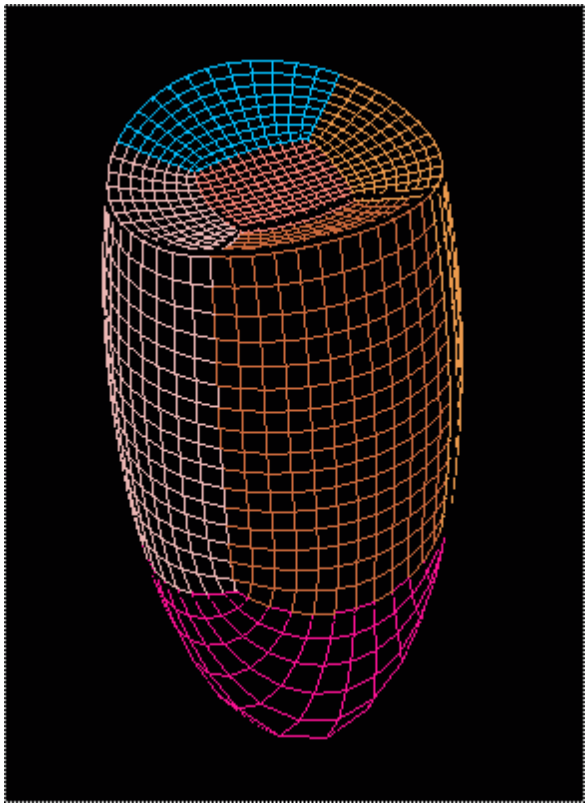
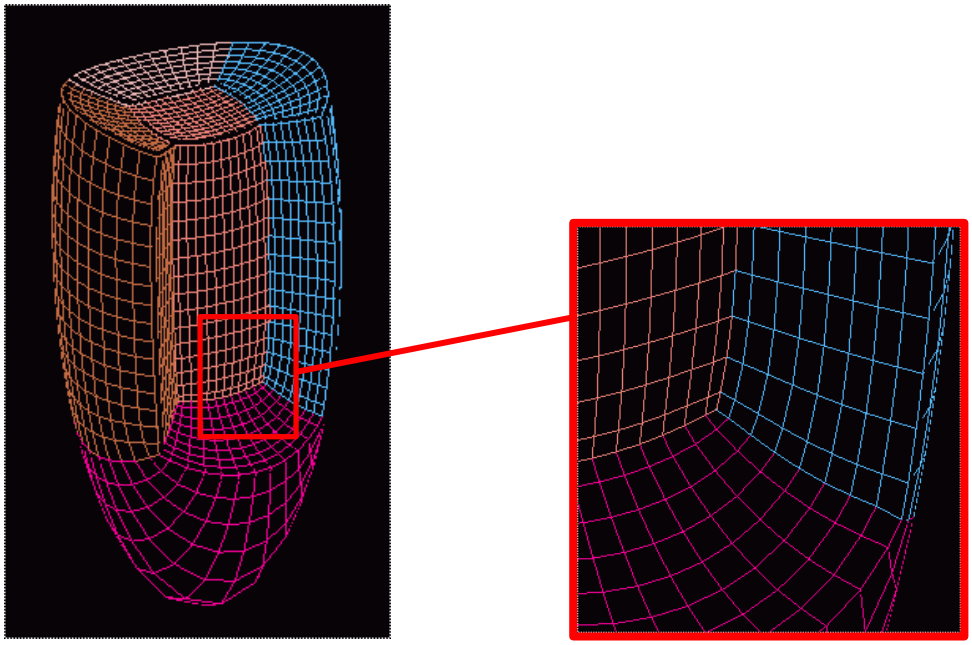
Tutorial 3.2

One of the most important aspects of using **GridPro** is in creating topology for complex geometries. Locating the corners and edges of the topology, particularly in 3D, can be time consuming and tedious without a tool that can automate the process. For this reason, the **Cut-Plane** in **GridPro** allows the user to speed up topology creation while increasing the grid generation efficiency. In this section, we will learn how to use the **Cut-Plane** and how to better visualize our mesh by using grid trimming commands. A brief example on how to use **GridPro** for CFD parametric design analysis will also be covered.

