Generating a 3D GAMBIT grid for concentric rotating cylinders
Prepared by: Jagannath R Nanduri
(jaggu.nanduri@mail.wvu.edu)
304-293-3111x2348

1. In order to overcome the problem of generating small size factors during grid generation phase of GAMBIT multiply all the above dimensions by 10^4. The grid will be scaled back to original dimensions in FLUENT.

2. Correct order of grid generation
   a. Put nodes on all the EDGES
   b. Mesh the FACE
   c. Mesh VOLUME

3. If any of the above steps are omitted i.e. you go directly to meshing FACE without meshing the EDGE, GAMBIT puts a mesh of default size on the lower topology (in this case the edge). This behavior of GAMBIT can be used to our advantage by specifying a default size factor.

4. Create two cylinders and split the outer cylinder volume using the menus.

5. Create two planes and split the outer cylinder volume into four quadrants using the menus.
6. You now have 5 VOLUMES and each volume has 4 FACES with some common faces belonging to multiple volumes. Before step 5 you had only 4 EDGES, now you have 32 EDGES. For other complex geometries you will have to grade (add mesh using successive meshing and grading factor) each and every edge before you can go to FACES. We are not going to grade each edge in this simple geometry. We are going to take advantage of boundary layer grids in GAMBIT.

7. Multiple boundary layers can be drawn to each EDGE depending on the number of FACES connected to the EDGE. For example in the figures below, the selected EDGE belongs to 3 FACES and hence 3 boundary layers can be drawn as shown. In order to select the boundary layer for a specific FACE you have to click the scroll button of your mouse on the edge while pressing shift button. This will cycle through all the available boundary layers with a red arrow showing all the direction of propagation of the boundary layer. For the example given in figure below, we want the boundary layers in the first and third figures but not the second.
8. You can attach boundary layers to the respective edges using the menu options as shown. The factor a, b and w are important. These will depend on the geometry, the turbulence model you will be using to simulate the flow. All turbulence models will require some kind of near wall models to predict the flow close to the wall. FLUENT provides three options for all turbulence models with 2-equations are higher.

a. Standard Wall Functions
In order to use standard wall functions the first grid node next to the wall should follow $5 < y^+ < 50$.

b. Enhanced Wall Functions
In order to use enhanced wall functions the first grid node next to the wall should follow $y^+ < 5$.

c. User Defined Wall Functions
The first grid node next to the wall should follow the requirements of user defined wall models

In this case we will use $a = 1; b/a = 1.2$ and 10 rows. This will make the first node at 0.1 mm from the wall in physical dimensions. We still need to check the value of $y^+$ in FLUENT and apply the appropriate wall functions. Notice that you have no control on w. You will have this control if you grade the EDGE before going to boundary layers. GAMBIT will use a default w when putting the boundary layer but it will change this to the required size when you grid the VOLUME.

Apply boundary layers to all the necessary edges as shown in the figures below. You will have a boundary layer for the inner surface of the outer wall, the inner surface of the inner wall and the outer surface of the inner wall. Make sure you use the center scroll button of the mouse to select the appropriate boundary layer.
9. Next we are going to directly mesh the volume. We will be creating both structured and unstructured meshes. First we are going to set default mesh size. Go to Edit » Defaults » Mesh tab, select SIZE, change the value to 30 and hit Modify button.

10. Got to Mesh » Volume, and select the four outer volumes. As shown below. The default meshing scheme is Hex, Type is Map and spacing is 1. Hit Default above the spacing field and it will change to 30 the default we specified. Hit apply and all four outer volumes will be meshed as shown. Then select the inner volume, scheme »Hex, type » Cooper and spacing » 1 and hit apply to mesh the inner volume. For more discussion on the various meshing techniques (Cooper, Map, Pave etc.) see the GAMBIT user guide.
11. The next step is to apply the respective boundary conditions. This is similar to the ones you did in the 2D cases. Select the respective faces for the inlets outlets and walls. Select the four outer volumes and make them air_zone fluid and the inner volume as fuel_zone fluid. In order to make the selection of the faces easier, you can turn off the meshes.
12. You can examine the mesh using the lens button in the bottom right corner of the GUI window. Just drag the x, y, z bars to see the mesh at various locations in your geometry. Pay attention to the color of the mesh, this shows the quality of the mesh. Blue is good quality and red is bad quality. Based on what you got you might have to go back and redo the mesh. See how the inner cylinder has an unstructured mesh (this type of map is called Pave mesh). This is mandatory for this geometry. Why?
13. Export the mesh to rot_cyl_3d_struc.msh and read it in FLUENT