Tokamak Modeling and Meshing Software (TOMMS) & SimModeler

To get better starting meshes for the M3DC1 adaptation, we are going to use TOMMS (a SCOREC tool dedicated to Tokamak meshing) for the initial model generation and SimModeler for the mesh generation. The details of the TOMMS are available at the following link.

https://github.com/SCOREC/Fusion_Public/wiki

The user guide and control parameters can be accessed using the following links.

https://github.com/SCOREC/Fusion_Public/wiki/Getting-Started-with-XGC-Meshing-Code-:User-Guide

https://github.com/SCOREC/Fusion_Public/wiki/List-of-input-parameters

The test cases for multiple Tokamaks (DIII-D, Alcator C-Mod, ITER, JET, LTX, KSTAR) can be found in the following repository.

https://github.com/SCOREC/Fusion_Public/tree/master/samples

1. Input Requirement

TOMMS requires a geqdsk file containing the information about the magnetic flux surfaces and wall curve of the Tokamak, and a file containing the control parameters for model and mesh generation. A sample geqdsk for ITER and control parameters file can be found in the following links:

https://github.com/SCOREC/Fusion_Public/blob/master/samples/ITER_10MA_xgca/ITER_10MA.geqdsk

https://github.com/SCOREC/Fusion_Public/blob/master/samples/ITER_10MA_xgca/xgc_mesh_in

There could be more input files for the better control of flux curves and mesh size, but we do not need them for our initial meshes here. I will be using the above mentioned ITER case in this document. You can find this test case in the samples.

https://github.com/SCOREC/Fusion_Public/tree/master/samples/ITER_10MA_xgca

Once we have the following inputs files ready, we can access the TOMMS executable from the following location at PPPL machines.

/p/tsc/m3dc1/lib/SCORECLib/rhel7/intel2019u3-openmpi4.0.3/tomms/xgc_meshgen

2. Model Generation

Follow the steps below to generate the model using the TOMMS. The model will contain the core region inside the separatrix curve, separatrix curve with both legs, SOL region and wall curve.

i. Load the following modules.

>> module load intel/2019.u3 openmpi/4.0.3 hdf5-parallel/1.10.5 cmake simmodsuite/15.0-200714

ii. Run the following command.

>> setenv OMP_NUM_THREADS 1

iii. Go the sample directory. In this case

>> cd/Fusion_Public/samples/ITER_10MA_xgca

iv. Modify the input file (xgc_mesh_in) according to our requirement. To create the model with just single flux curve (separatrix), set the following inputs in xgc_mesh_in as shown below.

a. radial_uniform_meshing_unit 1 (Make sure it is 1 and not -1)
b. inter_curve_spacing 1 (set spacing to 1, setting any value lesser than 1 will add additional flux curves- This spacing is set in terms of normalized psi value so setting the spacing 1 will make sure we only get separatrix curve)
c. Delete or comment out the following line if it exists in input file because it is used to get better control close to origin by adding an additional flux curve and we do not need it here.

inter_curve_spacing_axis_forcing 0.02

d. intra_curve_spacing_option 1

v. Only change the above-mentioned parameters, the rest should remain as it is in the input file.

vi. Once the input parameters are set, call the executable in the sample directory.

>> /p/tsc/m3dc1/lib/SCORECLib/rhel7/intel2019u3-openmpi4.0.3/tomms/xgc_meshgen

vii. There are a number of output files that we will get including model file (.smd), mesh file(.sms), file with nodes information (.node), elements file(.ele) and flux curves file (.flx.aif), some vtk files for the visualization in Paraview. We are only interested in model file (.smd) for our needs. (Note: The mesh file (.sms)