What You Will Create



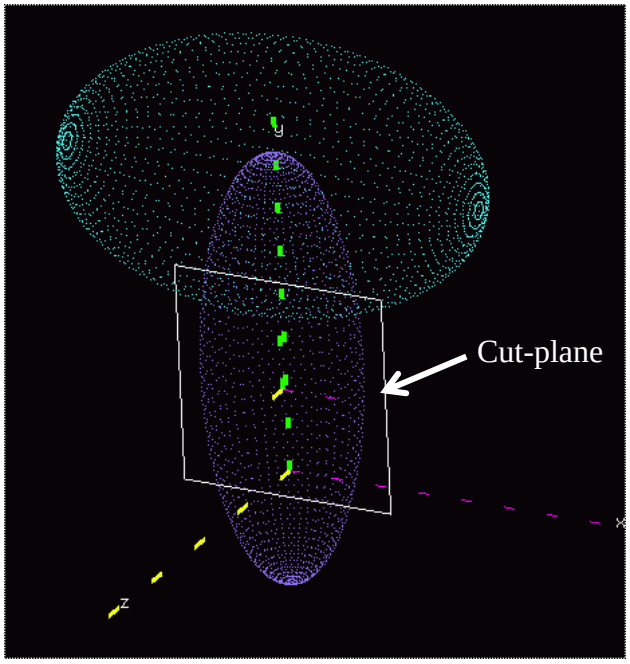


What You Will Learn

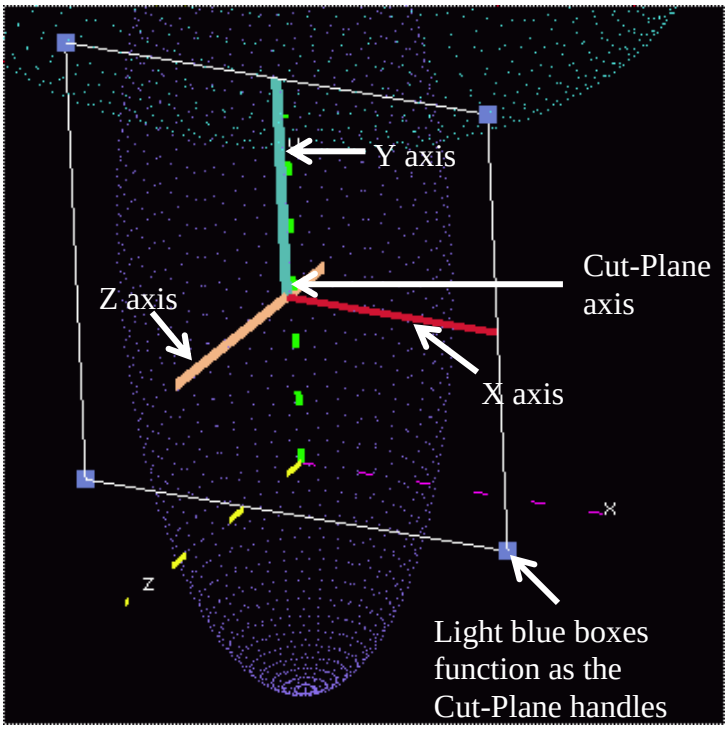
- An introduction to the **Cut-Plane**.
- Copying topology to the **Cut-Plane**.
- Introduction to advanced grid viewing using functions to trim blocks and sheets.
- An introduction to creating a grid for parametric design analysis

Step 1 Introduction to the Cut-Plane

The **Cut-Plane** is the most important feature inside **GridPro** that will be used for topology creation. You will find that when the mesh increases in complexity, you will become more dependent on using the **Cut-Plane**. Load the **Tutorial_3.2.fra** file and turn off shading and turn on points in the **STYLE** sub-command panel. Rotate the surfaces to get a nearly isometric view as in the picture below. Notice that the **Cut-Plane** is colored as a 2D white rectangle.

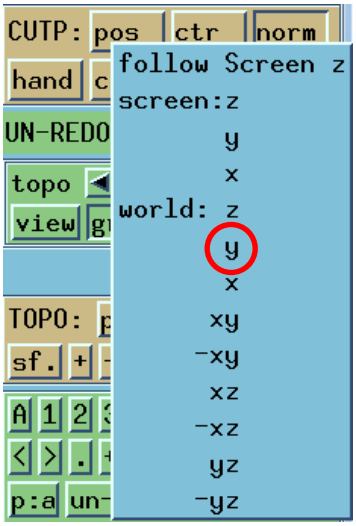


Go to the **CUT-P** sub-command panel and click on the hand button and notice that light blue boxes appear at the **Cut-Plane** corners. A three colored axis, **red** as the **x-axis**, **light blue** as the **y-axis** and **light brown** as the **z-axis** is also displayed. The boxes function as the **Cut-Plane** handles and the axes are used for translation and rotation. See the picture below.

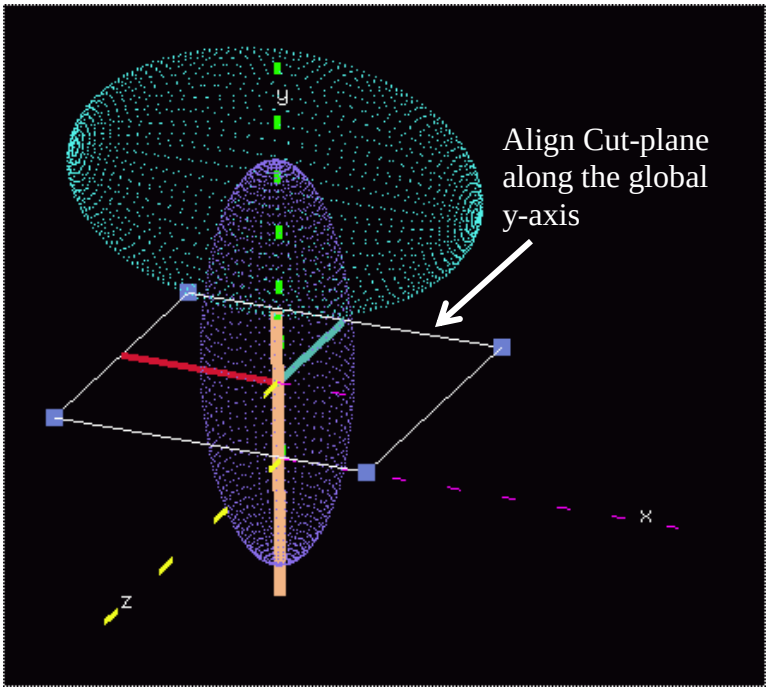


Make the **Cut-Plane** larger by holding down the left mouse button on one of the squares and dragging it outward. Since the stem of the mushroom will be meshed, orient the **Cut-Plane** perpendicular to the world (global) **y-axis** by going to the **CUT-P** sub-command panel and clicking on the 

