Turbulent Jet - Turbulent Setup and Solution

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

Turbulent Jet Setup and Solution

Background

As stated in the Pre-Analysis section, the k- model solves the Reynolds equations, which are time-averaged Navier Stokes equations with additional velocity fluctuation terms. This model provides closure needed for these equations with additional model specific transport equations for k and that account for the turbulent properties.

While k and can be specified, they can also be related to more physically understandable parameters I and d. I is the turbulent intensity, d_H is the hydraulic diameter, and d is the jet diameter.

$$I = 0.16Re^{-1/8},\ d_H = d,\ l = 0.07d$$

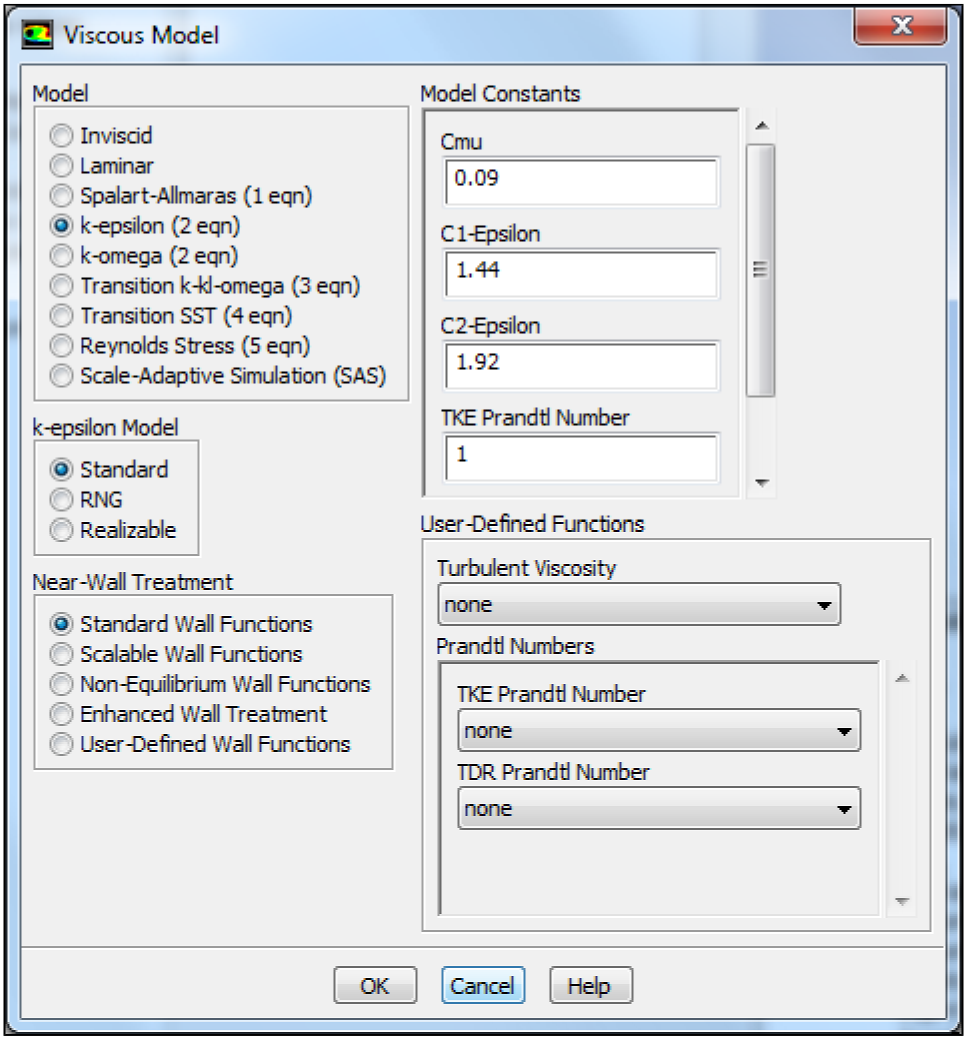
Note: for further understanding of model specifications, Fluent specifies certain constants as in the equations below. However, while these constants can be changed, Fluent initializes them at their standard values.

$$\epsilon = \frac{C_\mu^{3/4} k^{3/2}}{l},\ k = \frac{3}{2(UI)^2}$$

Setup

Use the same case and data files as you downloaded and checked in the Laminar setup for the geometry and mesh, but save this case as 'Turbulent Jet'.