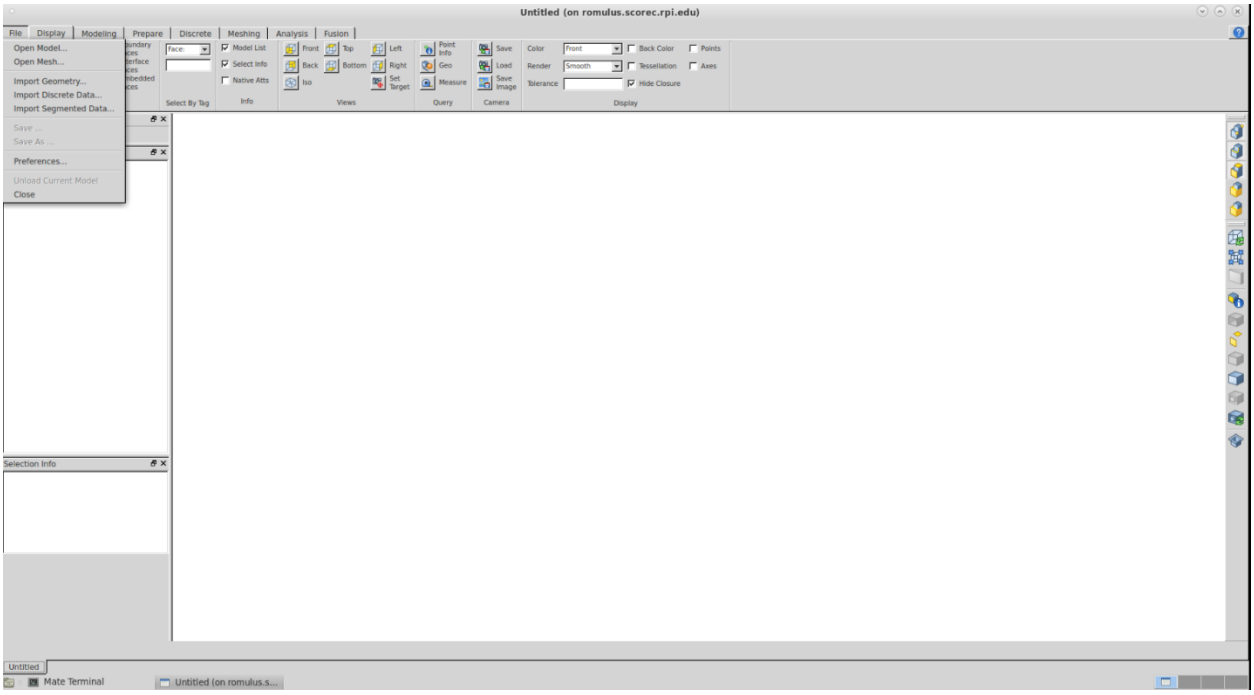
generated by the executable contains the mesh with one element deep mesh. Since the only flux curve is separatrix curves, it gives a bad looking one element deep mesh, but we are not interested in this mesh).

3. Mesh Generation in SimModeler

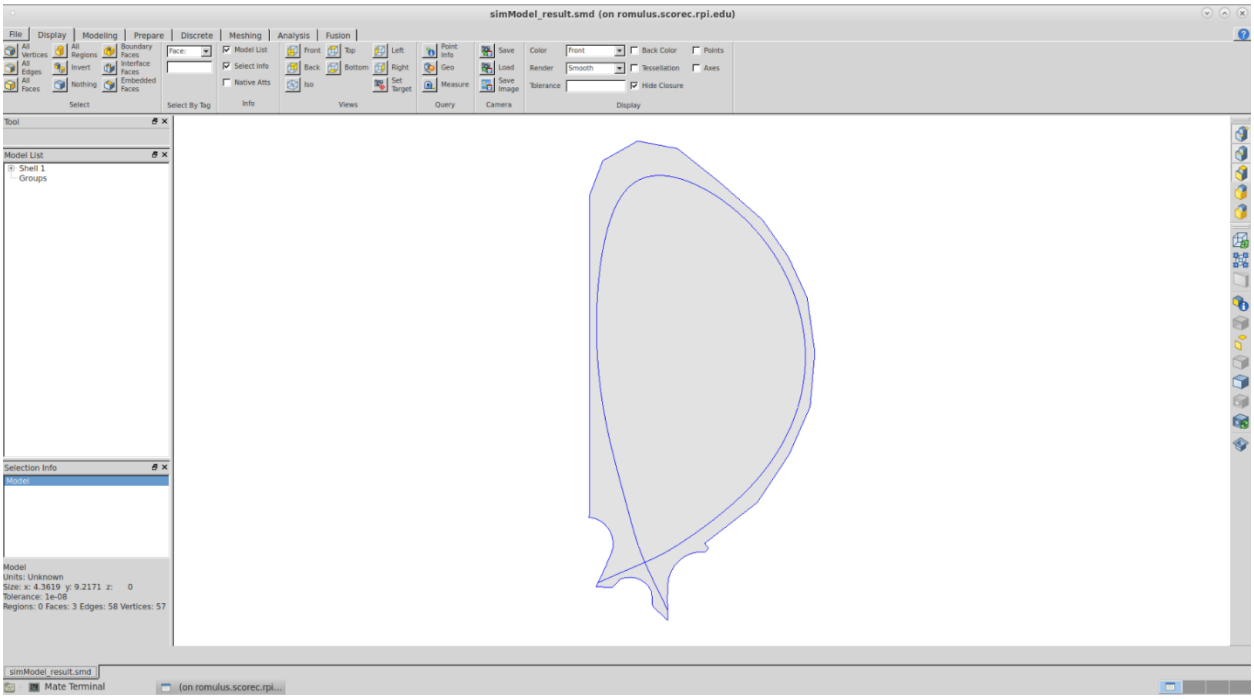
I. Load the following module if not already loaded.
>> module load simmodeler/9.0-200330dev
>> module load simmodsuite/15.0-200714

