Turbulent Jet - Verification & Validation

Authors: Lara Backer, Cornell University - taken in part from MAE 7340 Analysis of Turbulent Flows and Dr. S. Pope

Verification & Validation

To ensure that your analysis was performed on a fine enough grid so that there were minimal numerical errors, refine your mesh and run the turbulent or laminar solutions.
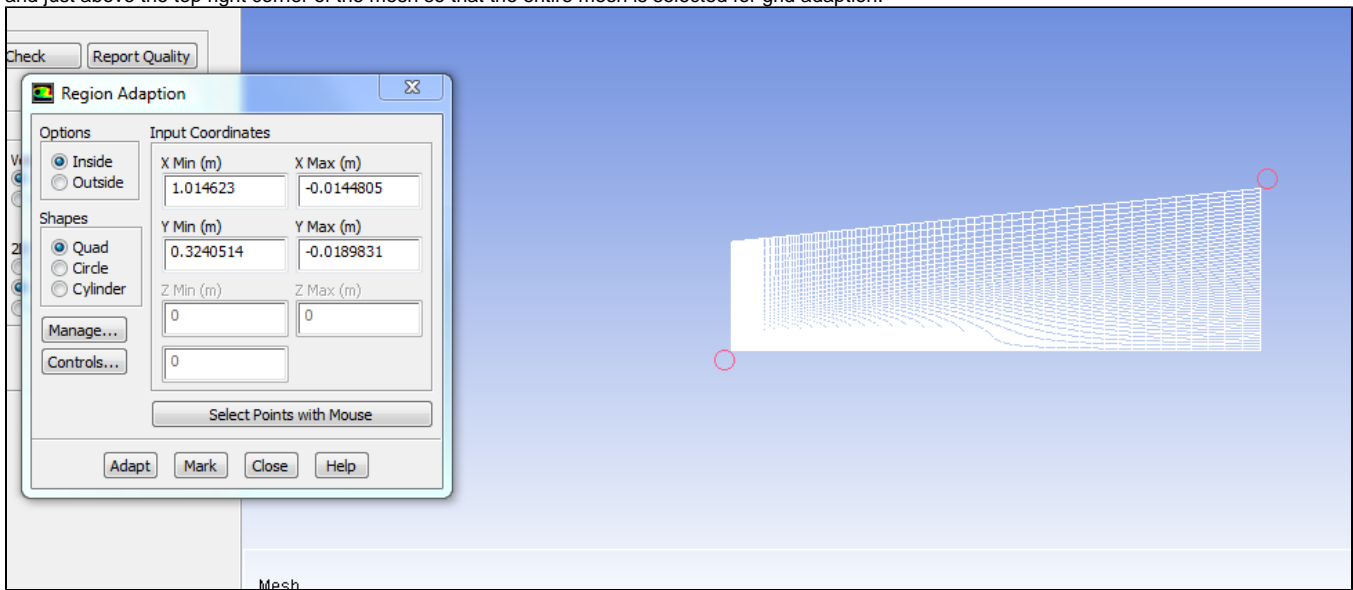
To do this, first display your mesh on the screen: Solution Setup - General; click on "Display" and select "fluid" as the surface. Press Display again, and then close the dialogue.
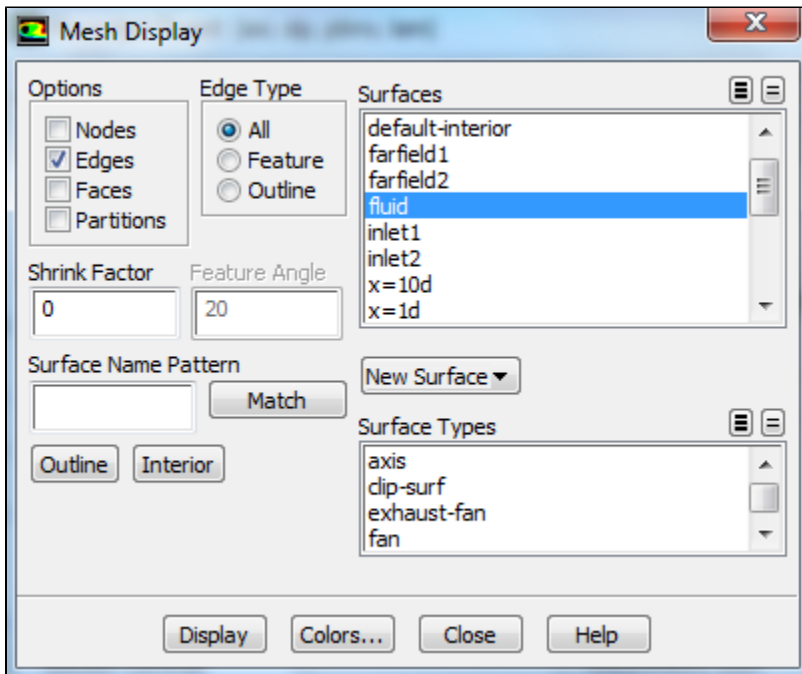