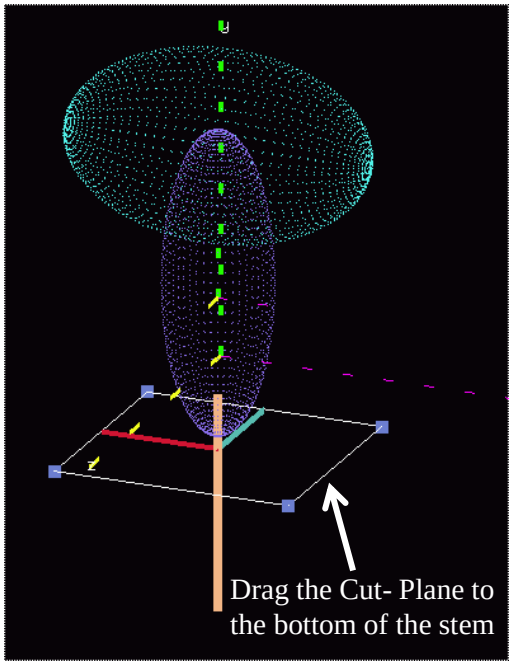
button and choosing **world:y** from the sub-menu.



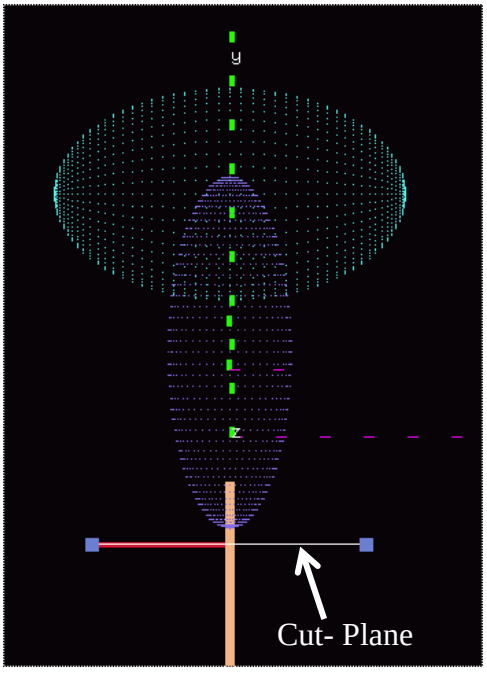
The Cut-Plane will snap perpendicular to the global y-axis.



Position the **Cut-Plane** at the bottom of the stem by grabbing onto its **z-axis** and dragging it downward. When you position the cursor over the **z-axis,** a double arrow in a circular pattern will appear, at this point, hold down the left mouse button and drag downward.

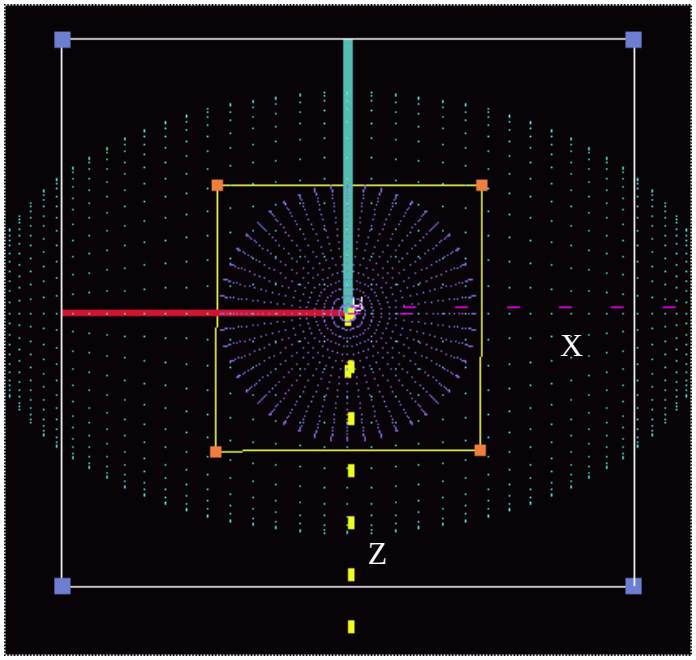


Make sure the **Cut-Plane** is at the bottom of the stem by snapping the geometry to the global XY axis as in the picture below.



Step 2 Creating the Wrap

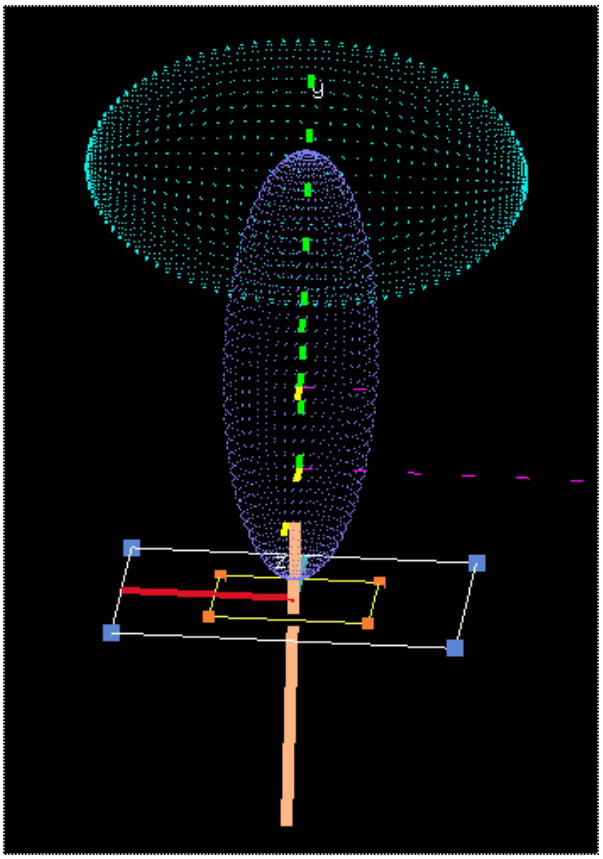
Now that the **Cut-Plane** is in the proper position we are ready to create the wrap. Snap the grid on the global XZ axis and place four corners on the **Cut-Plane** around the stem and connect them with edges.