II. Open Simmodeler
>> simmodeler

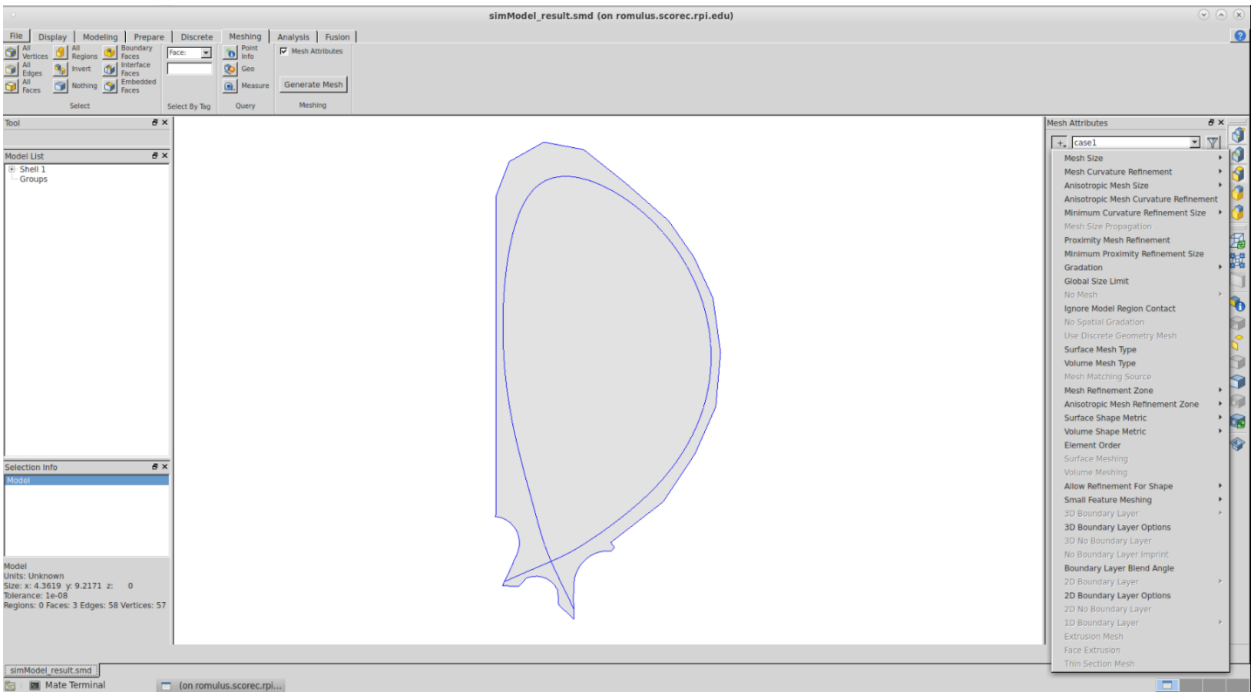
III. The interface should look like the image given below. Go to the file, open model, and then select model file simModel_result.smd from the working directory.



IV. The model should look like this with only separatrix curve. Change the view to top from the top menu to see model in this view.

