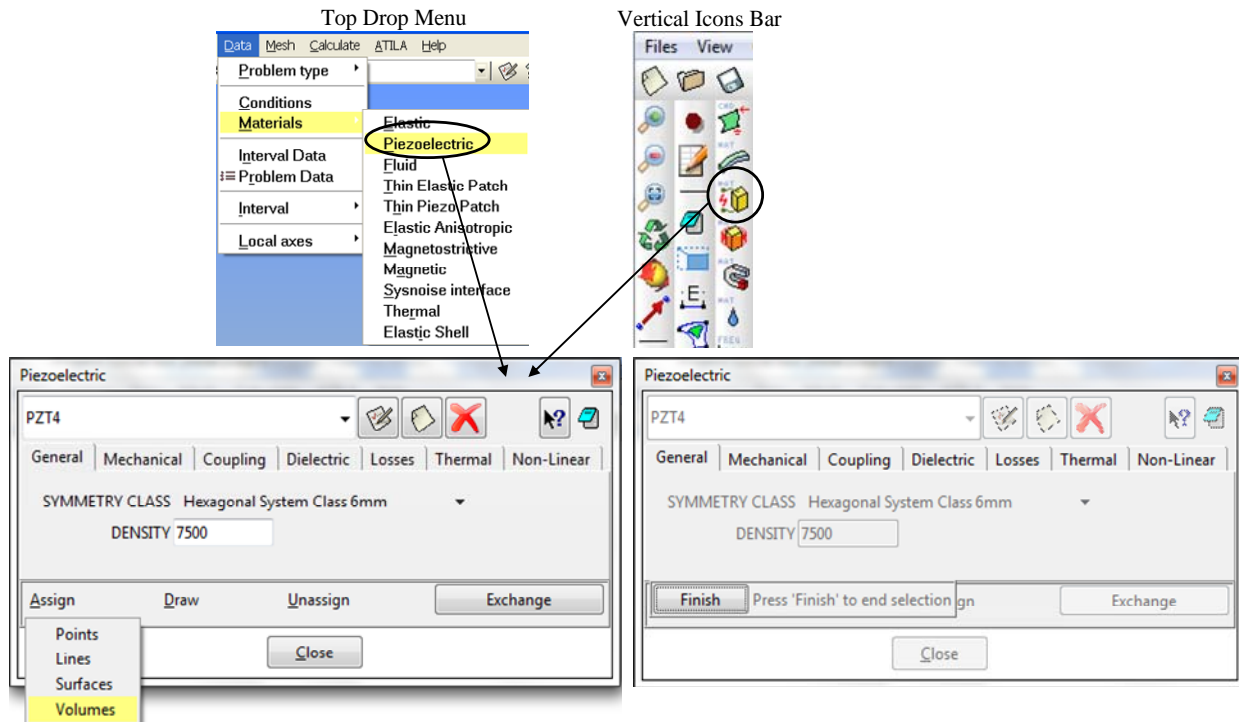


Figure C2.21. Piezoelectric Material Selection

Select all the volumes of the Disk, they 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. See Figure C2.22.

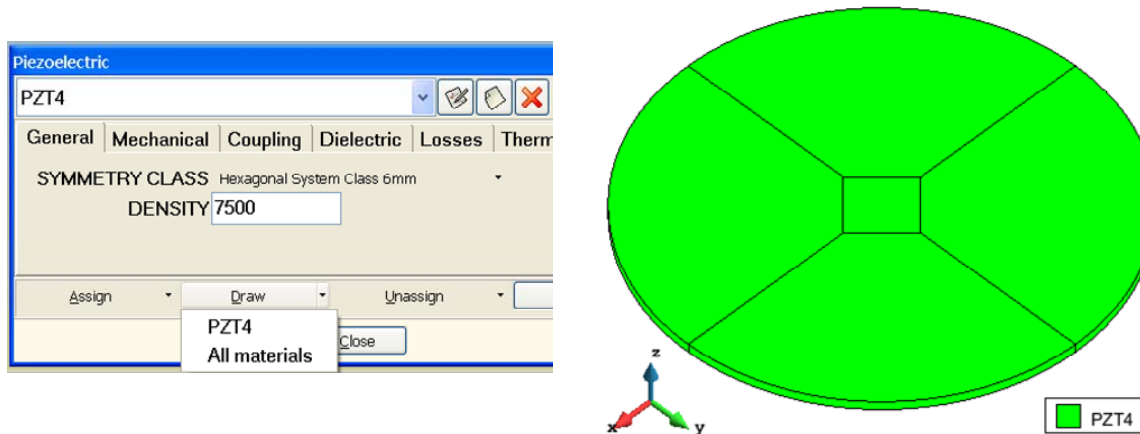


Figure C2.22. Piezoelectric Material Selection


## 4. Boundary Conditions Assignment

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 C2.23 left panel. The conditions panel appears as seen in Figure C2.23 bottom panel.

### Alternative 2: Using the Icons

Click (  ) as shown in Figure C2.23, right panel. The conditions panel appears as seen in Figure C2.23 bottom panel.

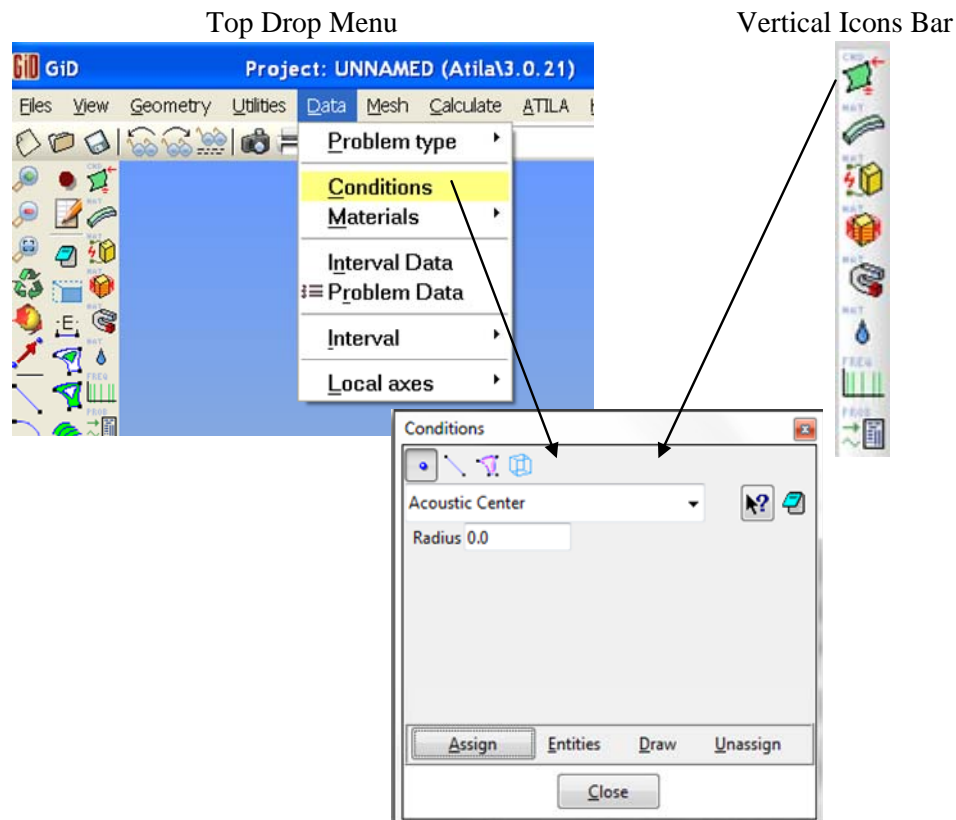



Figure C2.23. Boundary Conditions Assignment

## 4.2 ASSIGN ELECTRICAL POTENTIAL BOUNDARY CONDITIONS

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 C1.24.