Cut-Plane

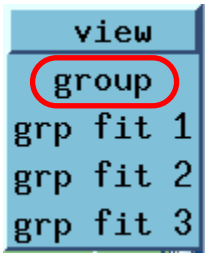
The Cut-Plane is not a surface. Internally the code recognizes the plane as infinite but for the purposes of constructing topology, it is viewed as having four discrete edges. By design, the topology corners can be placed anywhere on the infinite plane.

Rotate the geometry in 3D space again so that the topology can be seen in relationship to the stem and head intersection as in the picture below.

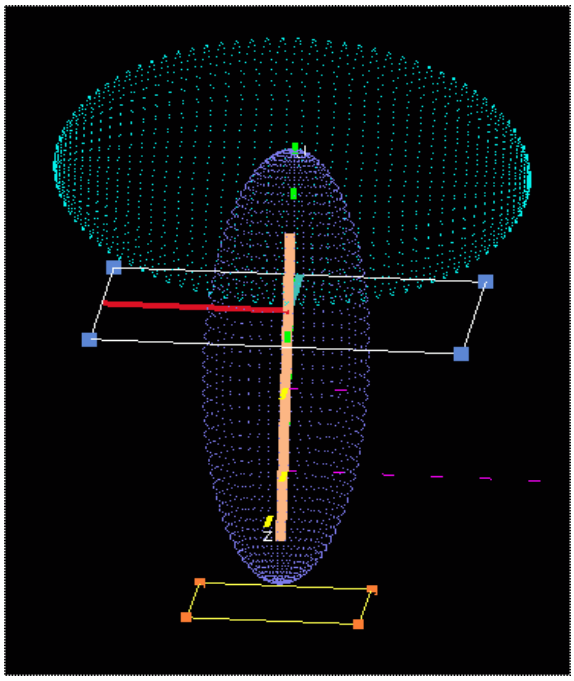


At this point, a rectangular shaped topology needs to be created around the stem by copying the 2D topology to the intersection of the surfaces. To ensure that the topology remains uniform, group the **Cut-Plane** to the center of the topology by clicking on 

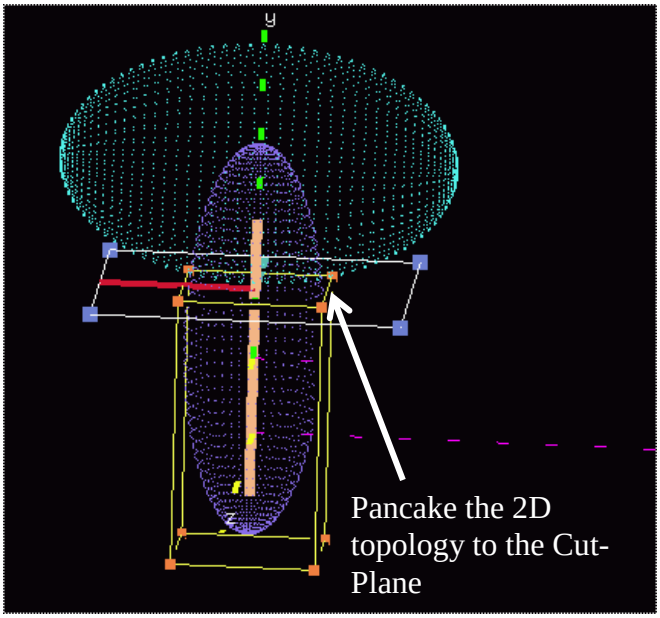
and choosing **group fit** from the sub-menu.



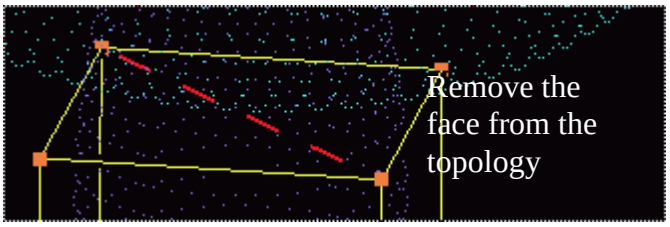
Group fit will snap the center of the **Cut-Plane** to the topology center of gravity. Now grab onto the **Cut-Plane z-axis,** colored in light brown, by left clicking and holding down the mouse button. Drag it upward along the global **y-axis** toward the point of intersection. If the plane moves off of the global **y-axis**, snap it back onto the center with another **group fit**. The plane should be positioned as shown in the picture below.



