Supersonic Flow Over a Wedge - Numerical Solution

Author: Rajesh Bhaskaran, Cornell University

Numerical Solution

Solution Methods

In the *Outline* window, select ***Solution Methods*** to open the *Solution Methods* window. Under *Spatial Discretization*, ensure that the option for *Flow* is specified as ***Second Order Upwind.*** (Hint: If you don't see *Flow,* go back and ensure that you changed the solver from Pressure-based to Density-based under the *General* tab.)

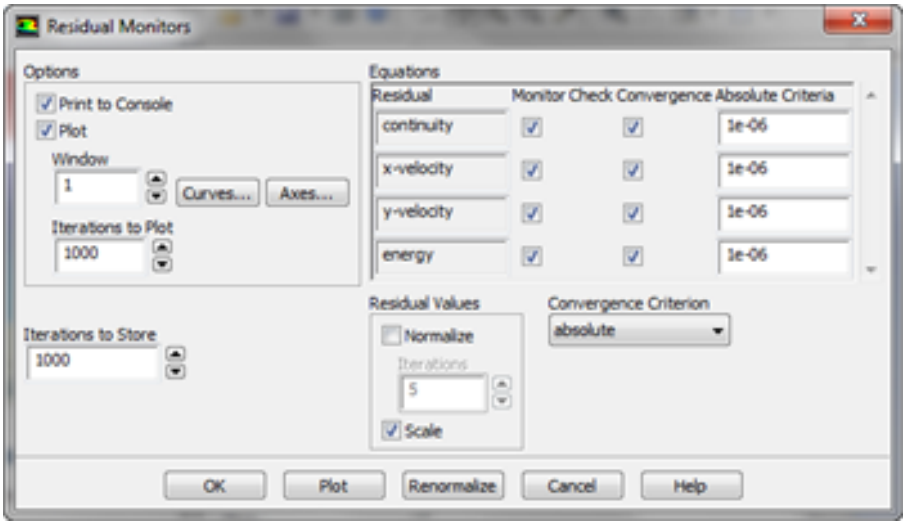
Solution Controls

In the *Outline* window, select ***Solution Controls*** to open the *Solution Controls* window. (Note: In later versions, this is simply named *Controls.*) Ensure that the ***Courant Number*** is set to ***5.0***.

The Courant number can be considered a non-dimensionalized timestep. The density-based solver obtains the steady-state solution by starting with the initial guess and marching in pseudo-time until convergence is obtained. The Courant number controls the time step the solver uses. The larger it is, the faster the solution will converge but it will not be very stable and can diverge. The smaller it is, the slower it is to reach convergence but the solution is much more stable.

Monitors

In the *Outline* window, click ***Monitors*** to open the *Monitors* outline. In the *Monitors* outline, double-click ***Residuals***. This will open the *Residual Monitors* window. We want to change the convergence criteria for our solution. Under *Equations* and to the right of ***Continuity***, change the ***Absolute Criteria*** to `1e-6`. Repeat for ***x-velocity***, ***y-velocity***, and ***energy***, then press ***OK***.



Solution Initialization

In the *Outline* window, select ***Solution Initialization***. (Note: In later versions, this is simply named *Initialization.*) We need to make an "Initial Guess" of the solution so FLUENT can iterate to find the final solution. In the *Solution Initialization* window, select ***Standard Initialization***, then under ***Compute from***, select ***farfield*** from the drop down box. Check to see that the values that are generated match the values that we input, then press ***Initialize.***

Run Calculation

In the *Outline* window, select ***Run Calculation***. Change the ***Number of Iterations*** to `4000`. Double-click ***Calculate*** to run the calculation. It should take a few minutes to solve. After the calculation is complete, save the project. Do not close FLUENT.

If your solution is having convergence issues, try to use Third Order MUSCL under Solution Methods > Spatial Discretization.

Third Order MUSCL helps to converge the solution especially if you are seeing many oscillations between a few values in your residuals.

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules