Partially Premixed Combustion - Mesh

Author: Lara Backer, Cornell University - taken from ANSYS Inc. tutorial

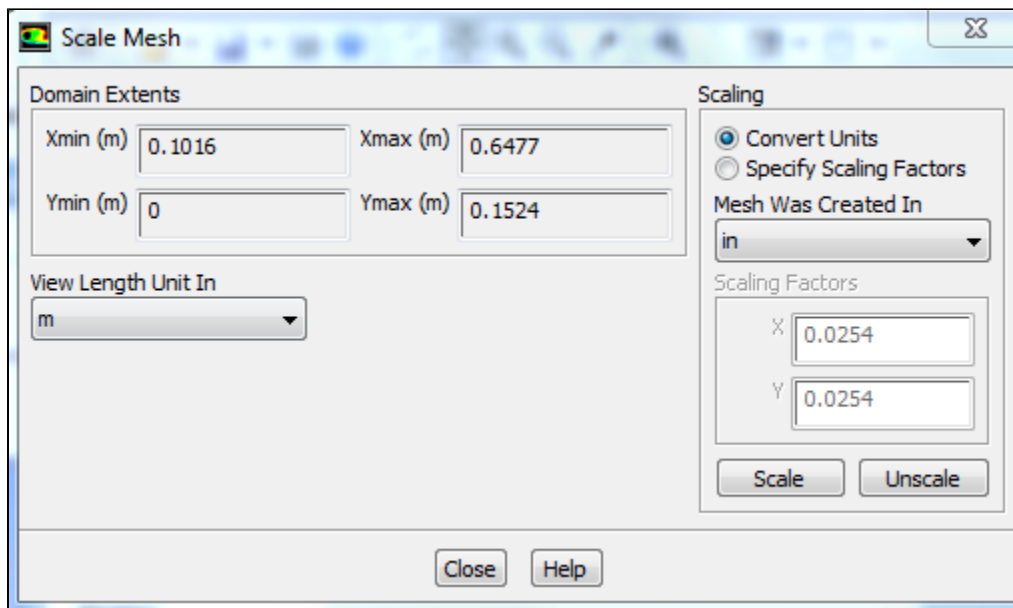## Mesh

Download the mesh file here. Open Ansys Workbench. Drag and drop a Fluid Flow - Fluent case into the main workbench area and save the project. Double click on "Setup", skipping the Geometry and Mesh options. In the pop-up dialogue, select the **2D** and Double precision options. With FLUENT open, go to **File-Import-Mesh** and select the file that you just downloaded. Go to Solution Setup-General and click "Display" under mesh options to show the mesh. It should look like this:
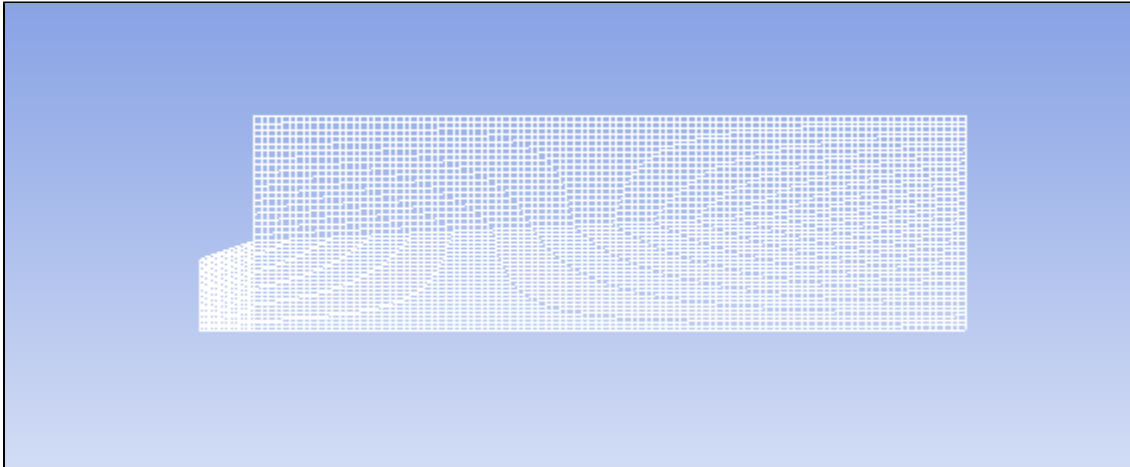
If you go to **Mesh-Info-Size** at the top menu of the screen, there should be 4700 cells in the domain.

The mesh was originally created in inches. Click on "Scale" under the Mesh options then select "in" in the drop down box called "Mesh Was Created In", then click "Scale" and close the dialogue box.

To speed up future computations, select **Mesh - Reorder - Domain** in the menu at the top of the screen. This increases memory access efficiency.

Go to Step 4: Physics Setup

Go to all FLUENT Learning Modules