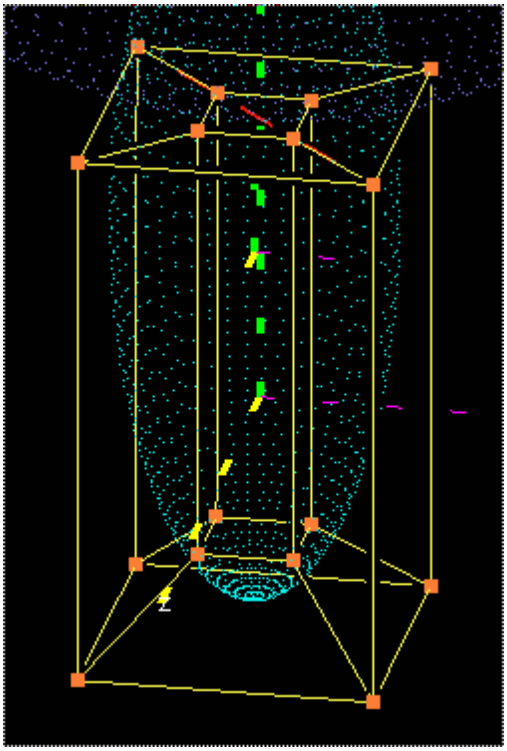
Now that the plane is in position, we can create the outer 3D topology by copying the 2D topology to the **Cut-Plane**. First, the topology must be added to a group. Add the loop of four corners to **Group 1** (**Note:** topology cannot be copied unless it is added to a group and the group is active). Go to the **TOPO** sub-command panel, click on  to pull down the menu options and chose **+drop back edges** which will copy the topology loop to the **Cut-Plane** with its connected edges.



The new topology will remain in **Group 1**. Proceed to create the wrap by removing the face by pressing and holding down F on the keyboard while clicking two diagonal corners.

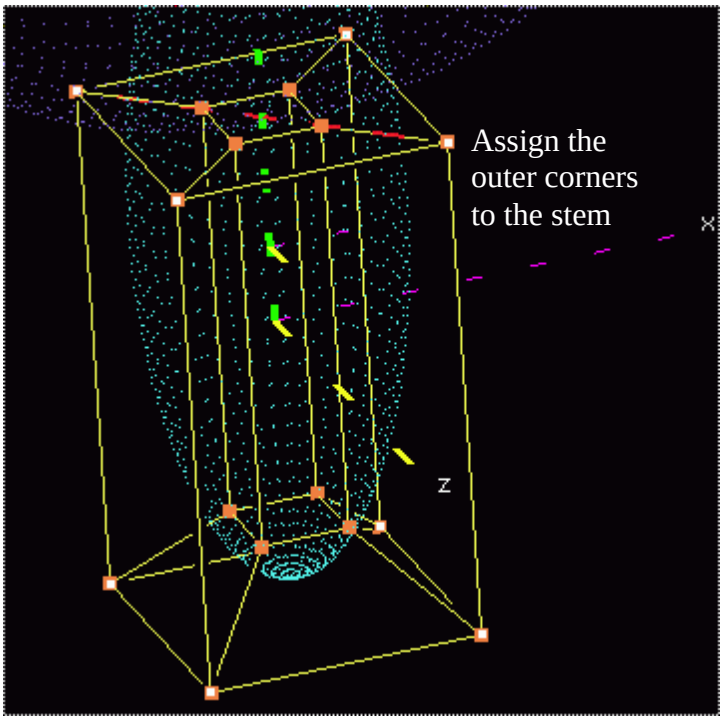


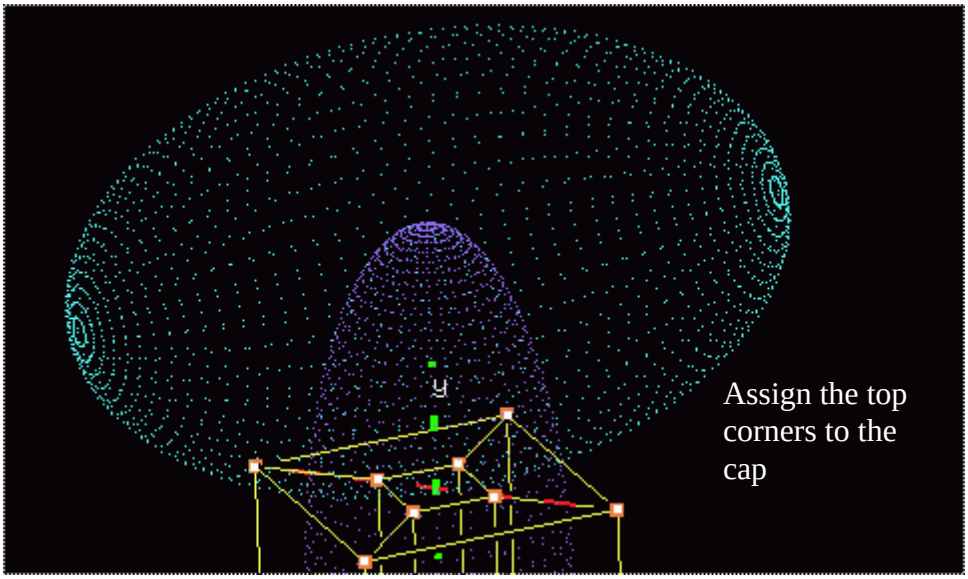
The topology is now prepared for wrapping. Go to the **TOPO** sub-command panel and wrap the topology **25% smaller** as in the picture below.