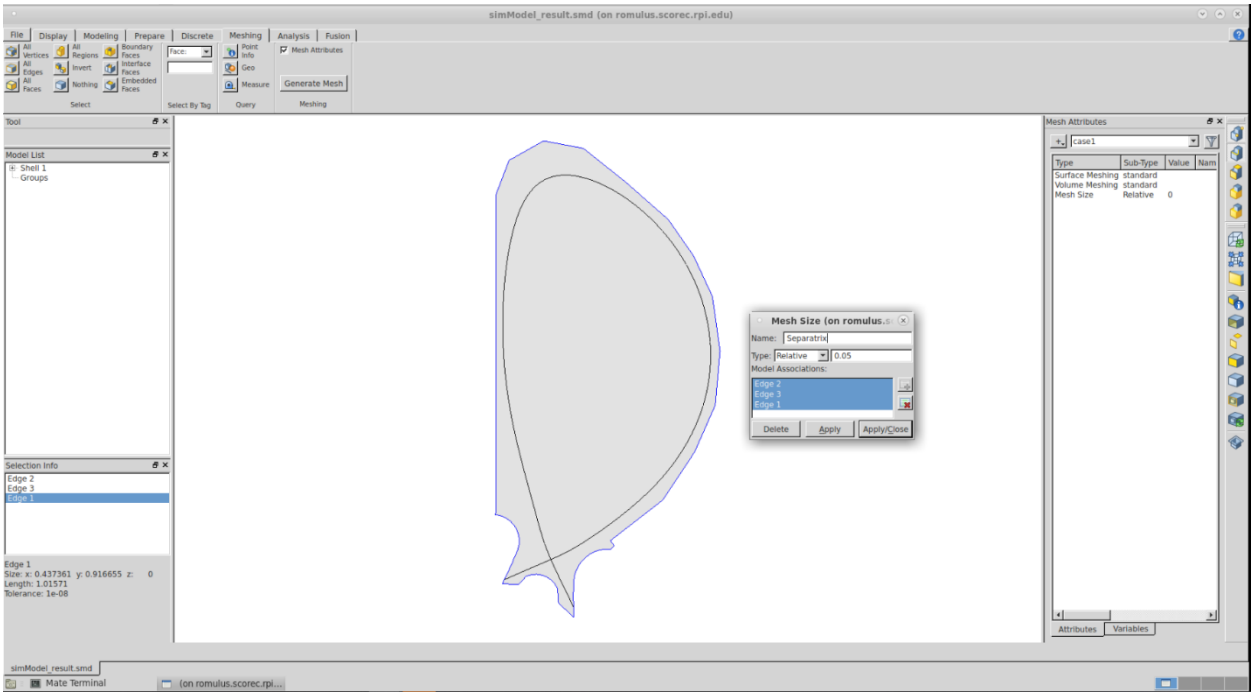


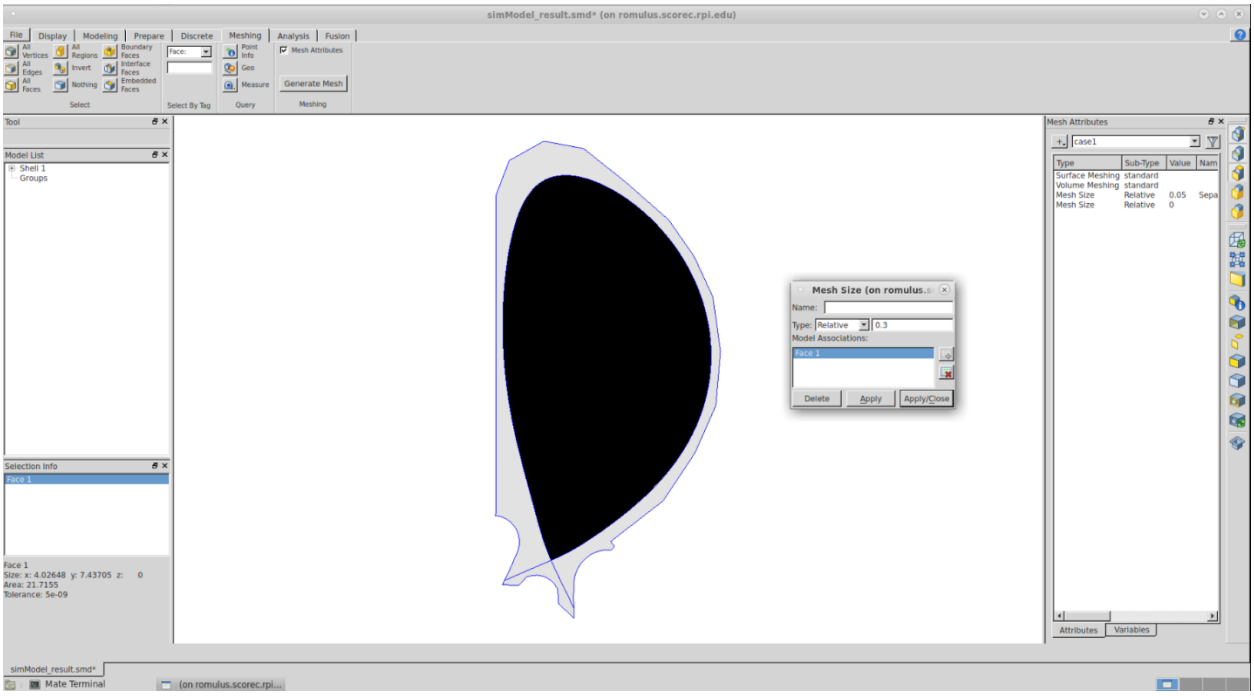
V. Go to the meshing menu and you will see meshing attributes in the drop down menu on the right side.

