Using FreeCAD for finite-element mesh generation

Stanley Humphries, Ph.D.

Field Precision LLC
E mail: techinfo@fieldp.com
Internet: https://www.fieldp.com

In a previous article, I discussed the symbiosis between our 2D mesh generator) and **LibreCAD** (an open-source 2D CAD program). There is an even closer relationship between **MetaMesh** (our 3D mesh generator) and **FreeCAD**, an open-source program for creating complex three-dimensional objects. Although you can achieve much with the constructive solid geometry operations applied in **MetaMesh** and the interactive **Geometer** preprocessor, the procedures may be difficult and time consuming when applied to a complex parts with curves, chamfers, fillets and other features. In these cases, it is often more efficient to create the part in **FreeCAD** and then to port it to **MetaMesh** as a stereo-lithography (STL) object.

FreeCAD has several advantages over commercial packages like **Pro Engineer**, **Solidworks**, **Inventor** and **AutoCAD**.

1. The outstanding advantage is that it is free and open (as opposed to costing thousands of dollars with strict license control).

2. The available documentation is quite good.

3. Its use of parts and their placement in coordinate systems is closely related to the logic of **Geometer** and **MetaMesh**.

4. There are good resources for creating and exporting STL meshes.

5. I found it relatively easy to learn following online tutorials.

With regard to the latter, there are some important concepts to master and several potential sources of confusion for new users. With that in mind, I thought it would be helpful to write a tutorial showing some basics of **FreeCAD** usage with tips on integration with **Geometer** and **MetaMesh**.

To begin, install and run **FreeCAD**. In the *Start* workbench, click on *Create new...* to open the display of Figure 1. To interact with **MetaMesh**, we are going to create a single part that will be exported as a stereo-lithograpy file. We need not worry about the location of the part in the **FreeCAD** coordinate space because we can adjust the position and orientation of the part in **Geometer** and **Metamesh**. Following Fig. 1, go to the *Part design* workbench and click the tool to start a new part. Click the *Model* tab on the left to see the organization. Right click on the *Part* and rename it *Cathode*.

Parts may include several *Bodies* joined together by logical operations. For this example, we ll need a single *Body*. Click the tool to add a *Body* to the current *Part*. The body will be a flattened pipe. As in **Geometer**, we make an *Outline* (*Sketch*) in a reference plane and then *Extrude* (*Pad*) it. Click the tool to start a *Sketch*. Choose *x-y* as the reference axis. *FreeCAD* switches to the *Sketcher* workbench (Figure 2), showing an appropriate set

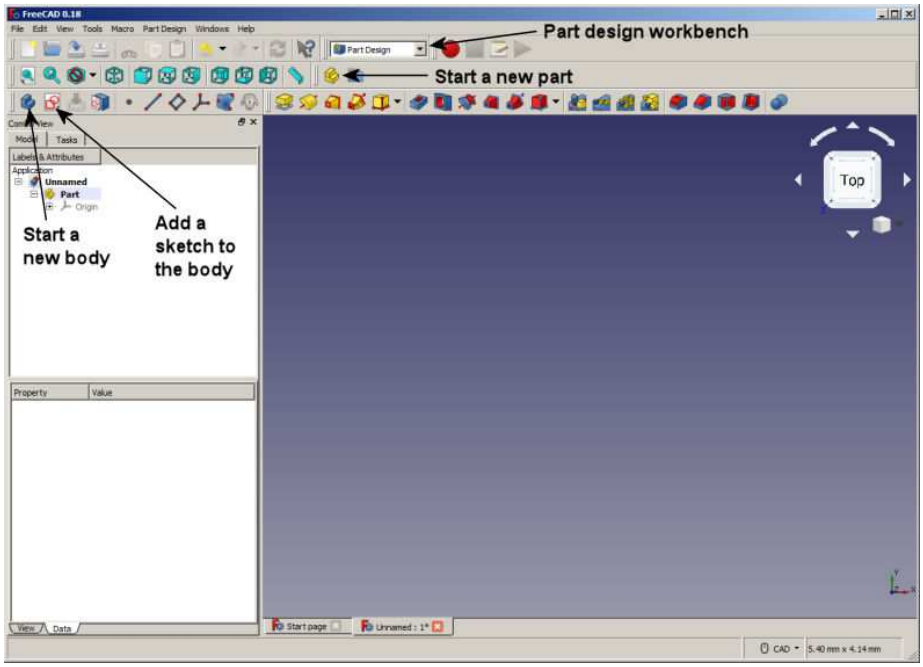

Figure 1: FreeCAD, starting a *Part* and adding a *Body*.