Step 3 Surface Assignments

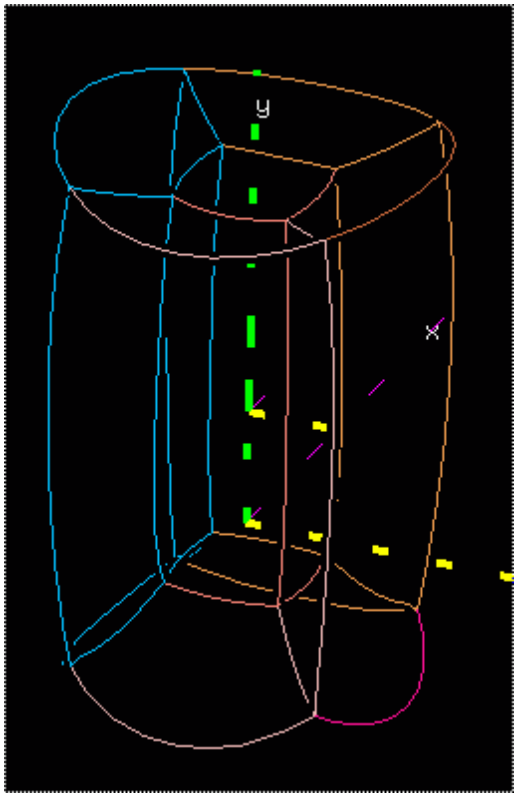
Make sure that the stem is the current surface and is in light blue. Assign the outer corners of the topology to the stem and the top corners to the cap as we did in **Tutorial 3.1.**



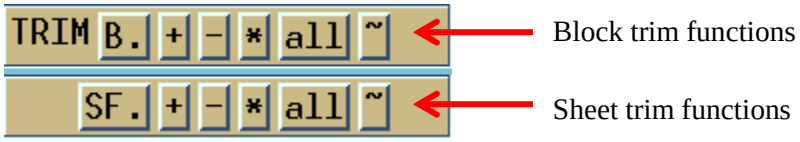


Step 4 Introduction to Advanced Grid Viewing

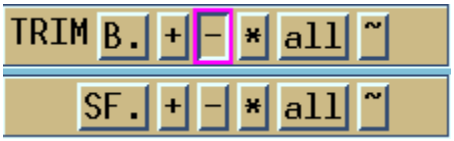
GridPro's advanced grid viewing functions contain techniques that lead the market for inspecting and visualizing the final mesh. Run the mesh generator and display the results in the **Grid Viewer**. All of the blocks are initially active for viewing as in the picture below.



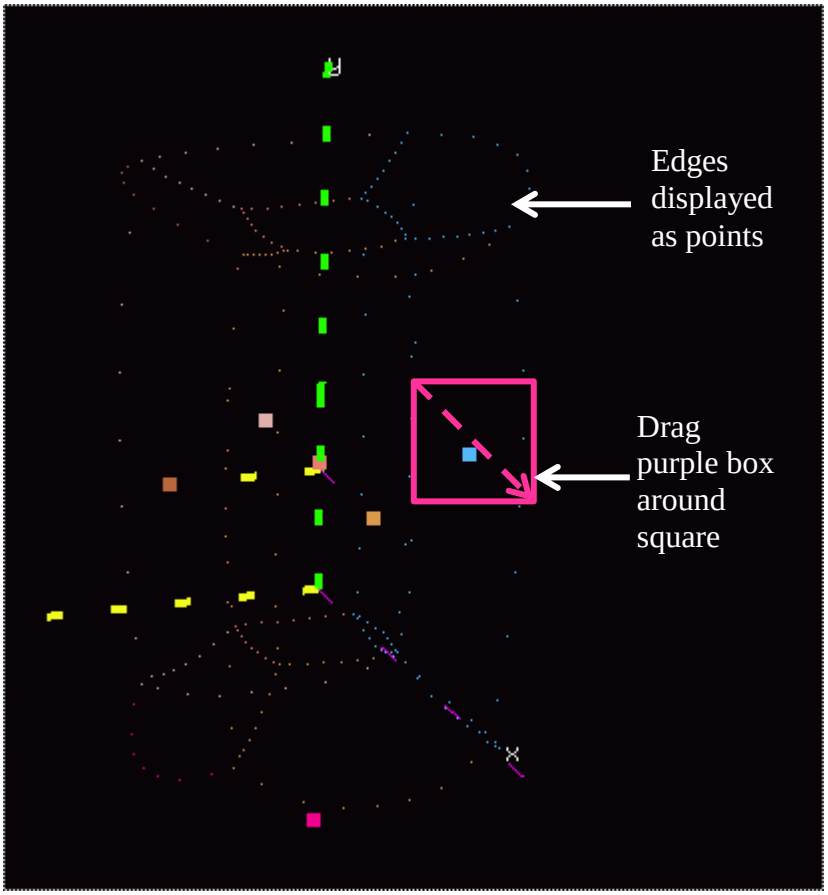
In the previous tutorials, to display the grid, we picked on one of the block edges that created a sheet cutting through the volume. To display the inside of the mesh volume, we can also remove a block or part of a sheet from the view by trimming it away. Go to the **TRIM** sub-command panel and, as you can see, the functions resemble that used in the **TOPO** subcommand panel. The first row contains the command functions for the **block**, and the second row the command functions for the **sheet**.