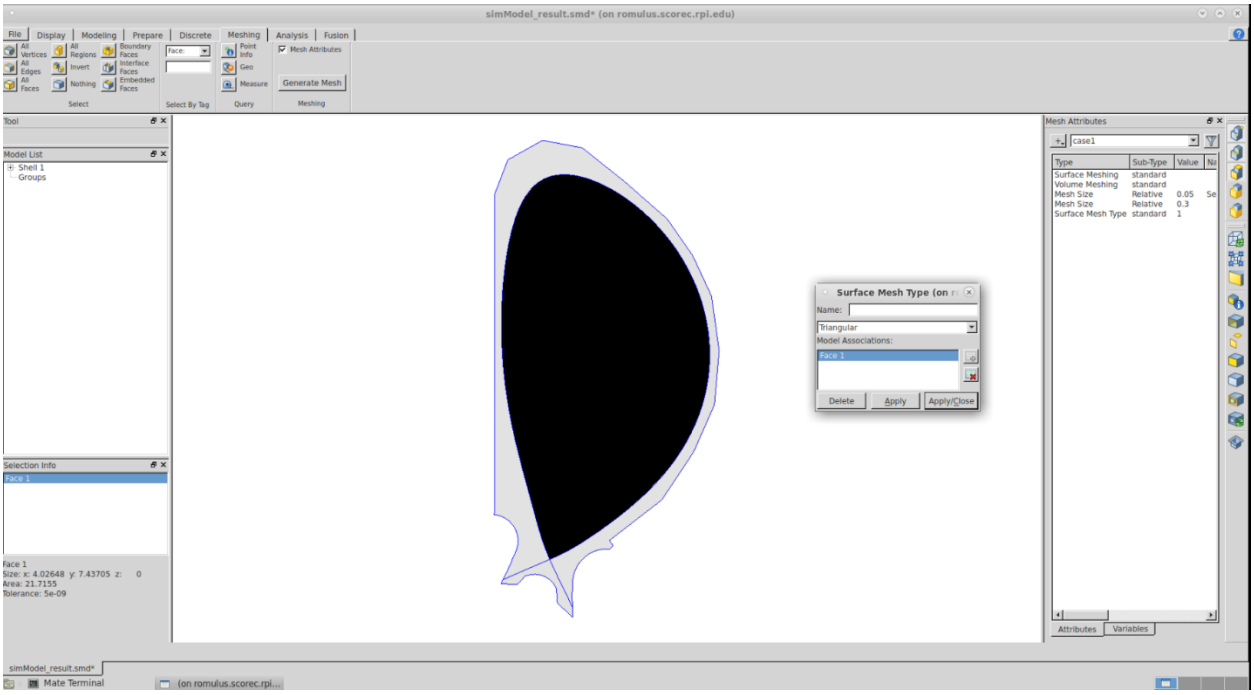


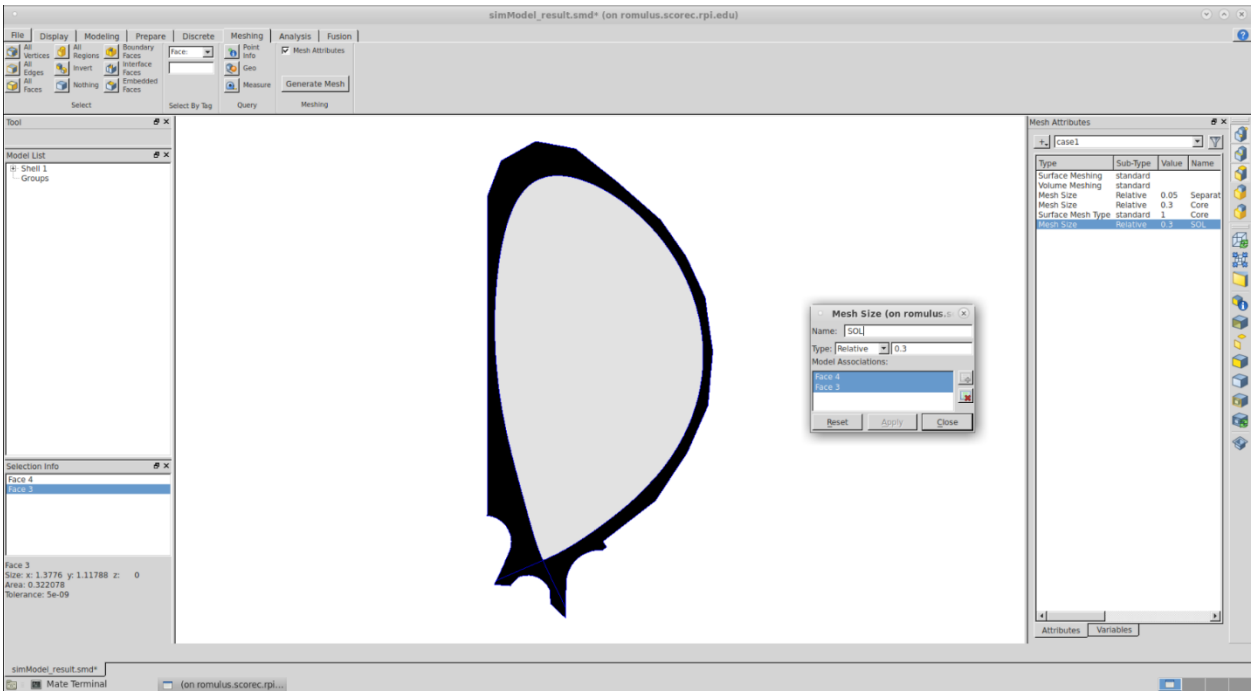
VI. We will need mesh size, surface mesh type, gradation, and no spatial gradation from the menu. First select the size required on the separatrix curve (probably much finer than core region). Select all the three edges (separatrix closed part+ two legs) and click on mesh size to select a mesh size. After selecting the size on

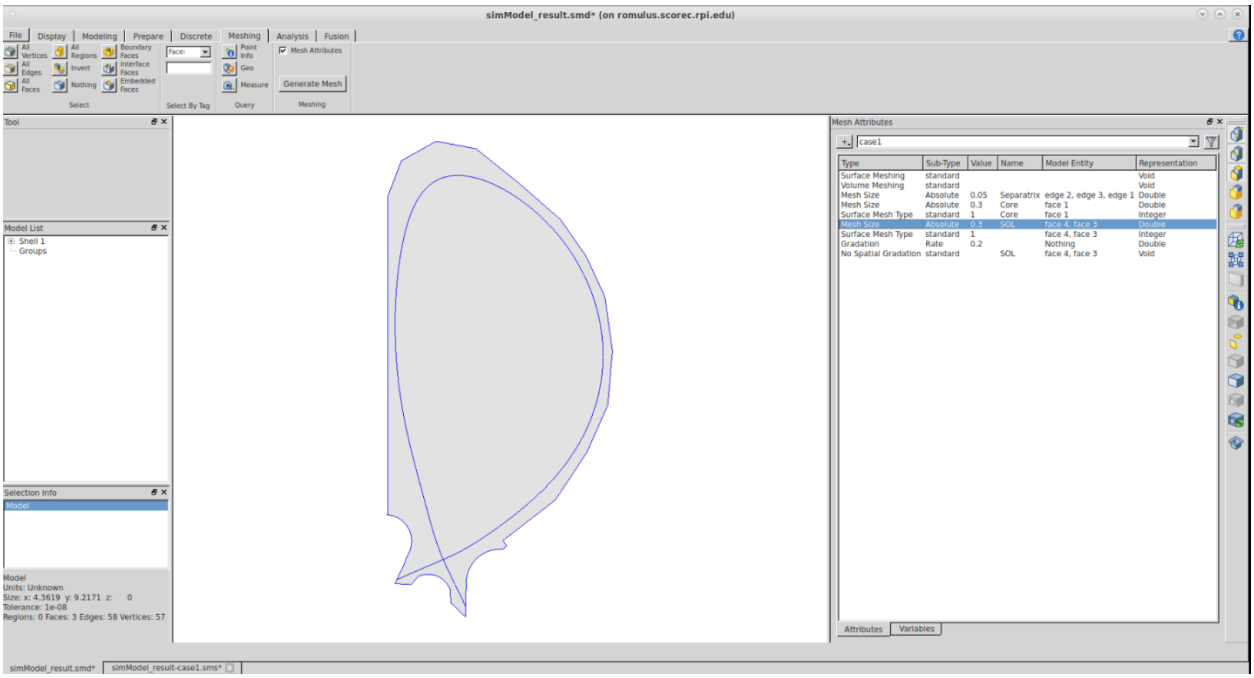
edges, select the mesh size and surface mesh type for core and SOL regions. It could be done in single step if you want the same size in all the faces by selecting all the faces and then setting the size and mesh type (triangular). Or select individual faces if you want to set different mesh size in all the faces. Once the sizes on edges and faces are set, set the gradation rate (working with gradation rate is suitable in our case rather than gradation factor). This rate will determine the gradation from the mesh size on edge to the mesh size on other faces. If you do not want the mesh size on the edge propagates to some face, then select the face and set it in no spatial gradation. There will still be a default transition between the sizes from edge to face, so it does not eliminate the gradation completely. Once these parameters are set, click on Generate Mesh to get the mesh. This mesh will be saved in (.sms) format and it can be converted to pumi mesh (.smb) for our use. The steps are shown below.