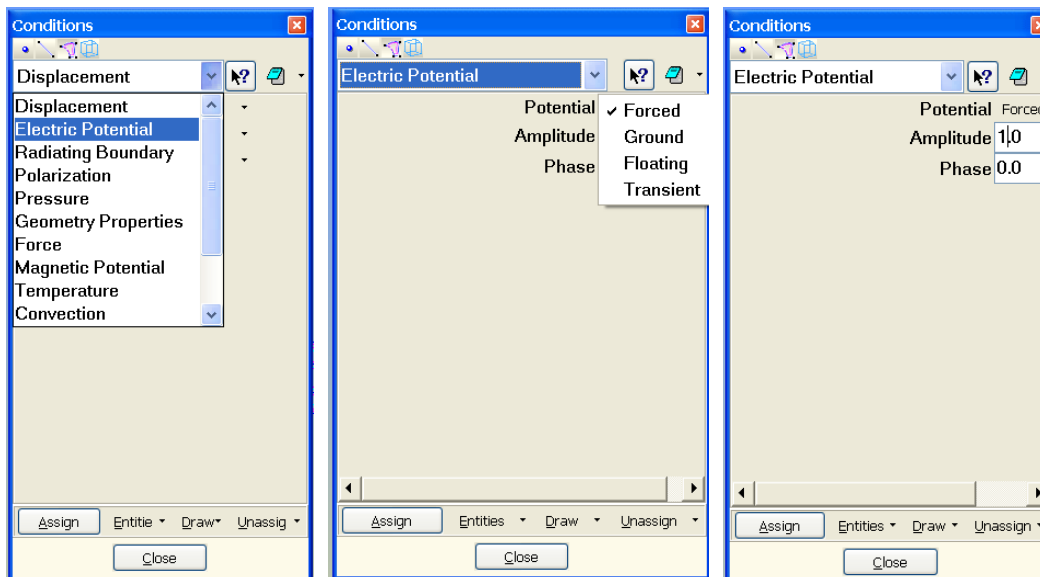



Figure C1.24. Boundary Conditions Control Panel

### 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 C.25. Click **Finish** to quit.

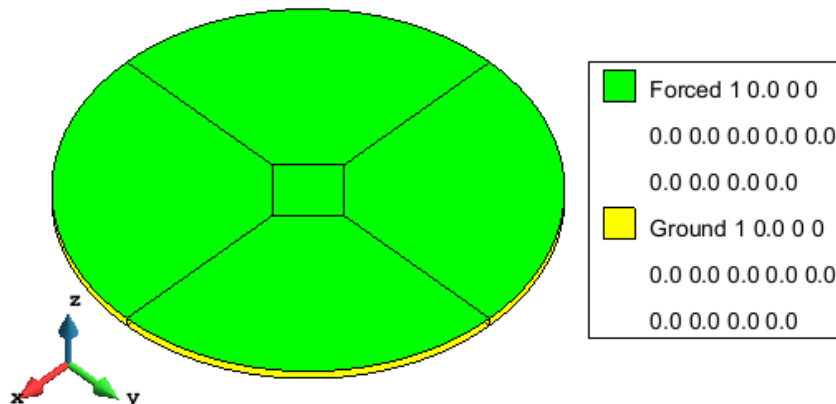



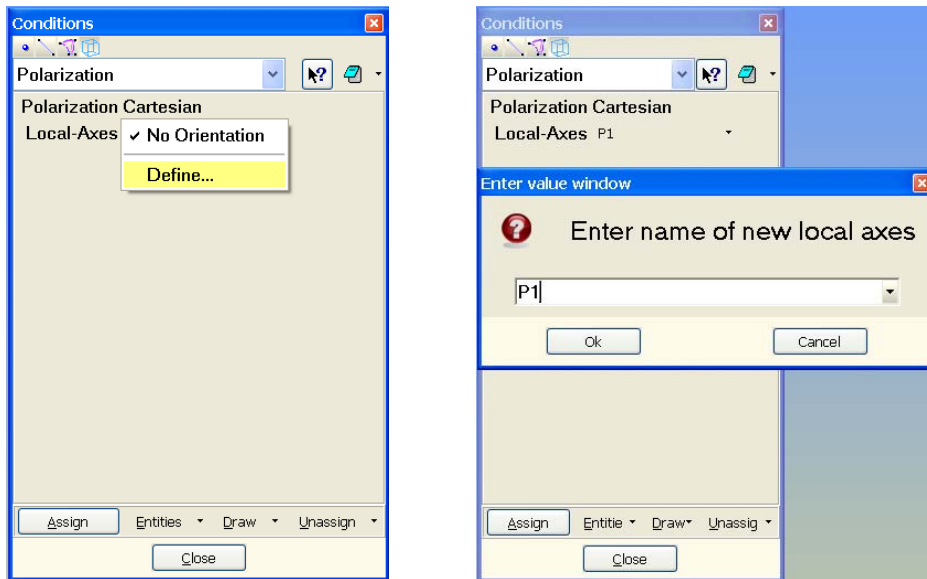
Figure C2.25. Electrical Potential assignment confirmation

### 4.3 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.

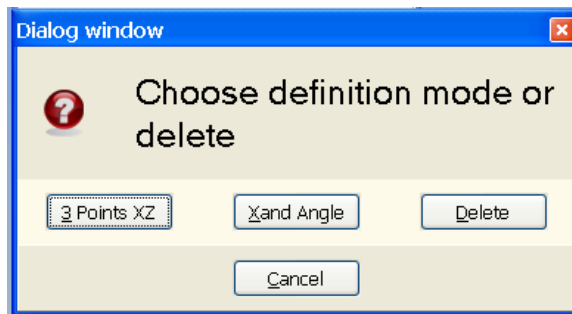
Click the volume icon (  ) on the top of the control panel, then select **Polarization** from the list. In this example the default, **Cartesian**, is selected. Click **Define**. The Local Axis naming panel will appear as shown in Figure C1.26.

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**.



**Figure C2.26.** Boundary Conditions Control Panel - Polarization

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 C1.27.



**Figure C2.27.** Local axis dialog window

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

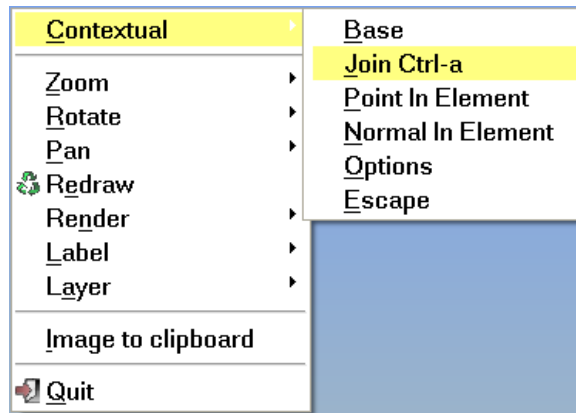


There are two options to define the local axis:

### 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 C1.28. 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**.

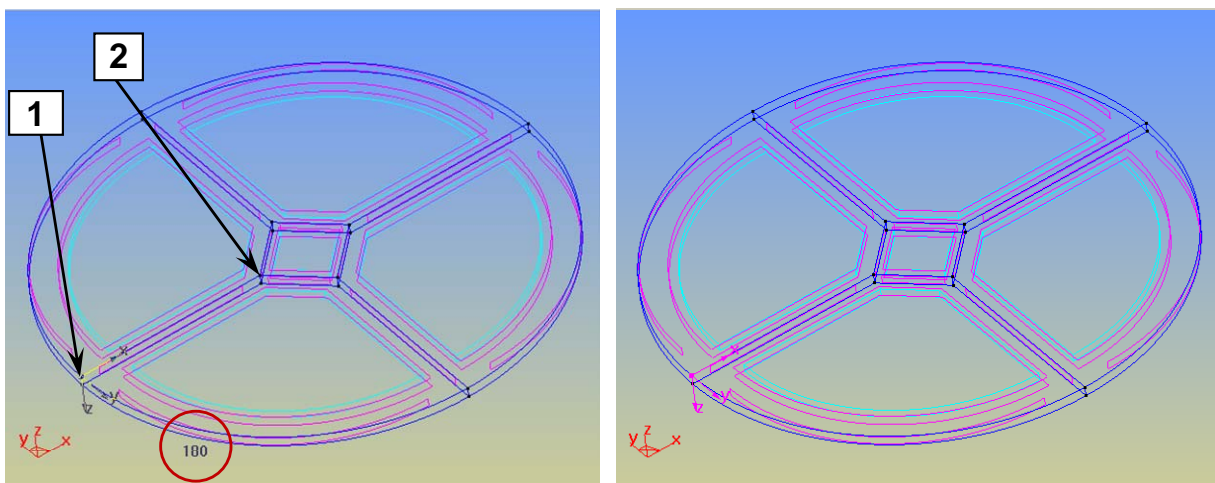
Click **X and Angle** → (+) → **Join Ctrl-a** → 1<sup>st</sup> point → **Join Ctrl-a** → 2<sup>nd</sup> point → **Select Angle** → **Assign volumes** → **Finish or ESC**.