Click on  the in the block menu to inactivate one of the blocks

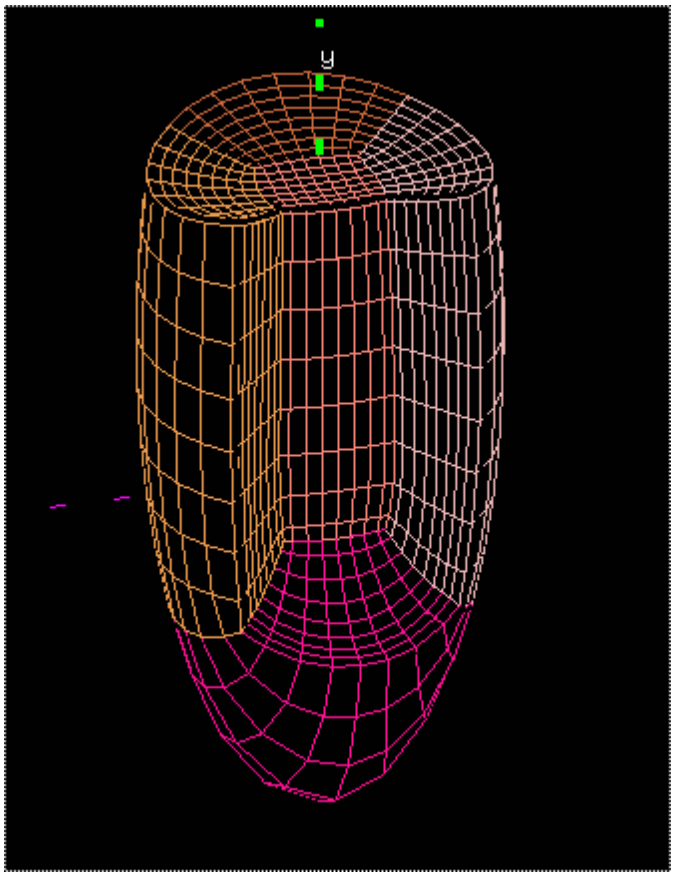


and notice that the block edges are now displayed in points and a small square appears at each block's center. Right click the mouse button and drag a purple box around one of these boxes as in the picture below.



Now go to the **MAKE SHEET** sub-command panel and click on 

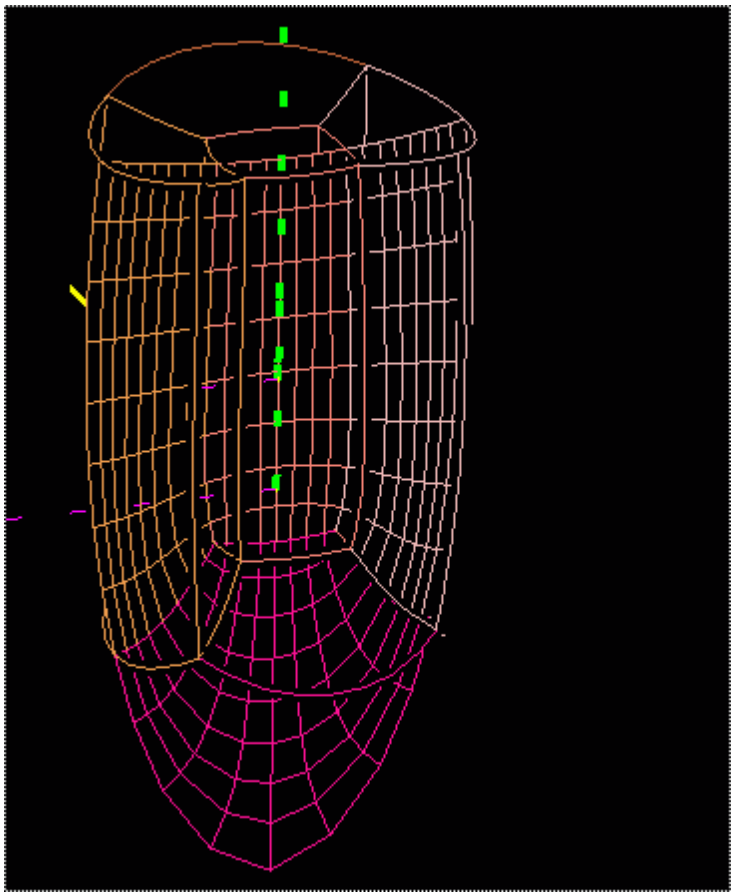
and the grid will be displayed. Rotate the grid so that a better view of the inner part of the volume mesh can be seen. Notice that the mesh for the block that was removed is not displayed.



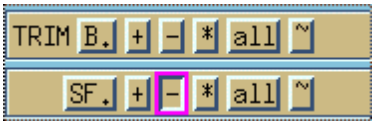
Now let's see how to trim a sheet. Delete the current sheet by going to the **CUR** sub-command panel and clicking on 

and make a sheet across the volume as we did in previous tutorials (**Note:** the 

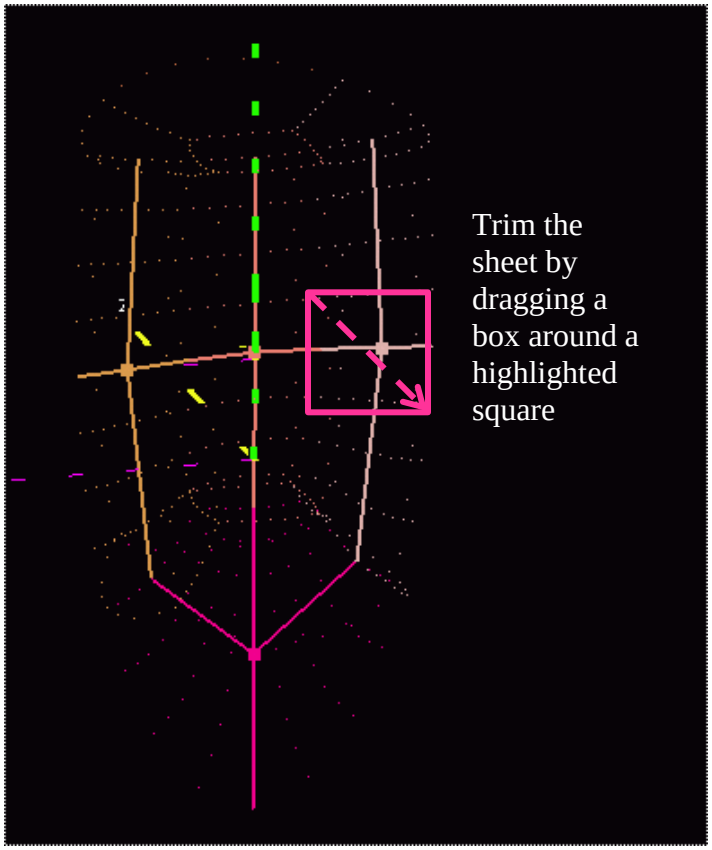
command in the **MAKE SHEET** sub-command panel makes a sheet by painting all faces that have an active block on one side and an inactive block or boundary on the other).