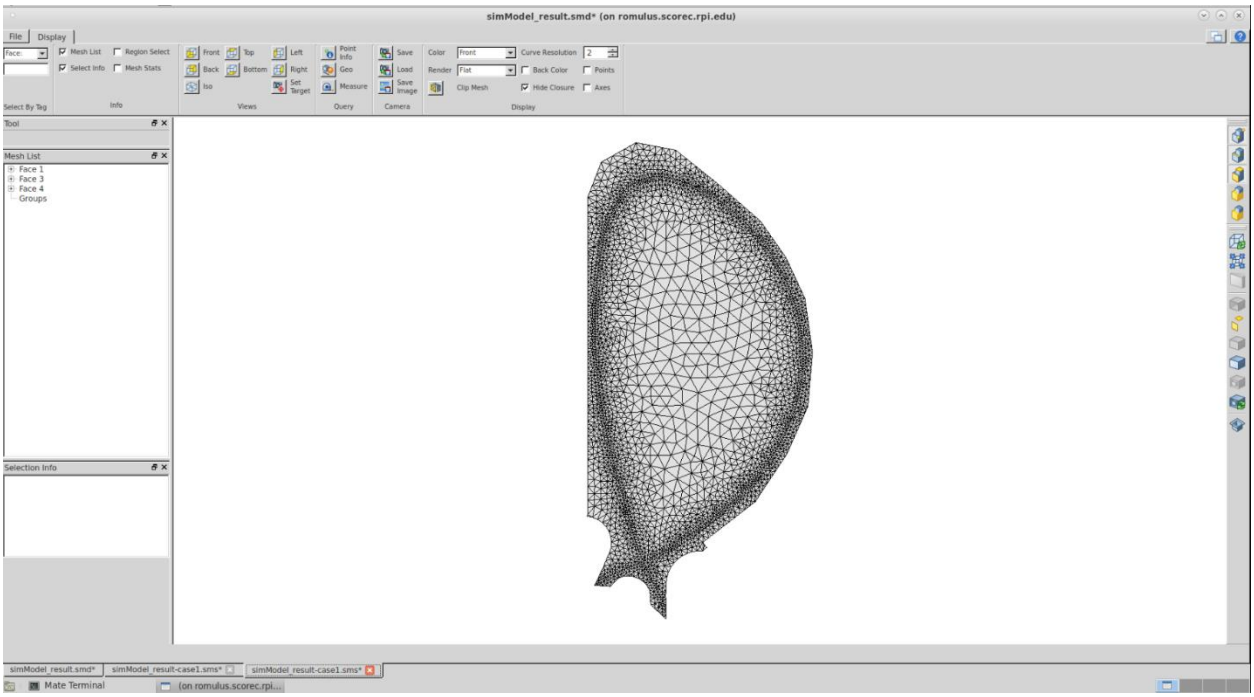



simModel_result.smd* (on romulus.scorec.rpi.edu)
File
Display
Modeling
Prepare
Discrete
Meshing
Analysis
Fusion
Mesh Attributes
Generate Mesh
Tool
Model List
Shell 1
Groups
Selection Info
Face 1
Face 1
Size: x: 4.02648 y: 7.43705 z: 0
Area: 21.7155
Tolerance: 5e-09
Mesh Size (on romulus.s
Name:
Type: Relative 0.3
Model Associations:
Face 1
Delete
Apply
Apply/Close
Mesh Attributes
case1
Type
Sub-Type
Value
Surface Meshing standard
Volume Meshing standard
Mesh Size Relative 0.05
Mesh Size Relative 0
Attributes
Variables
simModel_result.smd*
Mate Terminal
(on romulus.scorec.rpi...


simModel_result.smd* (on romulus.scorec.rpi.edu)
File
Display
Modeling
Prepare
Discrete
Meshing
Analysis
Fusion
Mesh Attributes
Generate Mesh
Tool
Model List
Shell 1
Groups
Selection Info
Face 1
Face 1
Size: x: 4.02648 y: 7.43705 z: 0
Area: 21.7155
Tolerance: 5e-09
Surface Mesh Type (on r
Name:
Triangular
Model Associations:
Face 1
Delete
Apply
Apply/Close
Mesh Attributes
case1
Type
Sub-Type
Value
Surface Meshing standard
Volume Meshing standard
Mesh Size Relative 0.05
Mesh Size Relative 0.3
Surface Mesh Type standard 1
Attributes
Variables
simModel_result.smd*
Mate Terminal
(on romulus.scorec.rpi...


simModel_result.smd* (on romulus.scorec.rpi.edu)
File
Display
Modeling
Prepare
Discrete
Meshing
Analysis
Fusion
Mesh Attributes
Generate Mesh
Tool
Model List
Shell 1
Groups
Selection Info
Face 4
Face 3
Face 3
Size: x: 1.3776 y: 1.11788 z: 0
Area: 0.322078
Tolerance: 5e-09
Mesh Size (on romulus.
Name: SOL
Type: Relative 0.3
Model Associations:
Face 4
Face 3
Reset
Apply
Close
case1
