

How to Build a mesh in ANSYS Fluent with Workbench

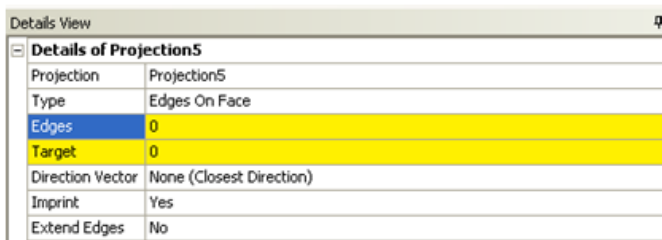
Building Meshes in Workbench

The processes of creating geometry and meshing are fairly straightforward, so there is no need to fill this report with details on how to perform these tasks. There is, however, one trick that is worth mentioning for future years since it is non-intuitive. In order to create a uniform mesh where cell sizes are the same through the core of the flow, the thickness of each baffle needs to be infinitely thin. Unfortunately, ANSYS has only a convoluted way of doing this. Simply creating a series of lines in a sketch and converting them to line bodies will not work. For some reason, the meshing tool does not recognize line bodies. The following are the steps needed to create infinitely thin baffles.

- **Step 1:** Draw the rectangular region that will serve as the fluid flow domain and create the corresponding surface body
- **Step 2:** Create a new sketch, draw the lines that will serve as the baffles, and create line bodies from this sketch
- **Step 3:** Under the Tools menu at the top of the DesignModeler, select Projection



- **Step 4:** At the bottom left of the screen, a details menu will appear; for Edges, select the line bodies that will serve as baffles and click Apply
- **Step 5:** For Target, select the surface body that will serve as the flow domain and click Apply



- **Step 6:** Click Generate (the lightening bolt)