Go to Adapt - Region at the top menu. Select Controls, set the Max Level of Refine to 1 and click "OK".
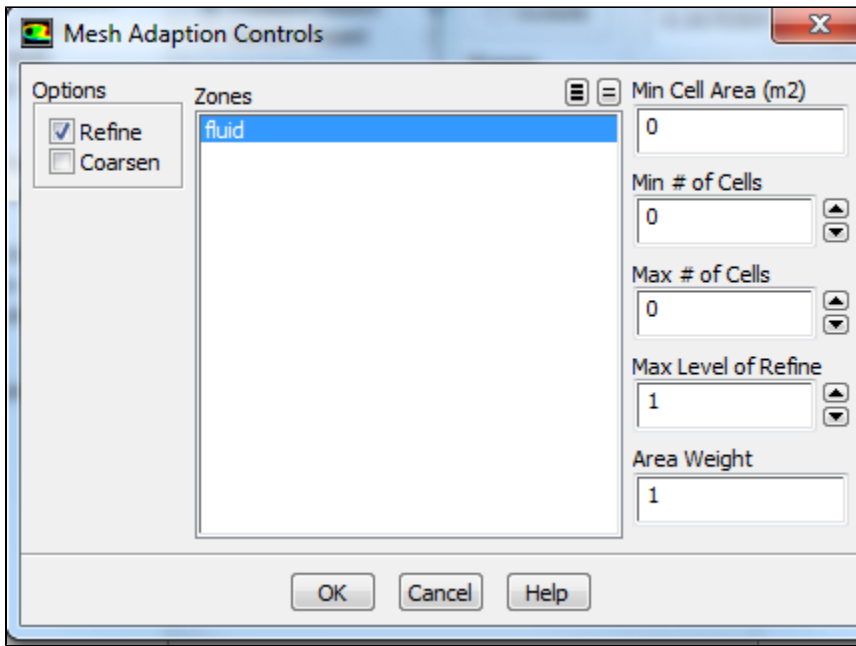
Then, click Select Points with Mouse. Select the probe button to enable point selection and right-click with your mouse just below the bottom left corner and just above the top right corner of the mesh so that the entire mesh is selected for grid adaption.

Finally, click Adapt. Display your newly adapted mesh: Solution Setup - General; click on "Display" and select "fluid" as the surface. Press Display again, and then close the dialogue. The mesh should now look like this:
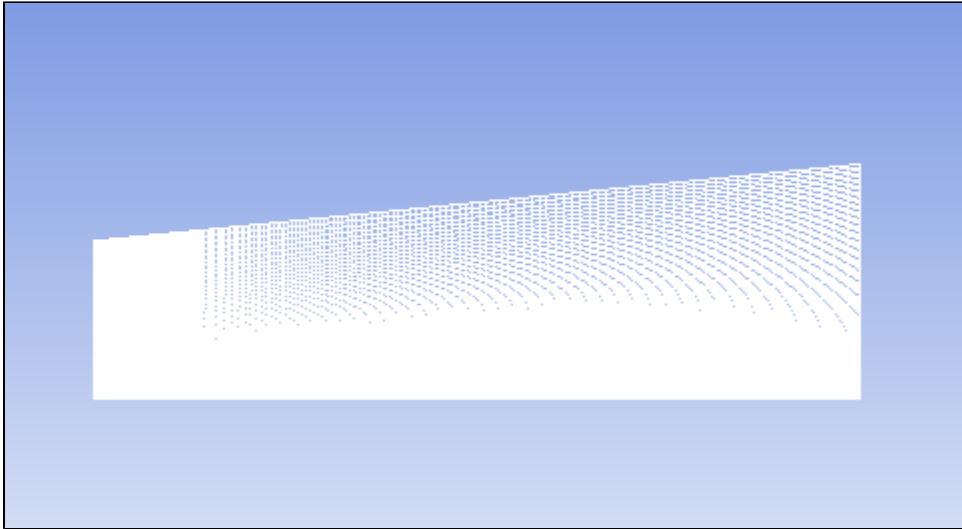
To accelerate convergence and just solve the flow equations initially without turbulence, go to Solution - Solution Controls - Equations. Unselect Turbulence and press OK.

Go to Solution - Run Calculation. Run the calculation for 2000 iterations, monitoring the residuals. Now go back to Solution - Solution Controls - Equations. Select Turbulence and press OK. Run the solution for additional iterations until you obtain steady state residuals as before.