Set up the k- model for turbulence. In Solution Setup-Models, double click the previous "Viscous-Laminar" option to open the dialogue. Select the "k-epsilon (2eqn)" model for turbulence. Leave the model constants the same as defined in the dialogue below; these values have been refined to constraints of the model and are typically not varied as mentioned above. Press OK. Note that wall functions are not required for this case unlike the turbulent pipe flow solution, as there are no wall boundary viscous effects to account for.

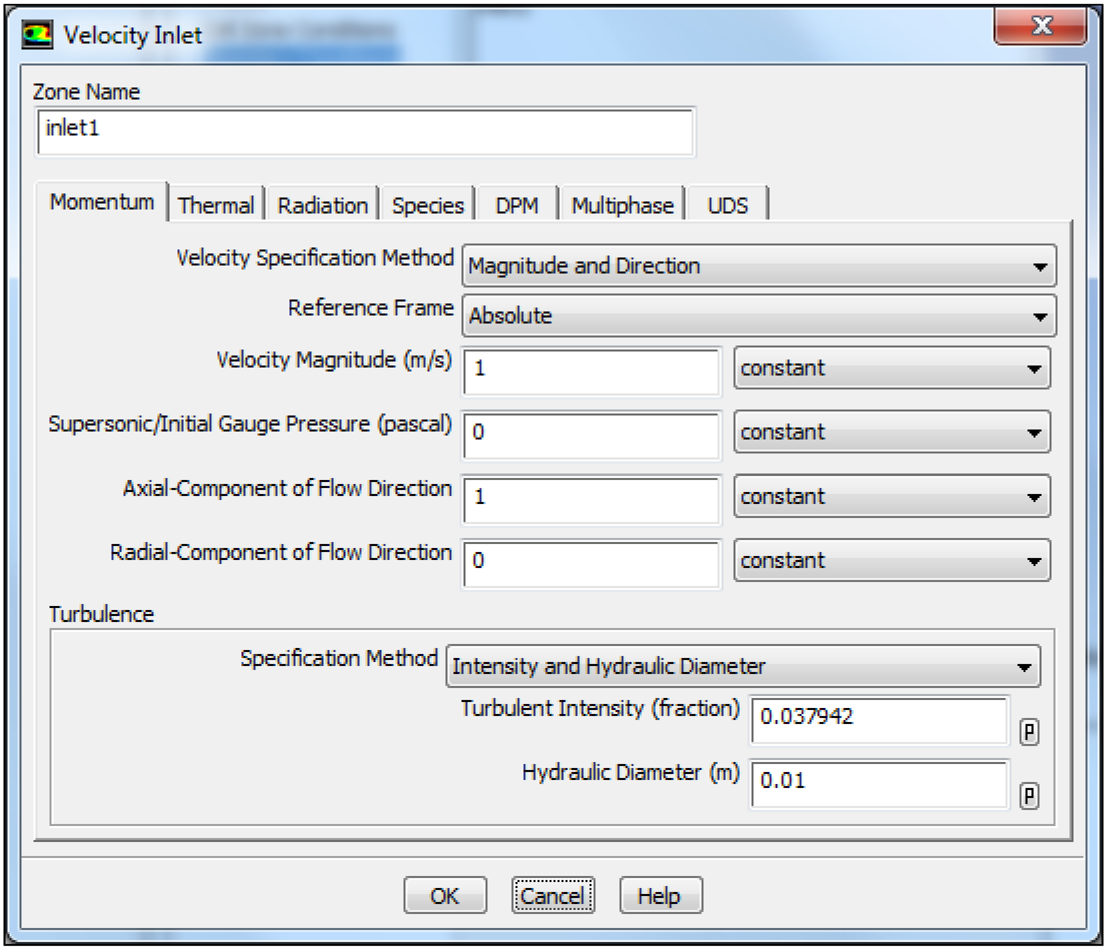


In the Materials dialogue. change the viscosity for a higher Reynolds number:

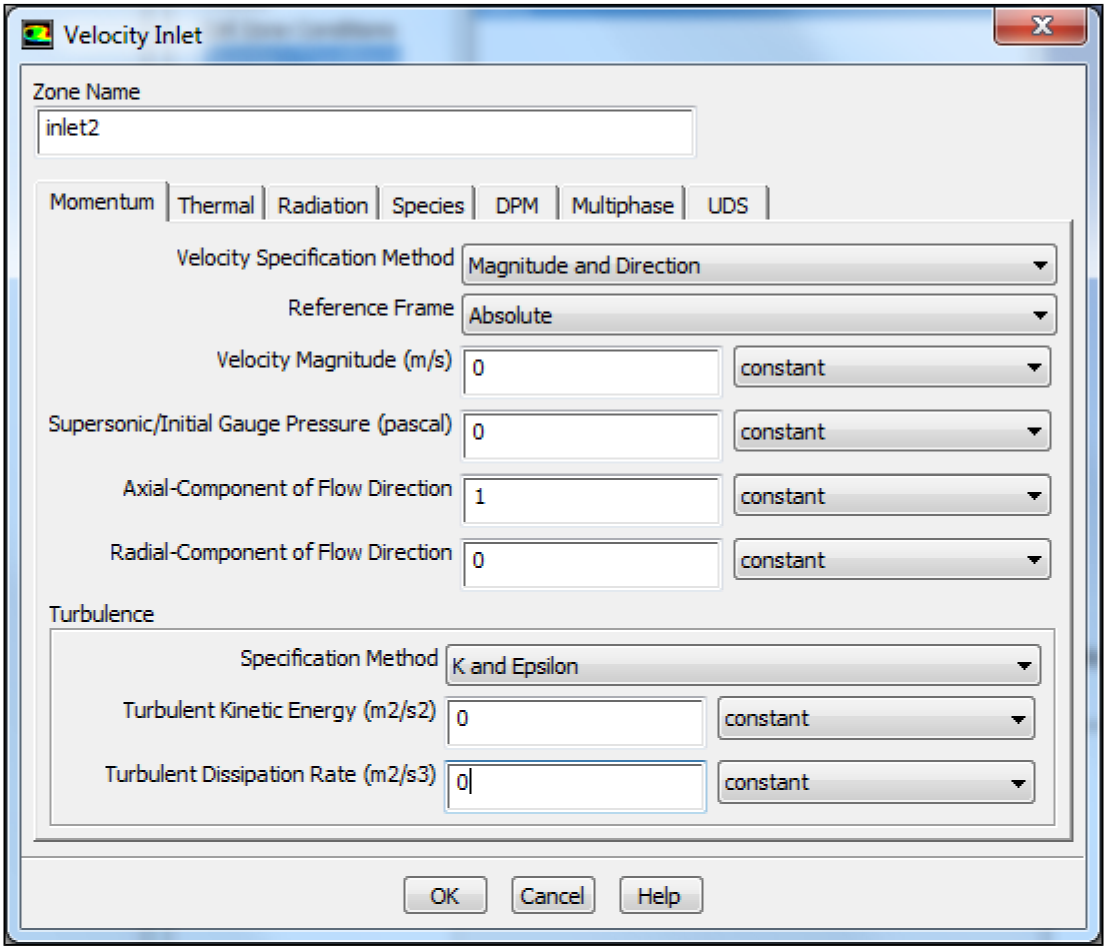
$$Re = \frac{Ud}{\nu} = 10^5$$

The inlet jet velocity will remain at 1m/s; the inlet diameter is still 0.01m from the geometry (ignoring the larger diameter with 0m/s velocity inflow), so enter the viscosity for air as = 1E-7. Select Change/Create and then Close.