**Figure C2.28.** Onscreen menu

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 these selections in Figure C2.29. left panel.

A moveable axis will appear, move the mouse to orient it in the desired direction, for this example match the graphic “Z” axis or 180 degrees. The angle will appear next to the cursor, when it is at “180” click the **Right Mouse Button** to select. See illustration in Figure C2.29 left panel, red circle. The polarization axis is shown in Figure C2.29. right panel.



**Figure C2.29.** Creation of Local axis

## Alternative 2: Coordinate Entry Option

Click **X and Angle** → Enter axis center → Enter → Enter point in positive X direction → Enter → Enter Angle for Y → Assign volumes → Finish or ESC.

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 shown in Figure C2.30. The Polarization axis will appear as shown in Figure C2.29, right panel.

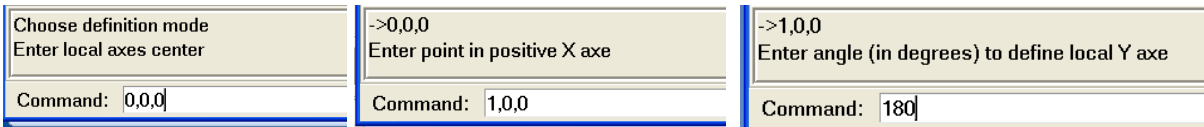


Figure C2.30. Boundary Conditions Control Panel

To confirm the assignment of polarization click **Draw** → **All conditions** → **Include local axis**. The process and results are shown in Figure C2.31.

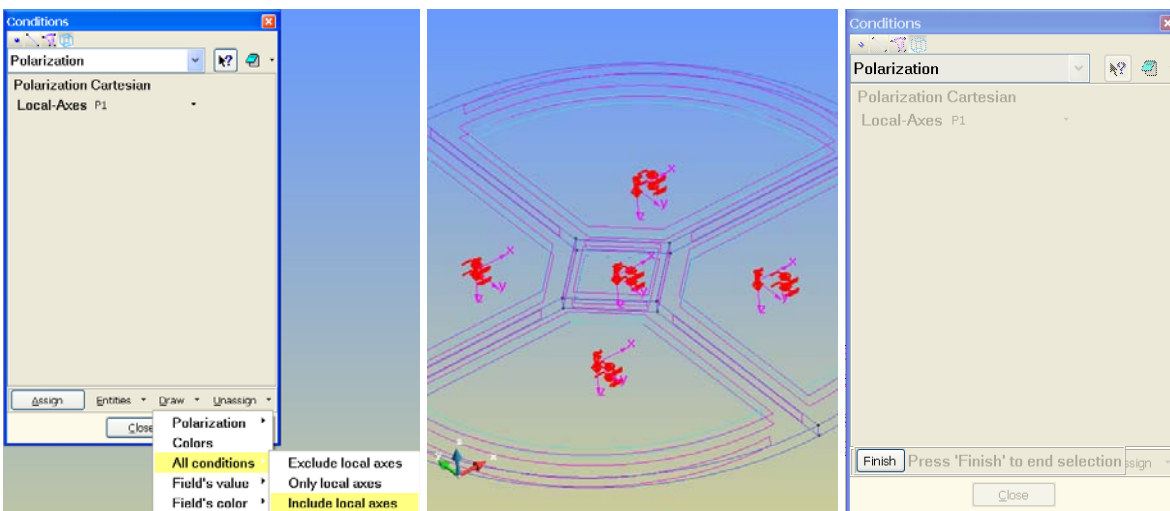


Figure C2.31. Confirmation of Polarization

**Note:** All of the conditions will be shown in red. See center panel of Figure C2.31.

Select **Finish** or **ESC** to quit **Draw**.

# MESH

## 5. 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.

### 5.1 SETTING MESH TYPE

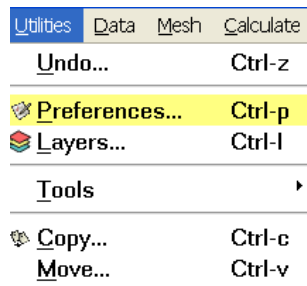
The default mesh type in GiD is Normal, this example will use Quadratic.

**Note:** Changes made in the preferences will remain until changed or the preferences reset to the default settings.

#### Alternative 1: Change mesh type Menu option

Click **Utilities** → **Preferences** → **Preference Control Panel** appears → **Meshing** → **Quadratic** → **Accept** → **Close**

The menu is shown in Figure C2.32, the Preference Control Panel is shown in Figure C2.33.



**Figure C2.32.** Menu selection for Preferences

#### Alternative 2: Change mesh type Hot Key option

**CTRL + P** → **Preference Control Panel** appears → **Meshing** → **Quadratic** → **Accept** → **Close**

The Preference Control Panel is shown in Figure C2.33.

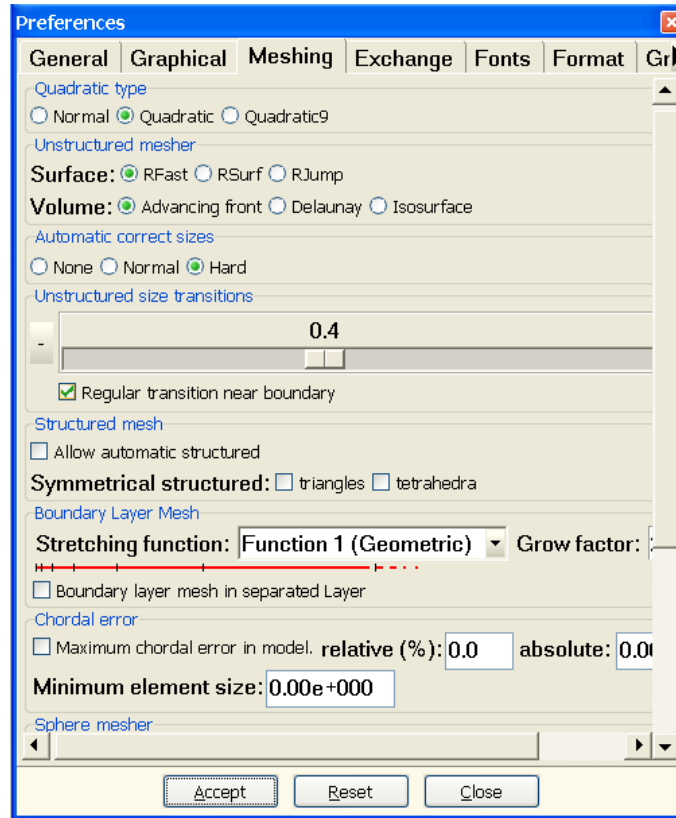


Figure C2.33. Preferences Control Panel

## 5.2 STRUCTURED MESH

To setup the structured mesh follow these steps as illustrated in Figure C2.34.

Click **Mesh** → **Structured** → **Volumes** → **Assign number of cells**

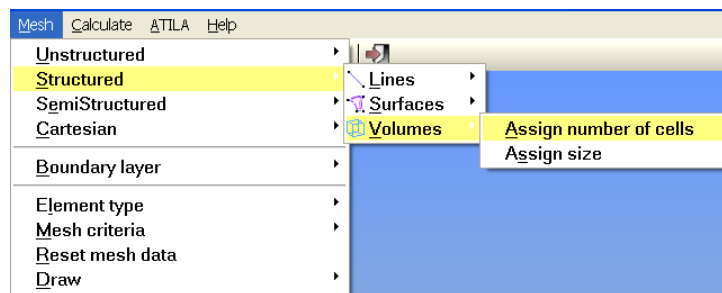
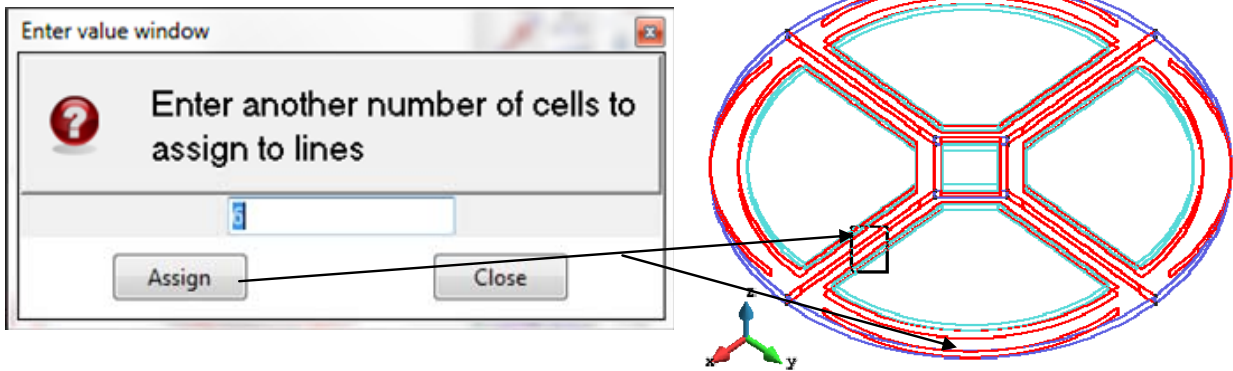