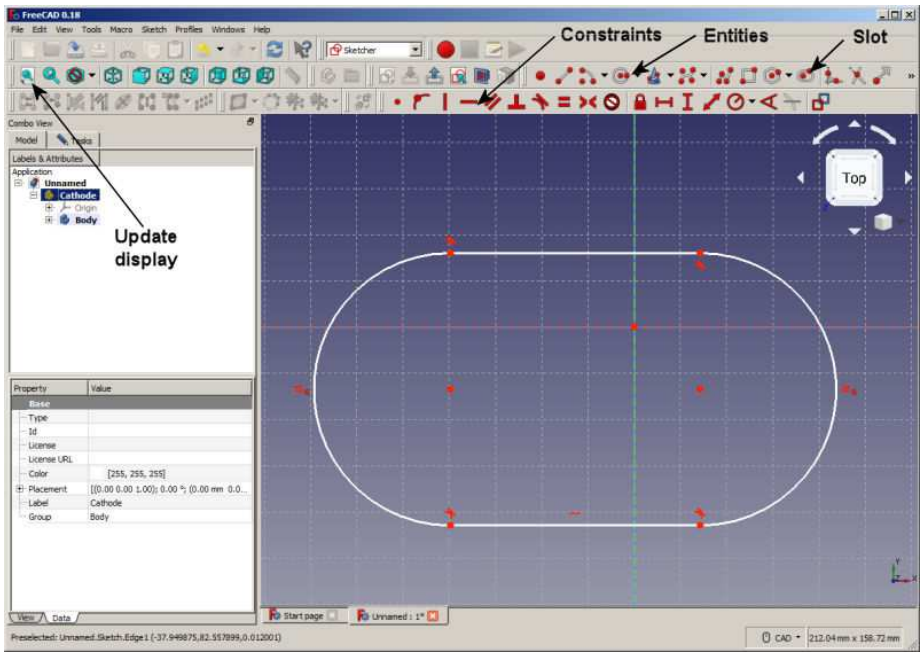

Figure 2: FreeCAD, creating on outline in the Sketcher workbench.

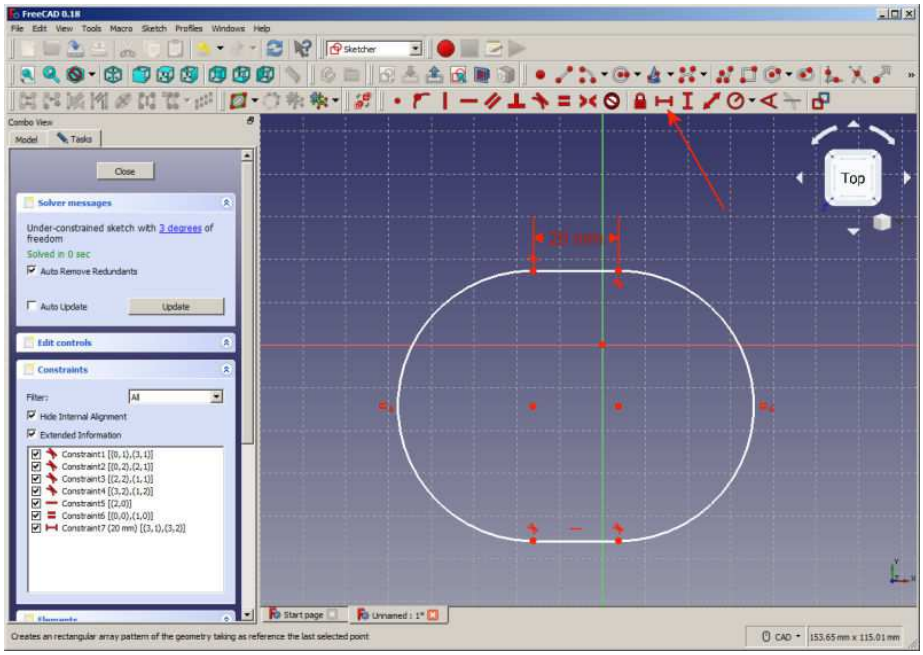

Figure 3: FreeCAD, adding a line length constraint in the Sketcher workbench.

of tools. The information will be associated with the *Body* in the tree view on the left.