The k- terms must be specified on all boundaries. Go to Solution Setup - Boundary Conditions and edit both **Farfield and Inlet boundaries**. Select "Intensity and Hydraulic Diameter" as the turbulence specification method. At Inlet 1, from the specified Reynolds number and above equations, the intensity should be I=0.037942, and the hydraulic diameter should be 0.01m (twice the measured inlet height).



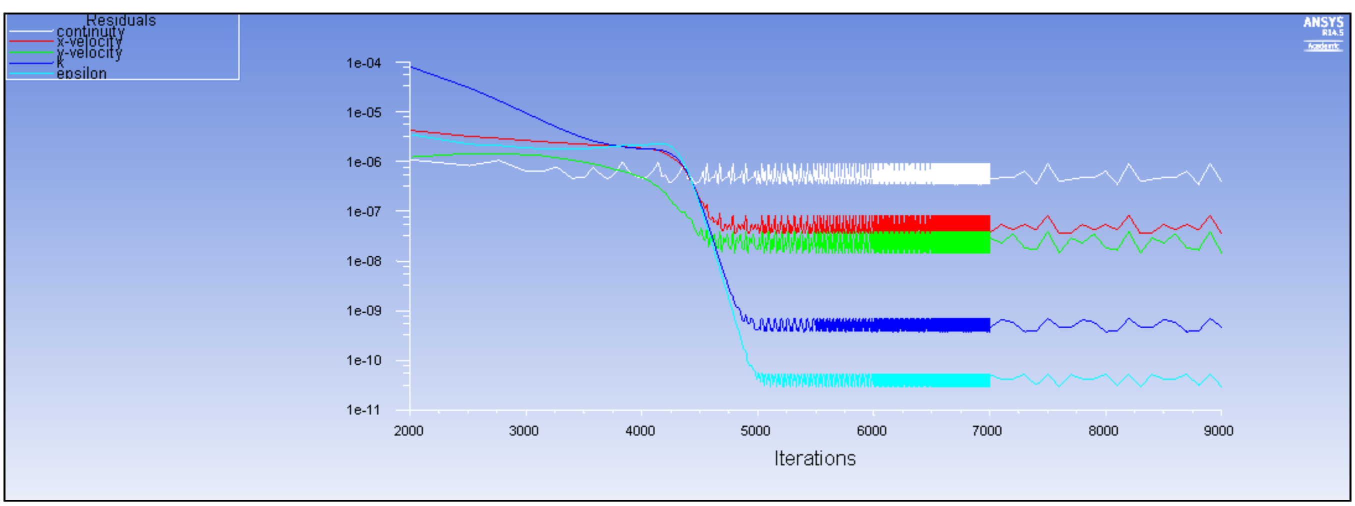
The other Inlet and both Farfield boundaries should have turbulent kinetic energy and dissipation rates set to zero, because there is no inlet velocity or turbulence associated with them. Press OK for all boundaries to save the inputs.



To initialize the solution, go to Solution Initialization. Select "Inlet 1" to initialize the domain, and press "Initialize".

Go to Monitors and edit the residual monitors. Note that both k and both have residual criteria; change it to 1E-06 for both.

Now go to Run Calculation. Run the solution until it all residuals appear to reach a steady state value. This should take a bit more than 5000 iterations, and you should end up with a residuals plot somewhat like the plot below. Note the fluctuations present in this plot.



Go to Step 5: Turbulent Jet Results

Go to all FLUENT Learning Modules