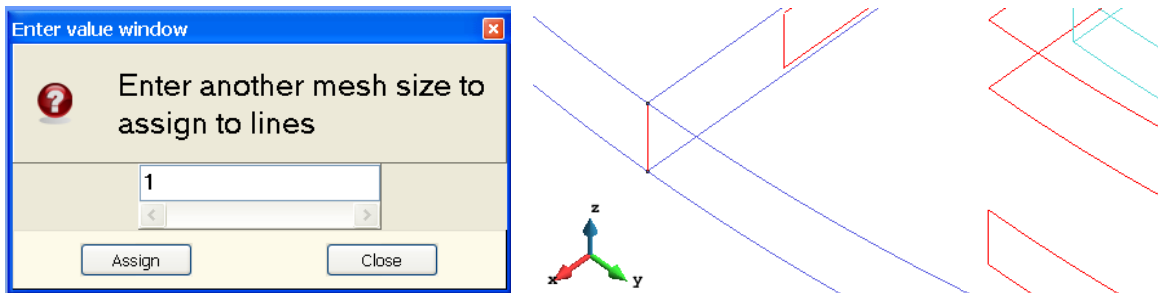
Figure C2.34. Creating a Structured Mesh

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. The mesh of adjacent volumes must be the same size.). This example will mesh all of the volumes with the same size elements. The control panel shown in Figure C2.35., left panel, will appear. Enter the number of divisions for a line. Click **Assign**.



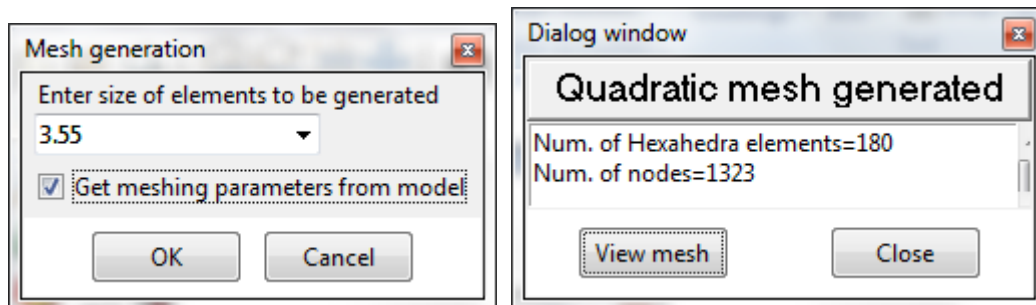
**Figure C2.35.** Assigning number of divisions. Select the 6 divisions for the radial lines and the circumference lines.

The cursor will change ( $\square$ ) to allow the selection of the lines for that number of divisions. When the lines are selected they will turn red indicating acceptance. (*Note:* Depending on the method of construction lines that are opposite one another will all be selected together, this is the case in this example. See Figure C2.36. 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 different value as shown in Figure C2.36., left panel. The selection process is the same as used in the previous step. When all of the lines have been assigned a number of divisions press **ESC**, then **ESC** again to quit.

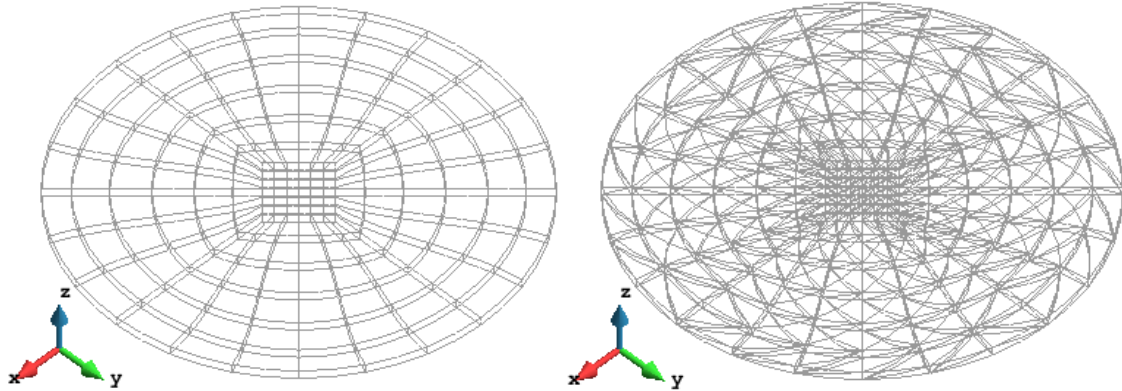


**Figure C2.36.** Assigning number of divisions for the plate thickness.

Press **CTRL + G**, the Mesh Generation control panel below will appear. Click **Get meshing parameters from model** and click **OK**, see Figure C2.37. left panel. The mesh will be generated, select **View mesh** to see the mesh as illustrated in Figure C2.38 panel.



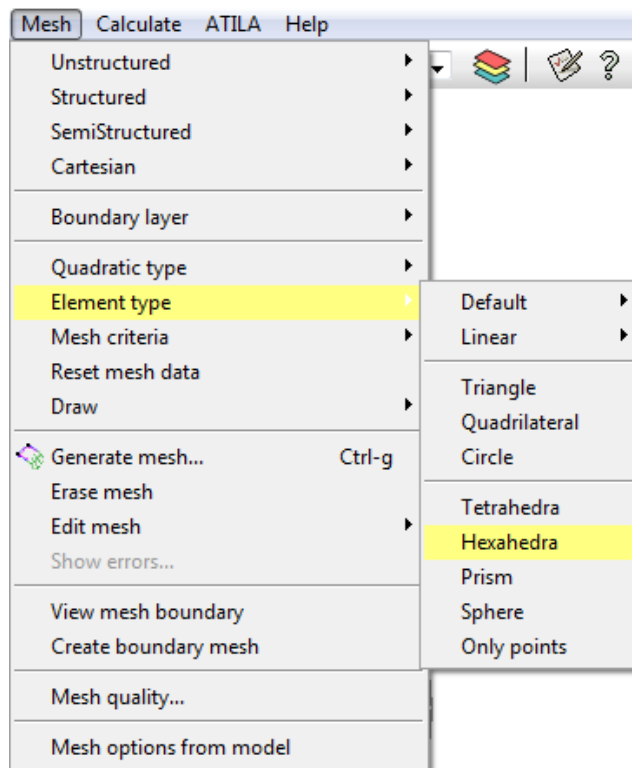
**Figure C2.37.** Meshing the geometry



**Figure C2.38.** Meshing the geometry. Left: using hexahedra elements; Right: using tetrahedra elements.

*Note:* In GiD 10, the default element type is hexahedra, so the resulting meshing will be the one shown in Figure C2.38 left.

However, in GiD 11, the default element type is tetrahedra and the results of using **CTR + G** will be as shown in Figure C2.38 right. Hexahedra elements uses less numbers of nodes than when using tetrahedra elements. To select hexahedra as the element for the mesh, select **Mesh → Element Type → Hexahedra**. This can be observed in Figure C2.39



**Figure C2.39.** Selection of Hexahedra elements for the meshing.

# CALCULATE

## 6. 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 menu assigns the rules for the simulation. These rules determine the type of simulation, the number of steps, etc that are to be run to generate the data for analysis.

### 6.1 HARMONIC CALCULATION

#### STEP 1: SELECT HARMONIC ANALYSIS FROM PROBLEM DATA

Click **Data** → **Problem Data**, the Control Panel will appear. (alternatively, click )

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 C2.40.

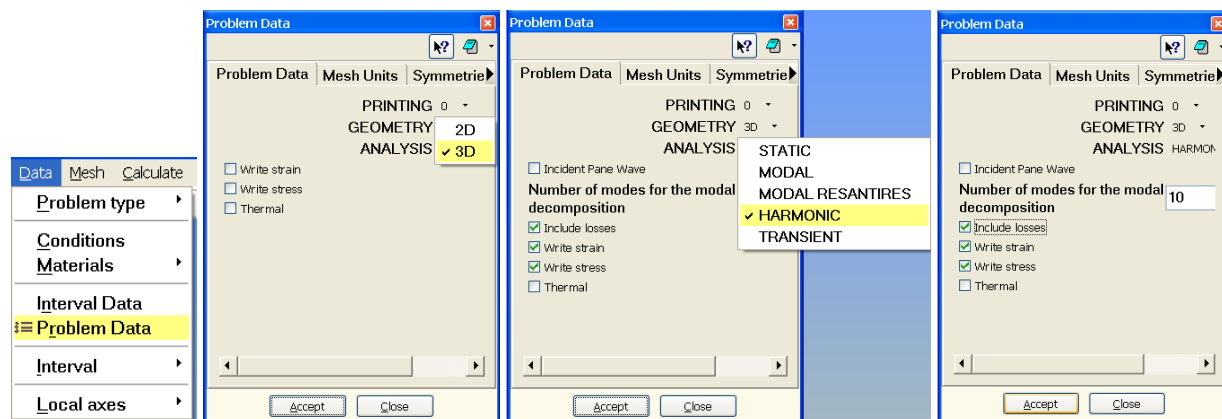


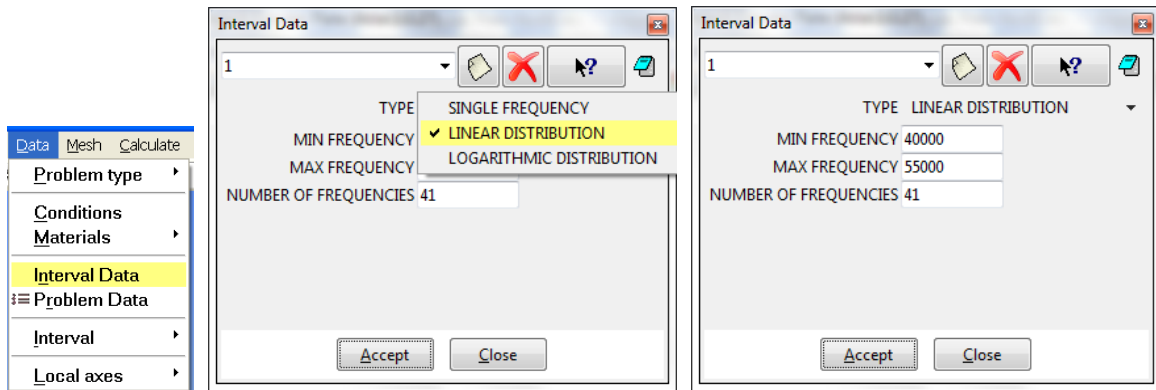
Figure C2.40. Assign Problem Data

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

Click **Data** → **Interval Data**. The Interval Data Control Panel will appear. (alternatively, click )

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

Enter the **Minimum** and **Maximum** frequencies and the **number of frequencies** 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*. The sequence is shown in Figure C2.41.



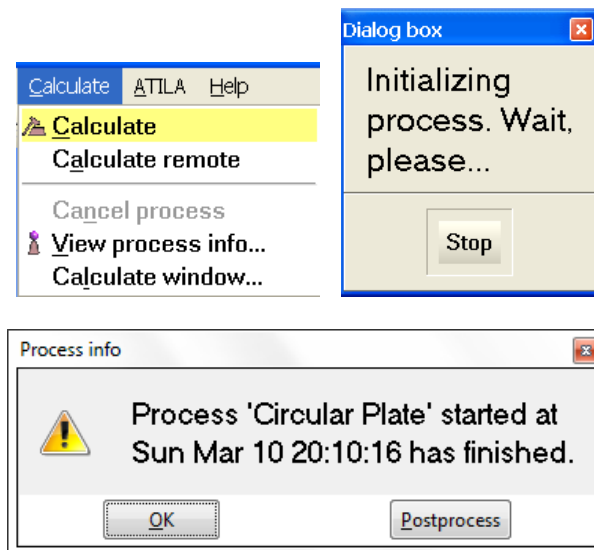
**Figure C2.41.** Assign Interval Data

At this point, all of the necessary information to perform an harmonic calculation has been entered. The next step is to run the calculation/simulation.

### **STEP 3: START THE CALCULATION**

Click **Calculate** → **Calculate** as shown in Figure C2.42 left panel. The calculation dialog box will appear briefly letting you know the calculation has started. See Figure C2.42 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 C2.42 right panel.





**Figure C2.42.** Calculation Panels



# RESULTS

## 7. 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 C2.43. On the left, select the Display Style icon  to change the style of visualization. In Figure C2.43, the display style  has been selected to activate the mesh.

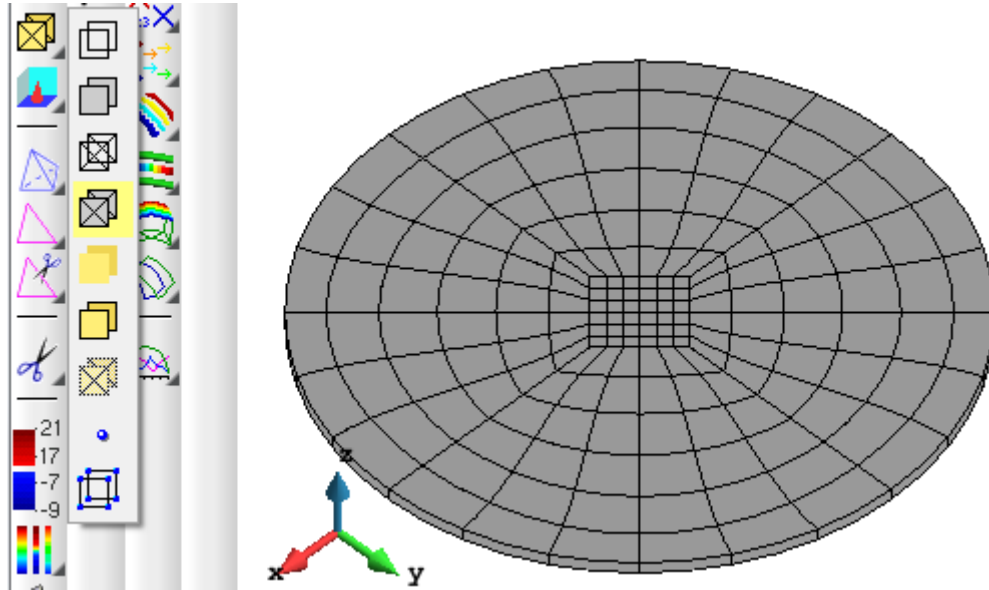



Figure C2.43. Post-Processor Window

**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.

### 7.1 ADMITTANCE GRAPH

Click () → **Admittance: Magnitude Results** → Select **Admittance** → Select **View Graph**.

The Admittance & Impedance Results control panel will appear, as seen in Figure C2.43.

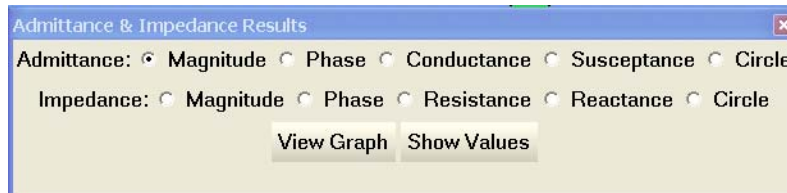
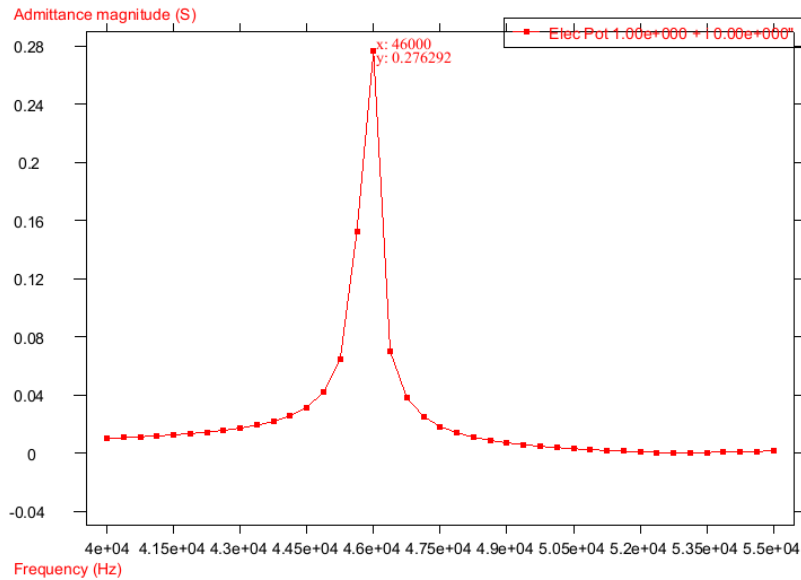


Figure C2.43. Admittance/Impedance Control Panel

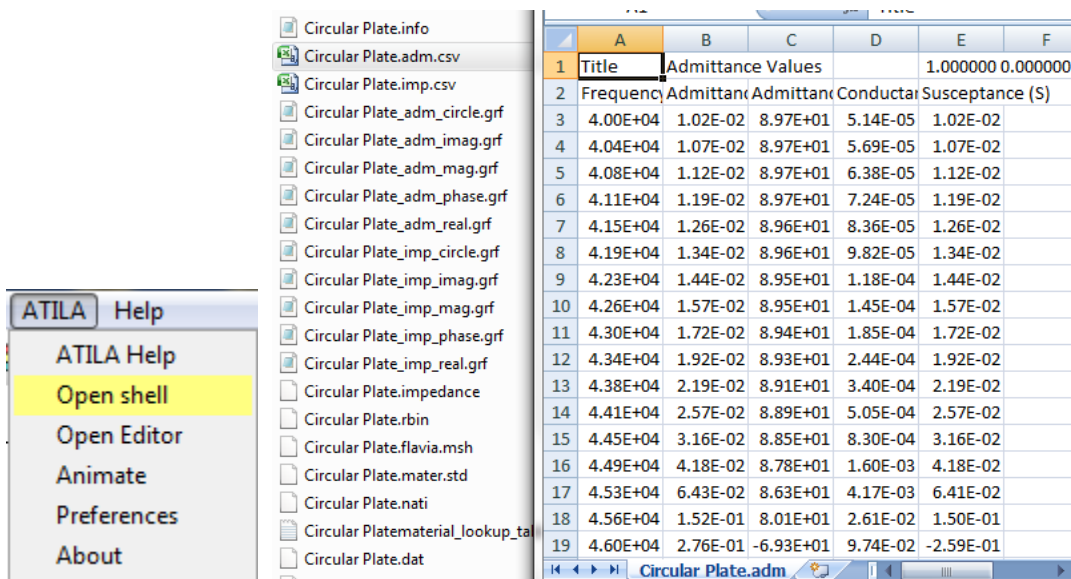
Select the desired characteristic, **Admittance: Magnitude**, then select **View Graph**. The Admittance magnitude graph will appear as seen in Figure C2.44.

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 C2.44. 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 C2.44 for the maximum value of the admittance plot.



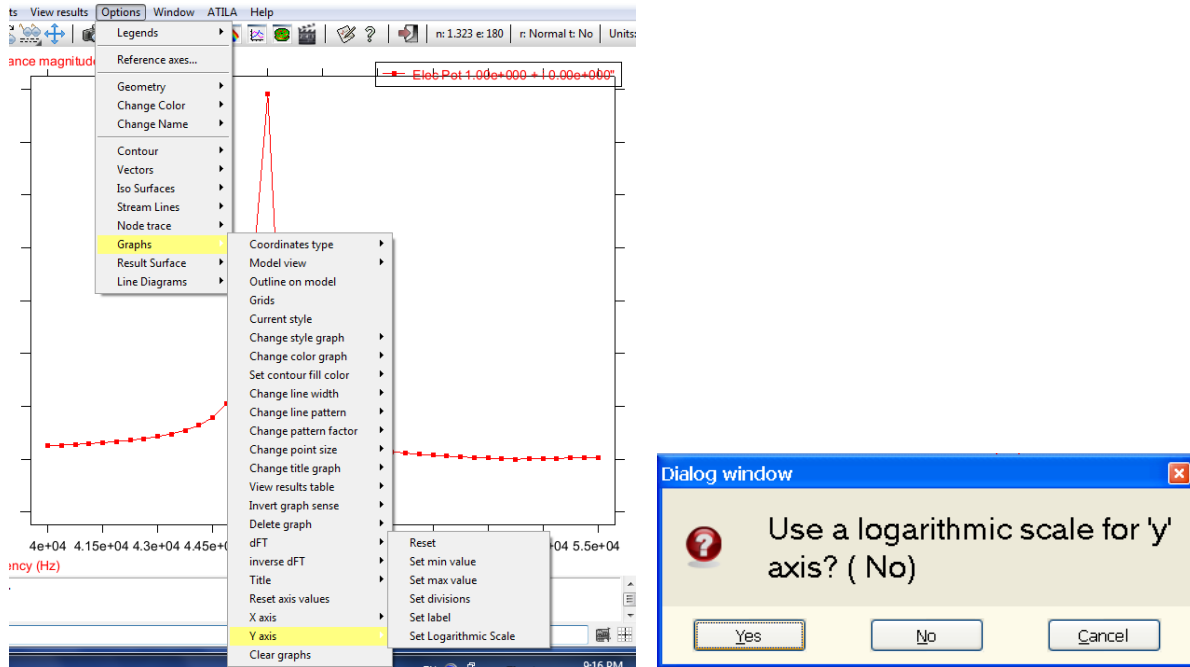
**Figure C2.44.** Admittance Magnitude Graph

The values of the graph can be exported to an Excel file for later analysis. In order to do so, select the **ATILA** menu and then **Open shell**. The file named with the **problem\_name.adm.csv** includes the admittance data. The file named **problem\_name.imp.csv** includes the impedance data. This is illustrated in Figure C2.45.



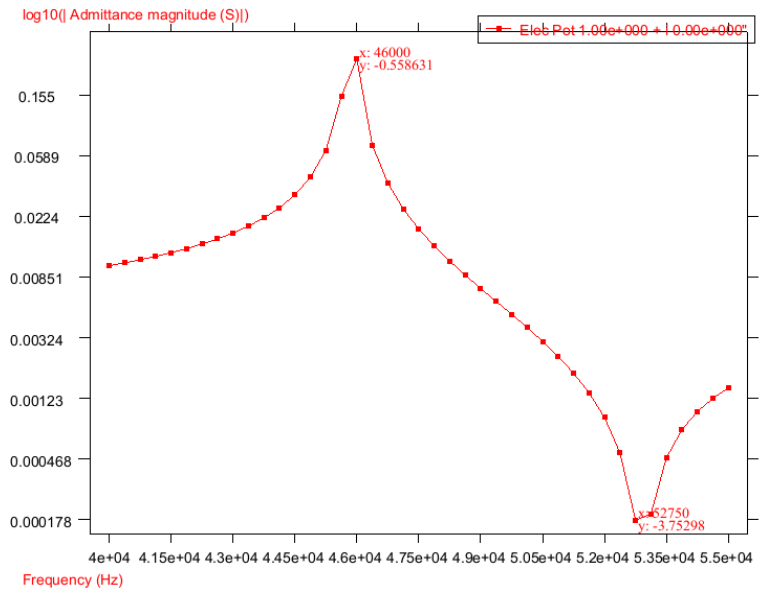
**Figure C2.45.** The impedance and admittance data can be open in Excel format.

A clearer picture of the admittance graph 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 C2.46. for this process.




**Figure C2.46.** 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 C2.47.

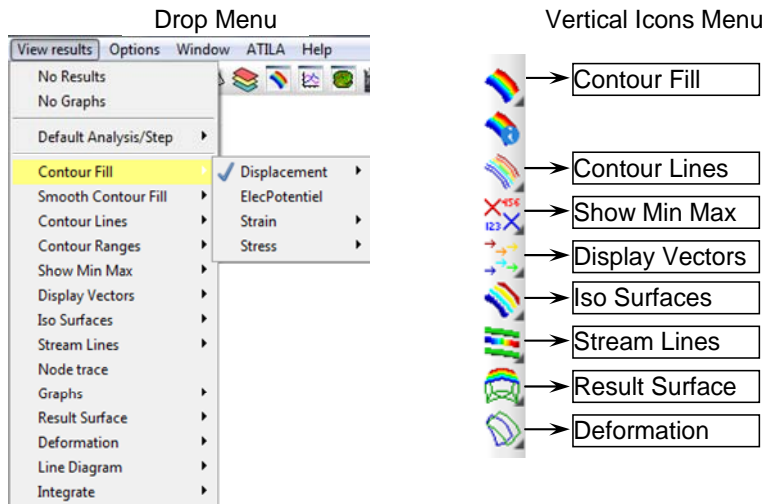


**Figure C2.47.** Admittance Graph

To remove the admittance graphs select the clear graphs icon ()

## 7.2 VIEW RESULTS - DEFORMED SHAPE

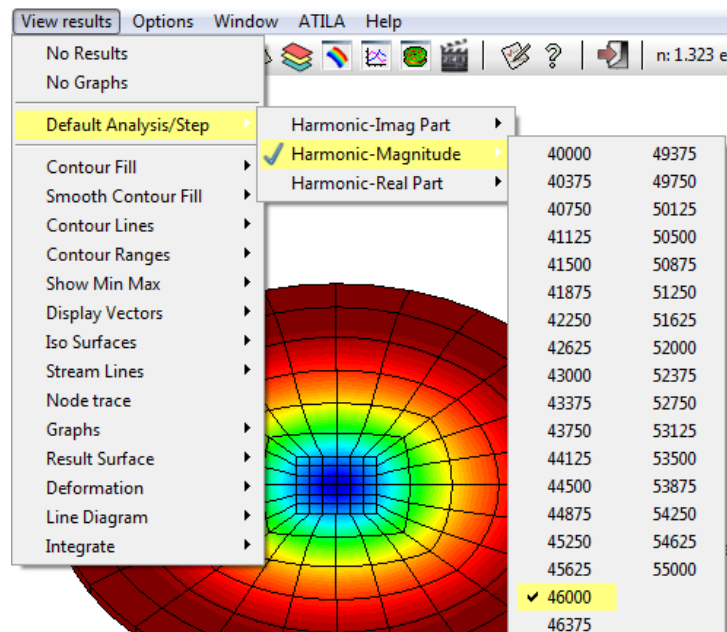
In addition to generate the admittance and impedance values, ATILA provides different type of results for the harmonic simulation. Select **View results** or select the icon vertical menu to select one of the visualization modes for the results as shown in Figure C.2.48.



**Figure C2.48.** View Results display options.

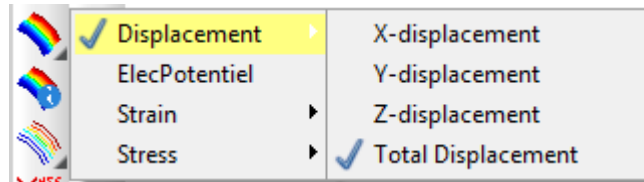
For an harmonic simulation, the results are provided for each frequency. Let's select the frequency of 46000 Hz corresponding to the maximum of admittance and let's display the Total displacement using the Contour Fill visualization.

First select the desired frequency using **View Results** → **Default Analysis/Step**. Then, it is possible to select the Imag Part, the Magnitude or the Real Part for the results. We will choose **Magnitude** for this example and then we pick up **46000** as shown in Figure C2.49.



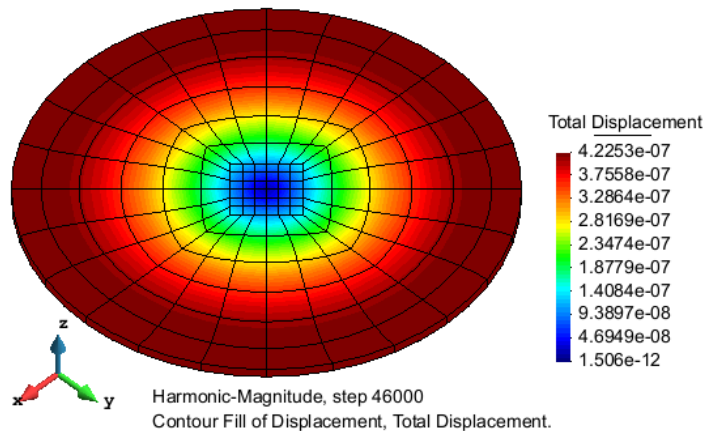
**Figure C2.49.** Selection of the specific frequency to show the results.

Next, we select **Contour Fill** → **Displacement** → **Total Displacement** as shown in Figure C.2.50.



**Figure C2.50.** Selection of Total Displacement using Contour Fill visualization option.

This will retrieve the total displacement in the disc, which in this case correspond to the radial vibration for the first resonance mode as shown in



**Figure C2.51.** Total Displacement in the disc for the frequency of 46000Hz.


### 7.3 ANIMATION

To run the animation follow the steps below.

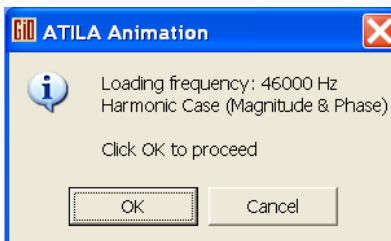
1. Select the frequency to view:

Click **View Results** → **Default Analysis/Step** → **Harmonic-Magnitude** → **Value** (for instance 46000 Hz)

2. Load the animation:

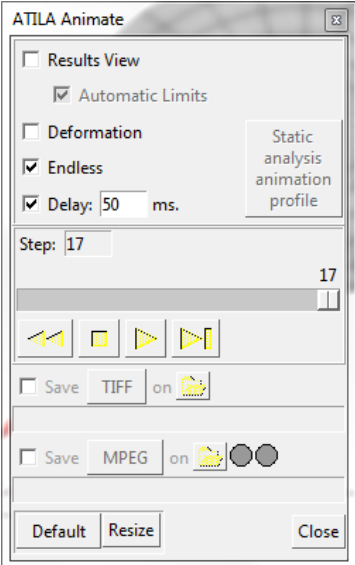
Click (  ) → Atila Animation window appears → Click **OK**

The Atila Animation panel illustrated in Figure C2.52 confirms the frequency selected and activates the loading process for the animation.




**Figure C2.52.** Animation panel.

Immediately after this, the ATILA Animate window will appear. This window allows the user to play the movie, save the movie, etc. The ATILA Animate window is shown in Figure C2.53.







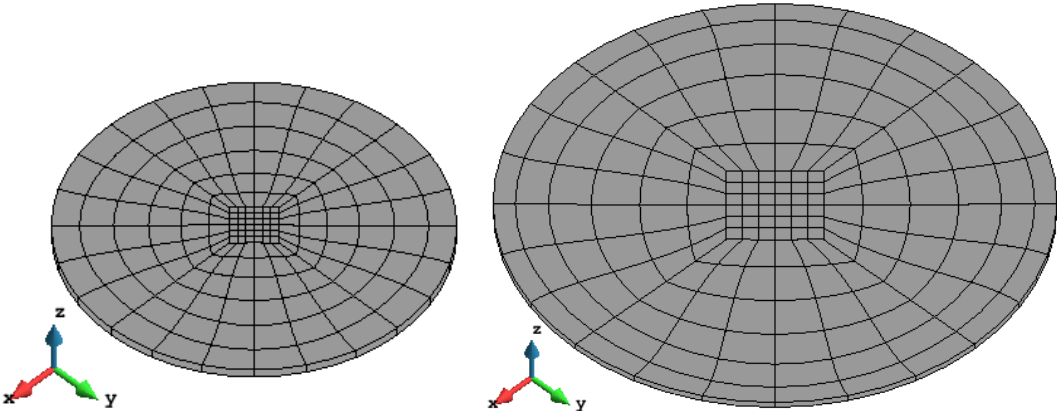
**Figure C2.53.** ATILA Animate window.

3. Select Deformed shape

Once the Animation Control panel has appeared, click (  ) and select a deformation magnitude factor to show the deformed shape. The factor can be adjusted progressively to obtain the best representation of the results.

4. Run the animation

Click (  ) to run the animation, the panels in Figure C2.54. illustrate the animation results at the maximum and minimum harmonic positions. As explained in Case 1 the animation can be stopped at any point by clicking (  ) and stepped through each modal point by clicking (  ) or moving the (  ).



**Figure C2.54.** Different steps observed during the animation.

5. Add visualization options to show specific results:

**EXAMPLE: Total Displacement**

The animation can be stopped and other conditions added to the visualization, such as the displacement. Note: To add or change the visualization the animation must be stopped and restarted after the changes are made. In this example the Total-Displacement will be displayed in the animation.

**Alternative 1: View Total Displacement using the menus**

Click **View Results** → **Contour Fill** → **Displacement** → **X-Displacement**

**Alternative 2: View X-Displacement using the icons**

Click (🌈) → **Displacement** → **X-displacement**

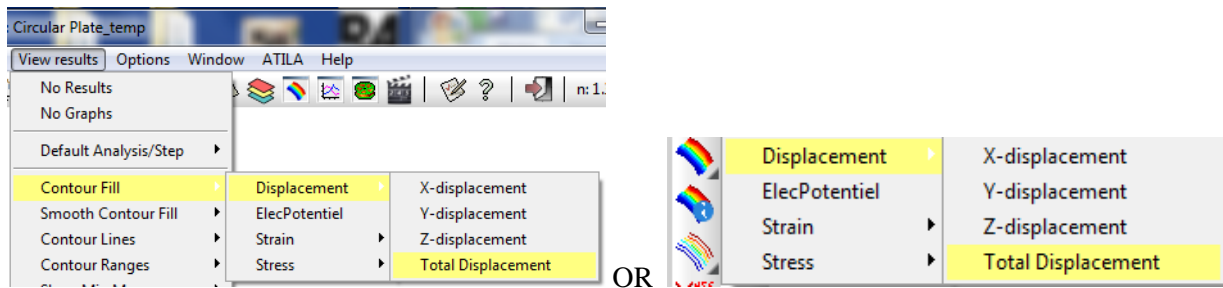


Figure C2.55. Steps to view X-Displacement

The average displacement is shown in the lower right corner of the graphics window, the value is keyed to the corresponding color on the part as illustrated below in Figure C2.56.

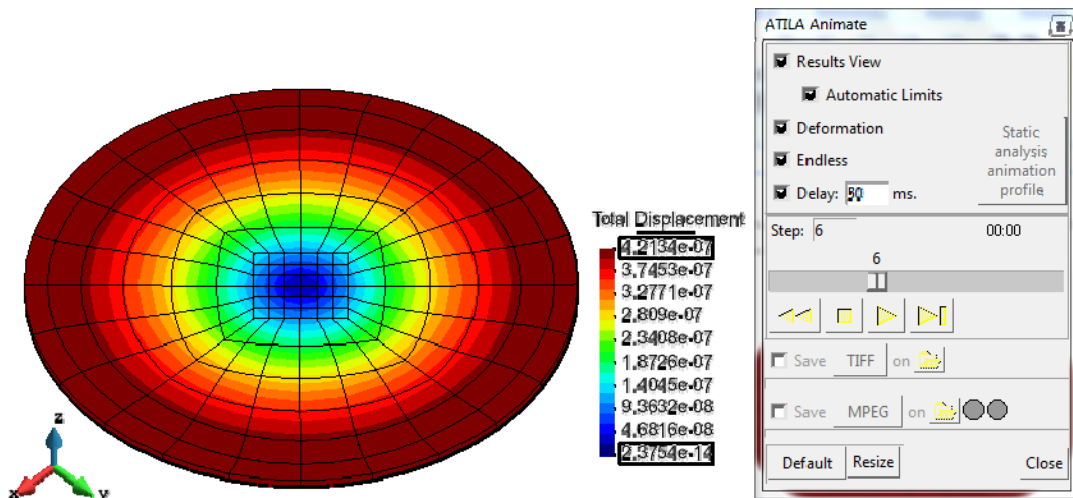


Figure C2.56. Animation of the total displacement for 46,000 Hz

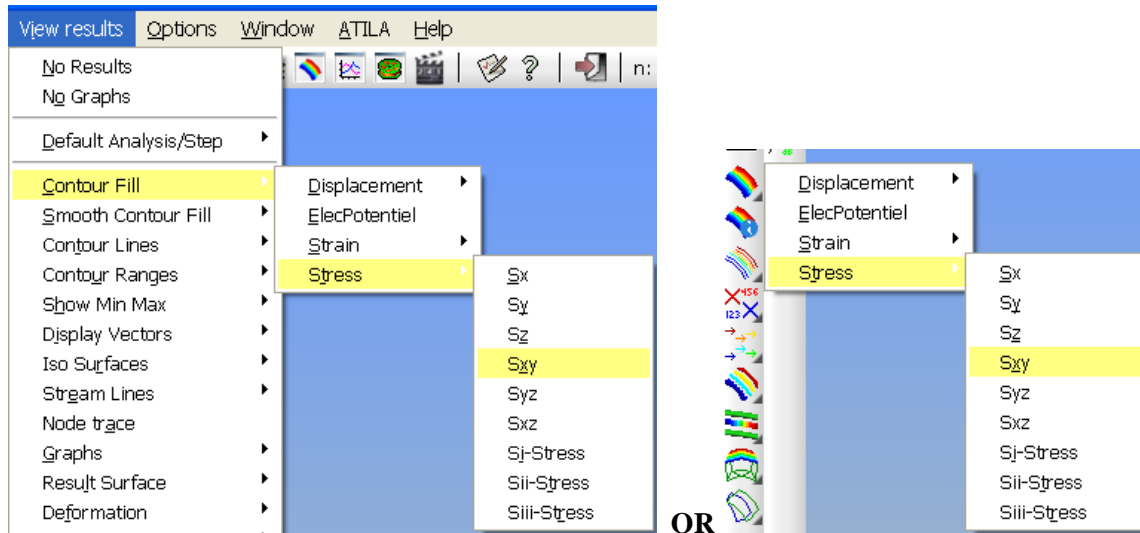
**EXAMPLE: Stress Sxy**

**Alternative 1: View Stress Sxy using the menus**

Click **View Results** → **Contour Fill** → **Stress** → **Sxy**

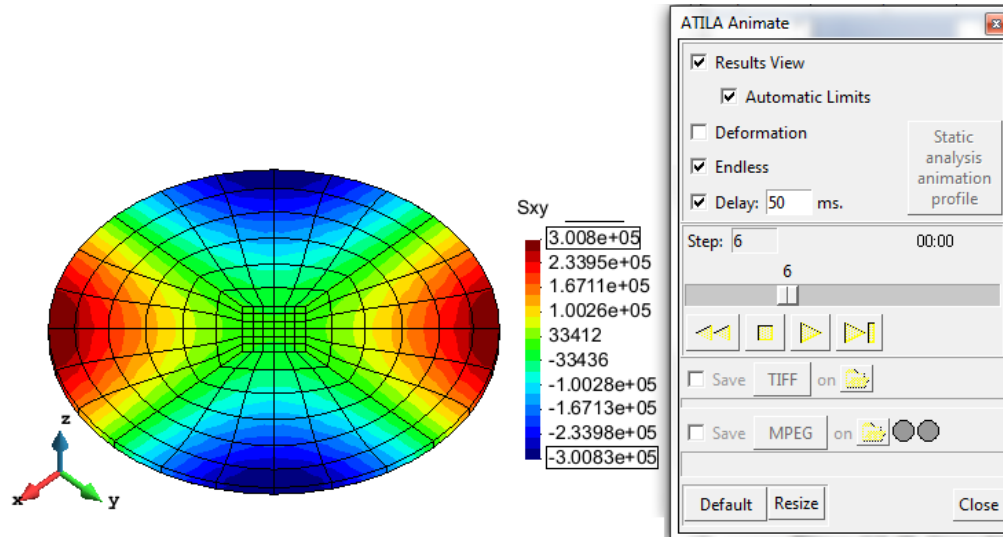
**Alternative 2: View Stress Sxy using the icons**

Click (  ) → **Stress** → **Sxy**



**Figure C2.57.** Steps to visualize Stress on Part

Figure C2.58. illustrates the results of stress visualization as seen in Atila/GID



**Figure C2.58.** Animation of the Sxy for 46,000 Hz.