It is important to recognize that the sketching procedure in **FreeCAD** differs from the standard method used in **Geometer** outlines and most 2D CAD programs. There is no grid-snap mode. Instead, the idea is to create a rough representation of the entity and then to apply constraints until it is fully defined. An example is the best way to see how this works. Click on the *Slot* model and use mouse clicks to create a slot of any size and position. Figure 2 shows the display. Points are shown as circles and the other red symbols are constraints. The default constraints for a slot have the following meanings: 1) the bottom line is horizontal, 2) the arcs and lines are tangential at the four meeting points and 3) the two arcs have the same radii.

We need to add constraints so that the sketch is specified unambiguously. Select the top horizontal line, click the horizontal length constraint and choose a value of 20.0 mm (Figure 3). Note the addition to the constraint list at the left. Now, select a top intersection point and hold down the control key to add a bottom intersection point to the selection. Then, apply a horizontal distance constraint (Figure 4) and specify the dimension 20 mm. Select 1) a left intersection point, 2) a right intersection point and 3) the vertical axis (green line) and then click on the symmetry constraint to center the shape in the coordinate system. Follow similar operations to place

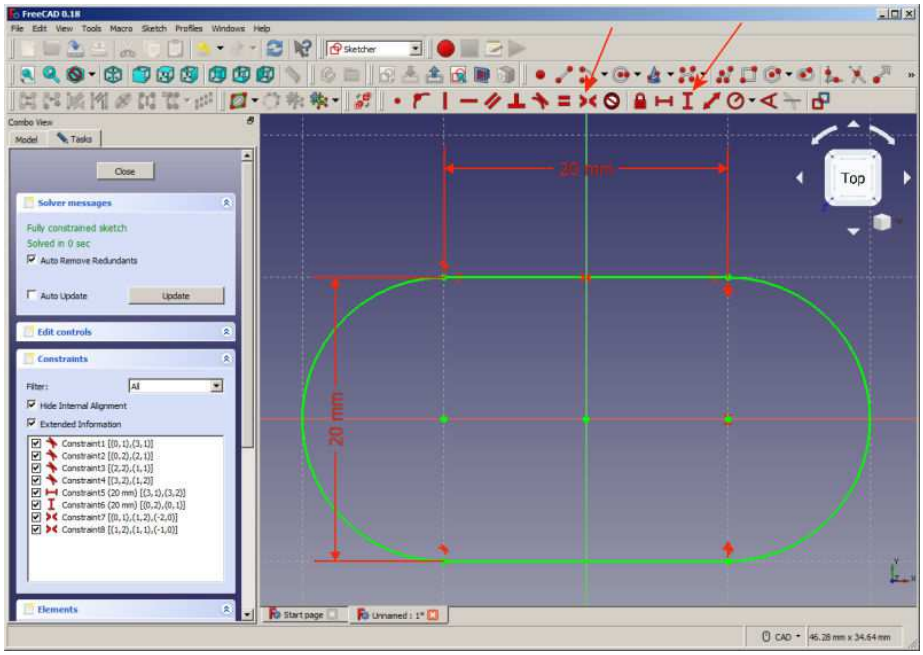

Figure 4: FreeCAD, fully constrained inner edge of the pipe.

the shape symmetrically in the vertical direction. With this addition, the outline turns green, meaning that it is fully constrained (*i.e.*, unambiguously defined). To complete the sketch, use similar operations to add the outer pipe wall as shown in Figure 5. Close the completed sketch.

Back in the *Part design* workbench, switch to an isometric view (Figure 6). The sketch outline is projected in the *x-y* plane. Click on the *Pad* tool and choose a value of 20.0 mm to create the solid body of Figure 6. Holding the control key, select the four vectors comprising the inner edge of the top face. Click on the *Chamfer* tool and specify a value 5.0 mm. Finally, select the vectors of the outer edge of the top face and add a 2.4 mm fillet. Figure 7 shows the final result. Save the work as *Cathode.FCStd*.

Finally, we create a stereo lithography mesh to export to **MetaMesh**. **FreeCAD** can export these files in either binary (*STL*) or text (*AST*) format. For compatibility with **Metamesh** and **Geometer**, use the text format. You could export the current part directly, but the default mesh procedure makes poorly-formed, acute triangular facets. Instead, we will create a well-formed mesh of the body surface within **FreeCAD** and then export the result. Transfer to the *Mesh design* workbench. Select the body from the *Model* tree list on the left. From the menu at the top, choose *Meshes/Create mesh from shape* to display the options shown in Figure 8. Pick *NetGen/Fine* and click *OK*. Figure 9 shows the resulting well-formed mesh. Select *Body (meshed)* from the tree list and then choose *File/Export* from the menu.

