Turbulent Jet - Verification & Validation

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 Results 6. Verification & Validation Exercises Comments

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.

Mesh Displa	ay		×
Options	Edge Type	Surfaces	
Nodes	All	default-interior	
Edges	Feature	farfield1	
Faces	Outline	fluid	=
Partitions	-	inlet1	
Shrink Eactor	Feature Angle	inlet2	
		x=10d	-
	20	[X=10	
Surface Name Pa	attern	New Surface -	
	Match		
		Surface Types	
Outline Inter	rior	axis	
		clip-surf	
		fan	-
		[[tart	
	Display Colo	rs Close Help	

Go to Adapt - Region at the top menu. Select Controls, set the Max Level of Refine to 1 and click "OK".

💶 Mesh Adapt	ion Controls	X
Options Refine Coarsen	Zones	Min Cell Area (m2) Min # of Cells 0 Max # of Cells 0 Max # of Cells 0 Max Level of Refine 1 Area Weight 1

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.

heck Report Quality	
Region Adaption 23 Options Input Coordinates Outside X Min (m) X Max (m) Outside X Min (m) Y Max (m) Outside Y Min (m) Y Max (m) Outside Y Min (m) Y Max (m) Outside Y Min (m) Y Max (m) Outside Z Min (m) Y Max (m) Outside Z Min (m) Z Max (m) Outside Z Max (m) Outside Exerct Points with Mouse Exerct Points with Mouse Adapt Mark Close	

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.

Go to Exercises Go to all FLUENT Learning Modules