

Turbulent Jet - Laminar Setup and Solution

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

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Laminar Setup and Solution](#)

[3. Laminar Results](#)

[4. Turbulent Setup and Solution](#)

[5. Turbulent Results](#)

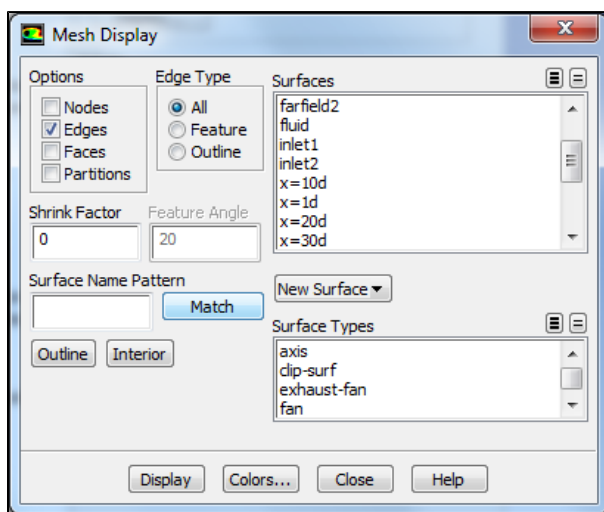
[6. Verification & Validation](#)

[Exercises](#)

[Comments](#)

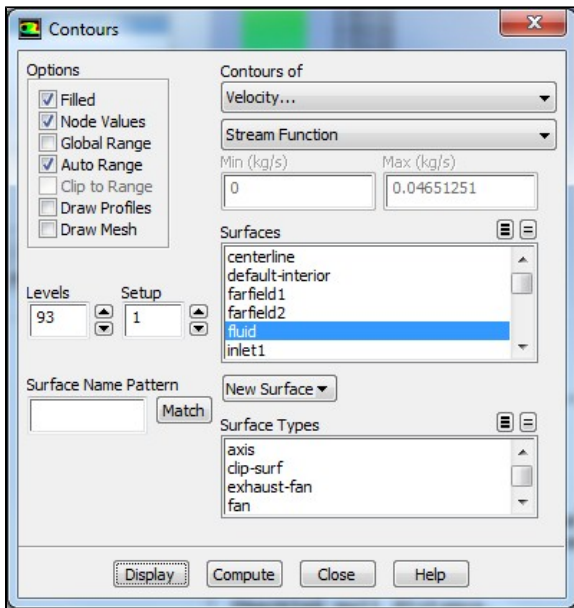
Laminar Jet Setup

To make sure that the mesh imported correctly, go to Solution Setup-General and under Mesh, click "Display" and the dialogue box as shown below will open. Display the mesh by selecting and displaying the appropriate surfaces (fluid).