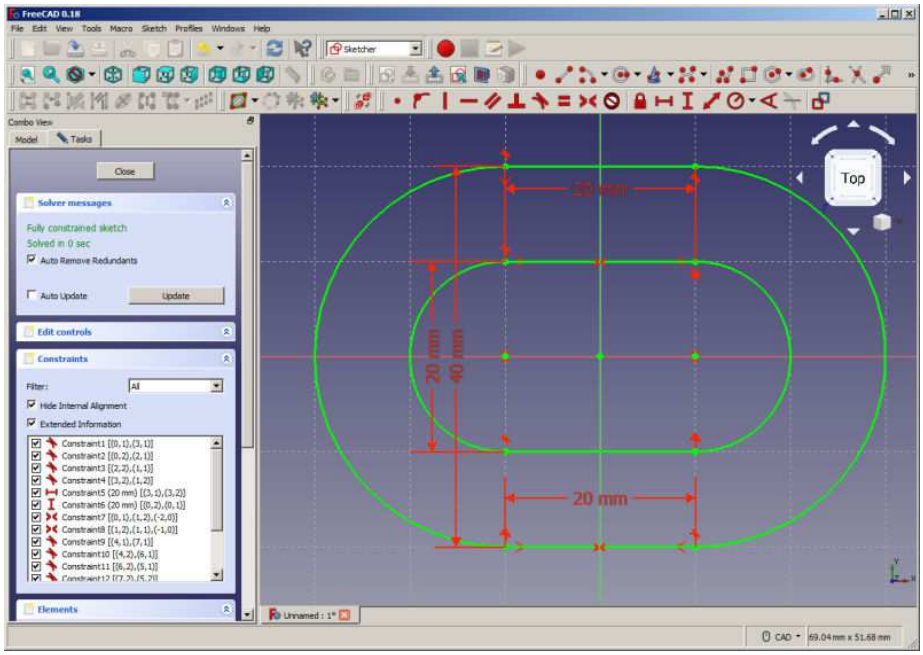

Figure 5: FreeCAD, completed sketch of the pipe.

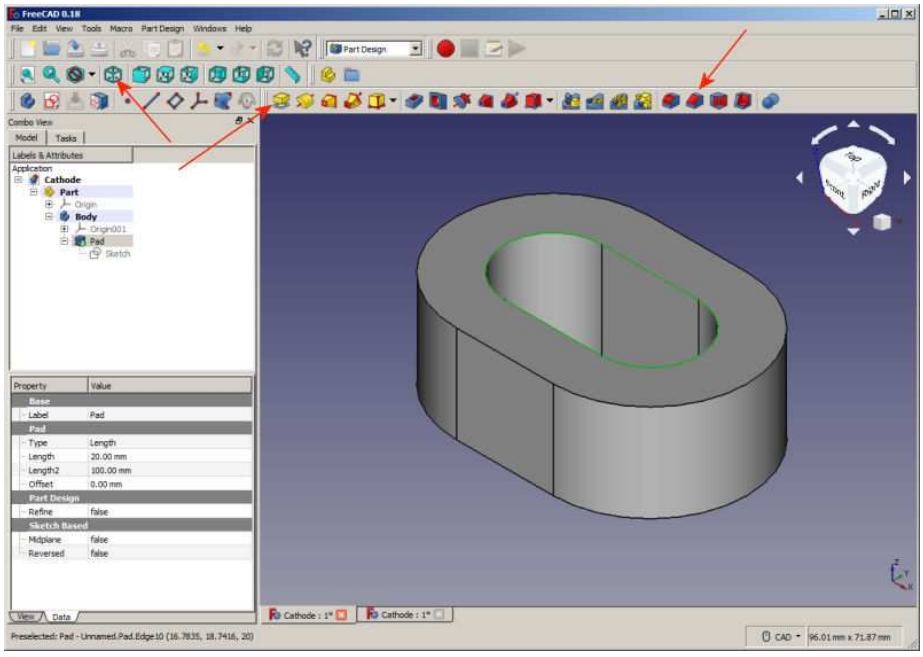

Figure 6: FreeCAD, extrusion of the sketch to create a pipe. Inner edge of top face selected.