Type
Sub-Type
Value
Name
Surface Meshing standard
Volume Meshing standard
Mesh Size Relative 0.05 Separat
Mesh Size Relative 0.3 Core
Surface Mesh Type standard 1 Core
Mesh Size Relative 0.3 SOL
Attributes
Variables
simModel_result.smd*
Mate Terminal
(on romulus.scorec.rpi...


simModel_result.smd* (on romulus.scorec.rpi.edu)
File
Display
Modeling
Prepare
Discrete
Meshing
Analysis
Fusion
Mesh Attributes
Generate Mesh
Tool
Model List
Shell 1
Groups
Selection Info
Model
Model
Units: Unknown
Size: x: 4.3619 y: 9.2171 z: 0
Tolerance: 1e-08
Regions: 0 Faces: 3 Edges: 58 Vertices: 57
case1
Type
Sub-Type
Value
Name
Model Entity
Representation
Surface Meshing standard Void
Volume Meshing standard Void
Mesh Size Absolute 0.05 Separatrix edge 2, edge 3, edge 1 Double
Mesh Size Absolute 0.3 Core face 1 Double
Surface Mesh Type standard 1 Core face 1 Integer
Mesh Size Absolute 0.3 SOL face 4, face 3 Double
Surface Mesh Type standard 1 face 4, face 3 Integer
Gradation Rate 0.2 Nothing Double
No Spatial Gradation standard SOL face 4, face 3 Void
Attributes
Variables
simModel_result.smd*
simModel_result-case1.sms*


simModel_result.smd* (on romulus.scorec.rpi.edu)
File
Display
Mesh List
Region Select
Select Info
Mesh Stats
Front
Top
Left
Back
Bottom
Right
Iso
Set Target
Point Info
Geo
Measure
Save
Load
Save Image
Color
Front
Curve Resolution
Render
Flat
Back Color
Points
Clip Mesh
Hide Closure
Axes
Select By Tag
Info
Views
Query
Camera
Display
Tool
Mesh List
Face 1
Face 3
Face 4
Groups
Selection Info
simModel_result.smd*
simModel_result-case1.sms*
simModel_result-case1.sms*
Mate Terminal
(on romulus.scorec.rpi...