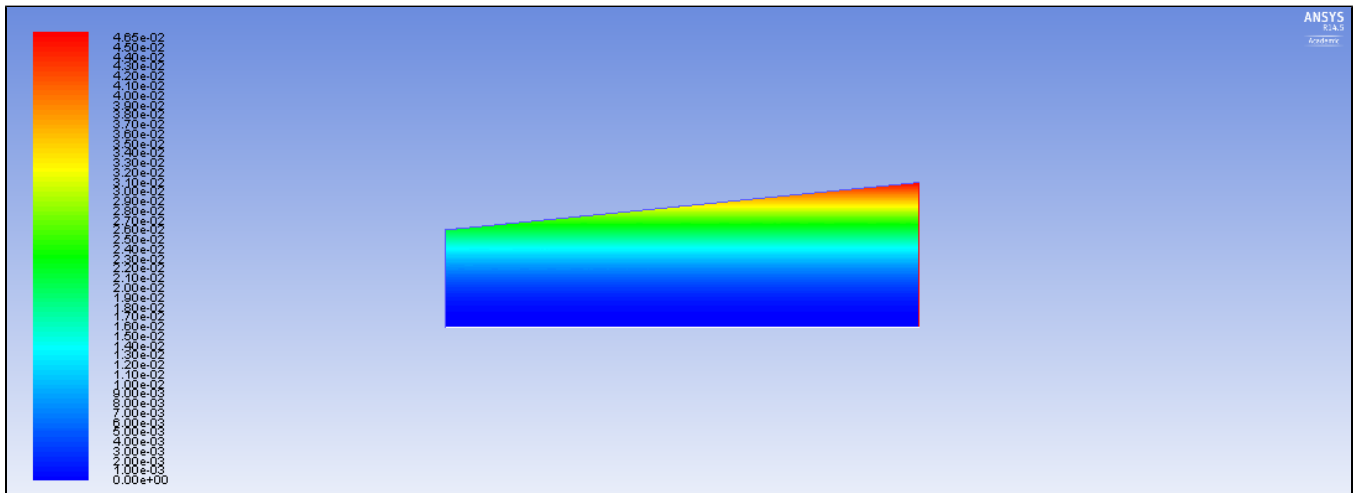


Closing the mesh display dialogue and going back to the Solution Setup-General, make sure that the axisymmetric solution is selected under 'General', viscous-laminar is selected under 'Models', and that the appropriate boundary conditions and materials for the **laminar case** are selected based on the Problem Specification.

Initialize the solution by going to Solution-Solution Initialization and using Inlet 1 with axial velocity 1m/s to initialize the domain. Check your initialization by going to Results-Graphics and Animations, selecting "Contours" under the Graphics section, and clicking the "Set Up" button. Display the stream function (constant values along streamlines, which are tangent to flow velocity) for the fluid surface as shown in the dialogue box below. Click the Display button.

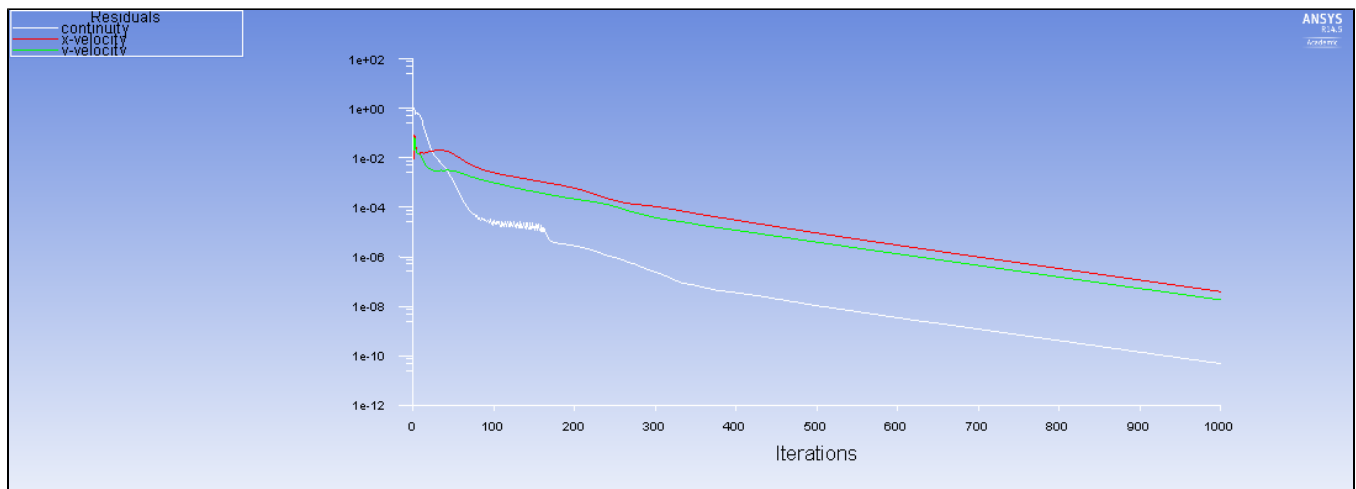


Your initial velocity stream function should look like this:



Laminar Jet Solution

Now obtain a converged solution by running the solution for 1000 iterations in Solution-Run Calculation. Your solution should be reasonably converged upon examining the x, y, and continuity residual plots:



[Go to Step 3: Laminar Results](#)

[Go to all FLUENT Learning Modules](#)