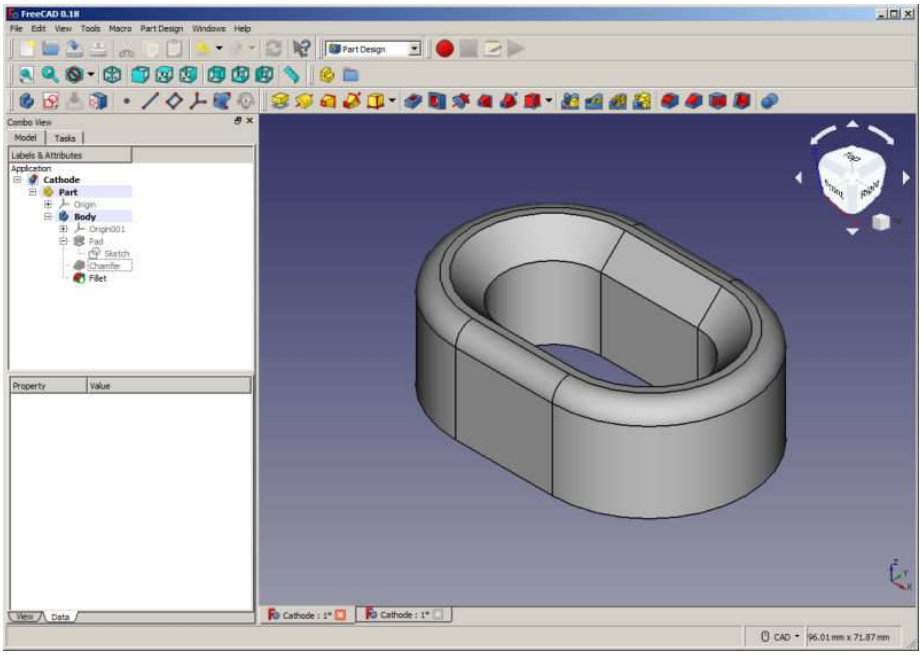

Figure 7: FreeCAD, addition of a chamfer and fillet to one end of the pipe.

Choose the option *STL mesh* and give it the name *Cathode.AST* to signal text format.

Because **FreeCAD** is the subject of this article, I will mention only briefly how to use the stereo-lithography file in **MetaMesh**. The downloadable file (https://www.fieldp.com/fpblog/examples/SheetGun.MIN) illustrates several techniques:

1. Loading *Cathode.AST*, shifting it in the solution space and fitting elements to the surface.

2. Combining the STL model with a standard **MetaMesh** shape (an extrusion) to create a emission surface within the focusing electrode (the part named *EMITTER*).

3. I defined another **FreeCAD** model to represent the anode and flipped it around for the proper orientation in the solution space.

4. The *COAT* command was used in the *EMITTER* specification to create a set of nodes to define emission facets for an **OmniTrak** solution.

Figure 10 shows the result. In conclusion, **FreeCAD** provides a path to design precision 3D objects of any shape with a moderate amount of work. The program creates good quality STL meshes that can be imported reliably by **MetaMesh**.

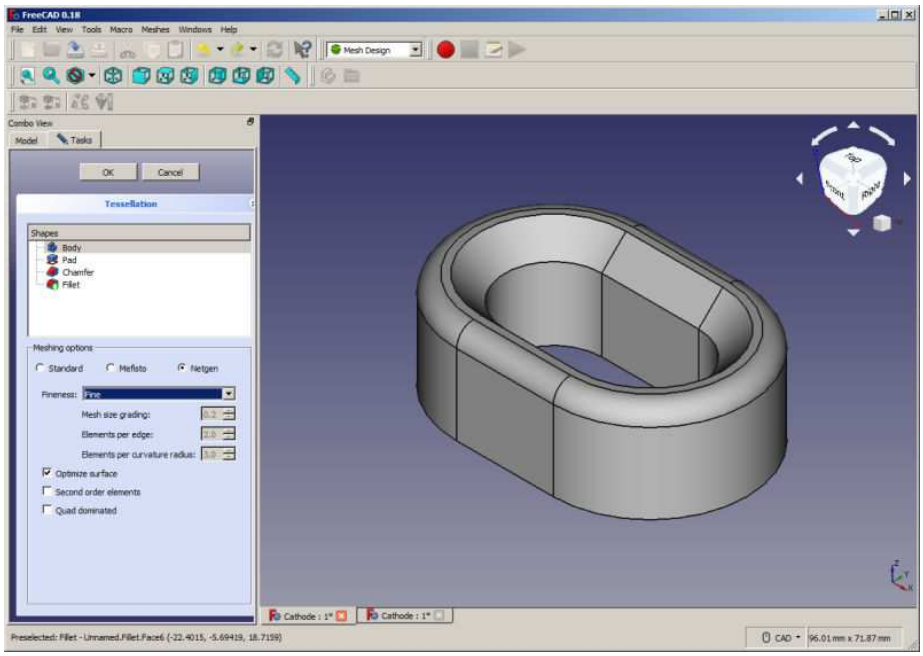

Figure 8: FreeCAD, mesh creation options.

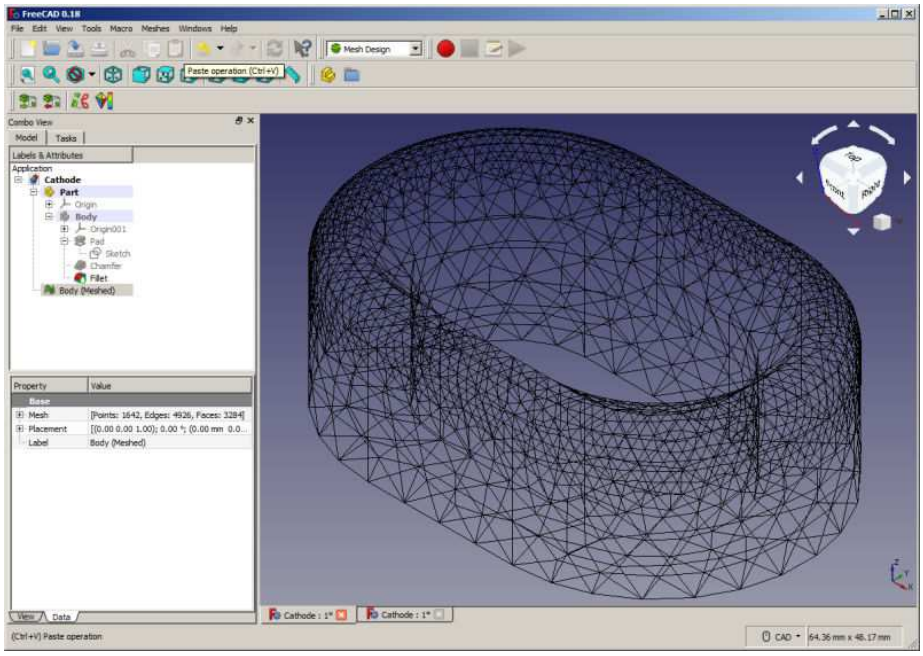

Figure 9: FreeCAD, mesh ready for export.

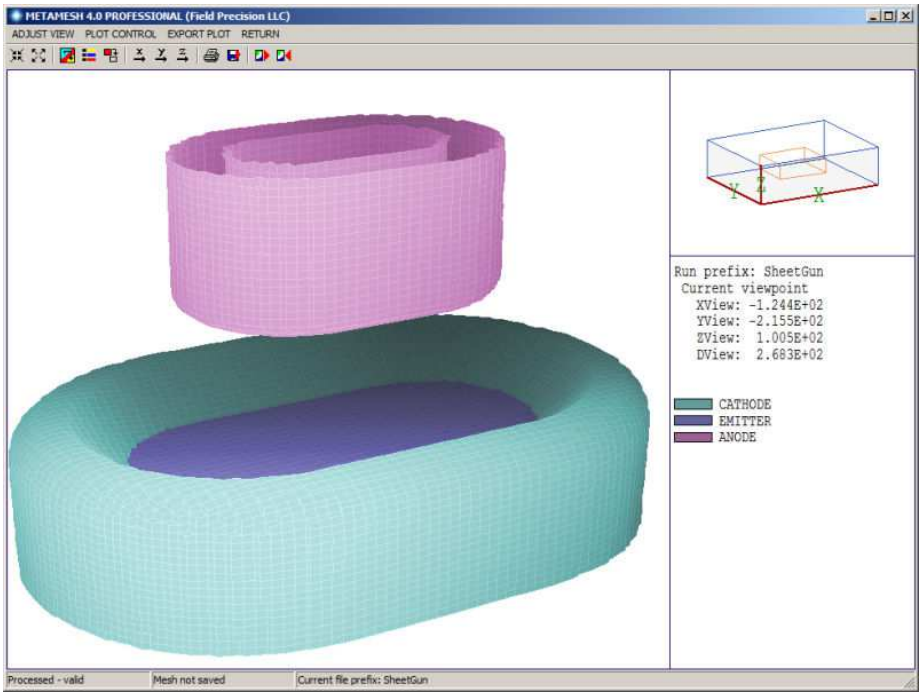

Figure 10: MetaMesh, assembly using the FreeCAD part.