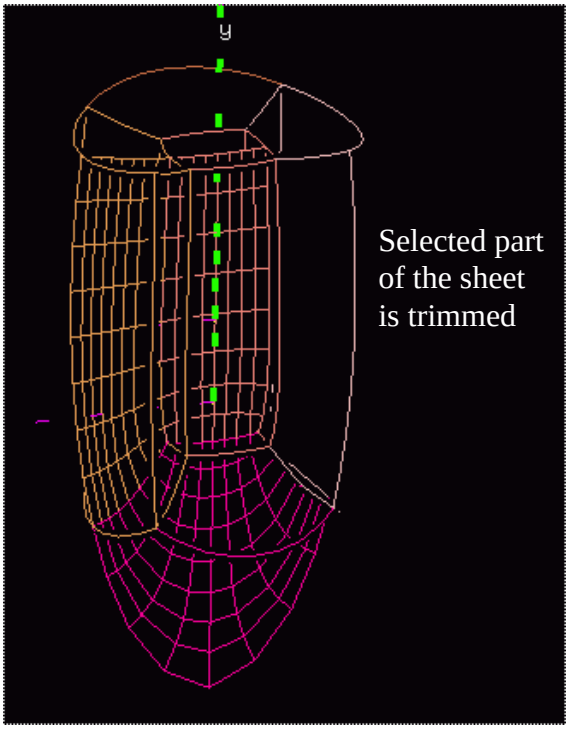
Trim the sheet by clicking on  in the sheet menu of the **TRIM** subcommand panel



and drag a purple box around a highlighted square at the center of the block as in the picture below. Notice that the skeletons of the blocks are also displayed and show the center of the blocks plus the links to the center of the faces.



Trim the sheet by dragging a box around a highlighted square



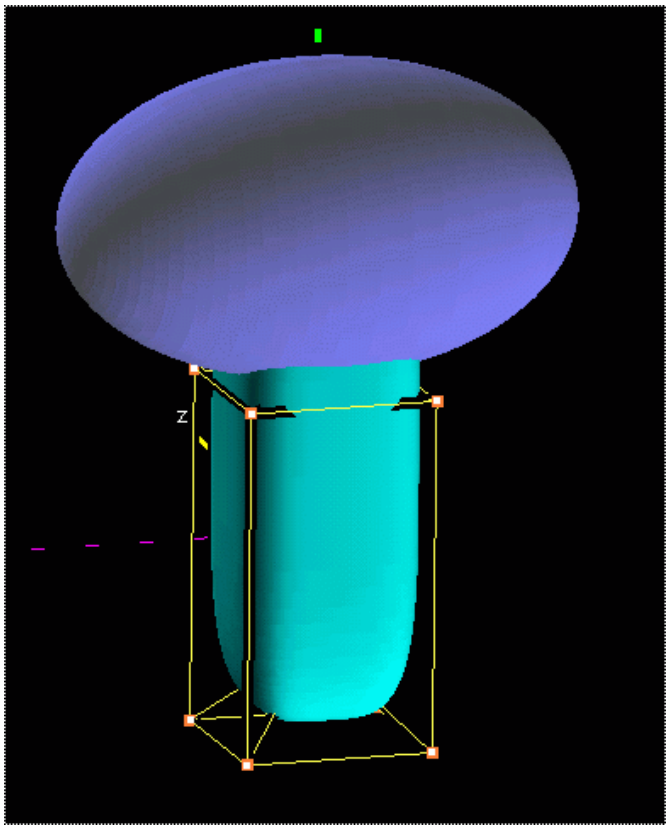
Selected part of the sheet is trimmed

The trimmed sheet or block can be displayed again by clicking on the 

button.

Step 5 Introduction to Creating a Grid for Parametric Design Analysis

One of the most useful features in **GridPro** is that once a topology for a given geometry is created, it can be reused again to mesh a geometry of a similar shape. This technique can be very useful for parametric CFD design analysis when an engineer needs to optimize the design of a part or an assembly of parts. Let's demonstrate this function by changing the shape of the stem and regenerating the grid using the topology that has already been created. Go back to the **Topology Builder** and choose the stem as the current surface. Pull down the **surf** menu at the top and **reload current** surface to view the stem **set surface parameters** pop-up menu. Change the **power** from 2 to 4, hit **apply** and close the pop-up menu. Get a better view of the stem by going to the **STYLE** sub-command menu and choose shade. The new surface should look like the picture below.



Start the gridding process and go back to the **Grid Viewer** to look at the mesh.