

# Copy of Supersonic Flow Over a Wedge - Verification & Validation

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

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

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

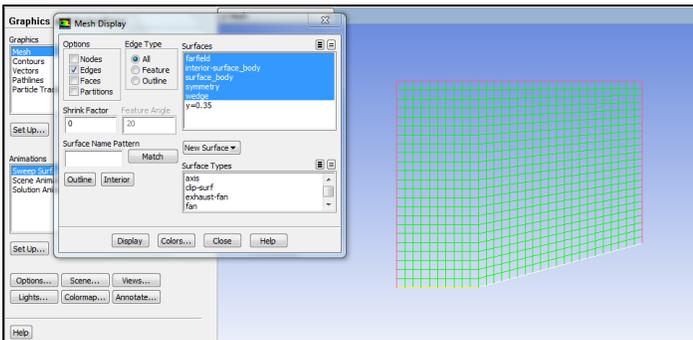
[Comments](#)

## Verification & Validation

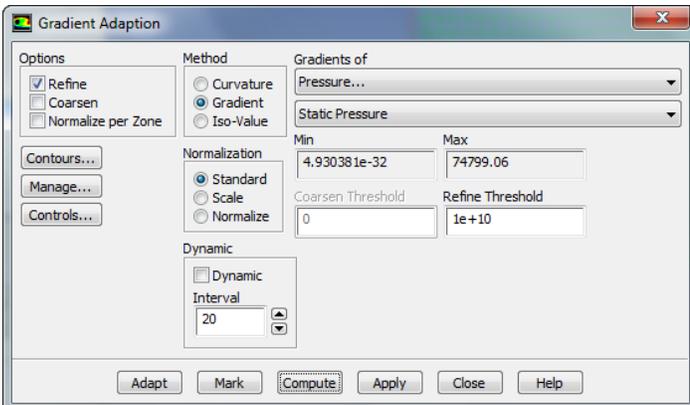
### Verification

#### Adapt the Mesh

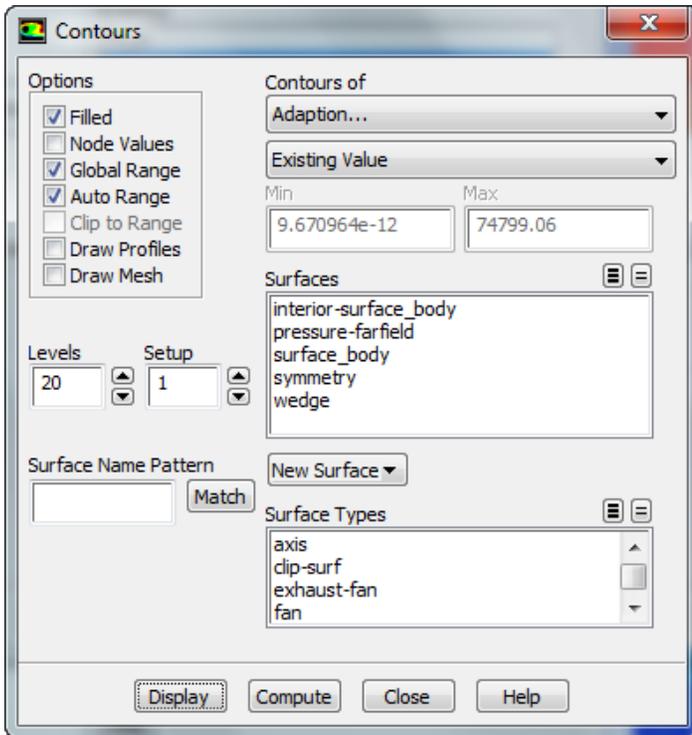
In order to test our simulation for convergence, we will refine the mesh. Refining the mesh will allow us to make sure that the results we are calculating are independent of the mesh. However, instead of refining the mesh everywhere (which would be wasteful, as most of the area of the domain far away from the shock has constant values), we will use our results to refine our mesh. Specifically, we are going to use the gradient of the pressure to determine where to refine the mesh. First, let's take a look at our mesh. In the *Outline window*, select **Graphics and Animations**, under *Graphics*, select **Mesh**, then press **Setup**. Select all of the surfaces (except  $y=0.35$ ) and press **Display**. This will display the current mesh.



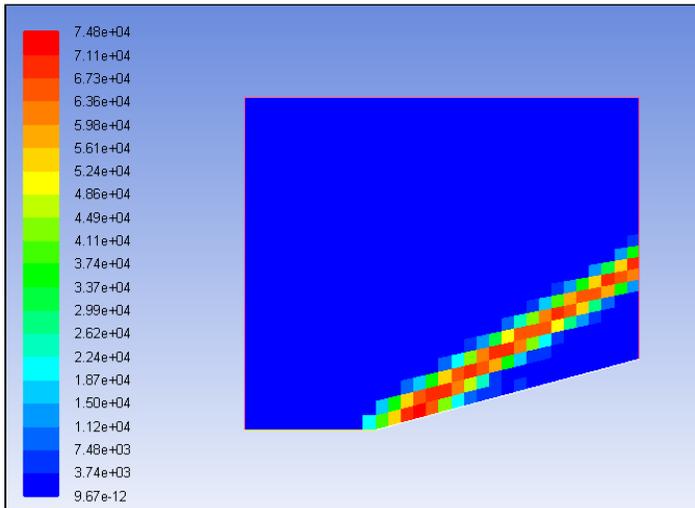
You may now close the *Mesh Display* window. In the menu bar, go to **Adapt > Gradient**. Under *Options* uncheck **Coarsen**. Under *Method* click the radio box next to **Gradient**. Ensure that **Gradients of > Pressure... Static Pressure** are selected. Then press **Compute**.



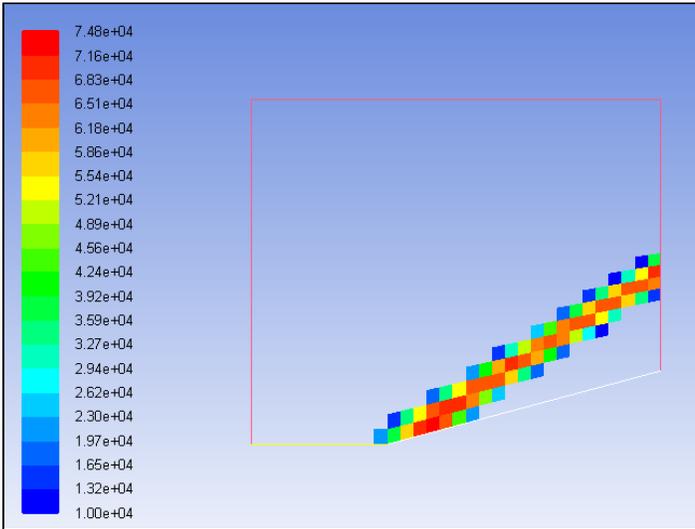
This will compute the maximum and minimum gradients of static pressure. Next, we need to pick a threshold. In order to do this, click on **Contours...** This will open the familiar *Contours* window. In the *Contours* window, select **Contours of Adaption... Existing Value**, then press **Compute**. This will populate the *Contours* menu with the values we were viewing for adaption, in this case, gradient of the static pressure. Also, make sure to uncheck **Node Values** (by first checking **filled**).



Finally, press **Display** to display the contours of static pressure gradient.

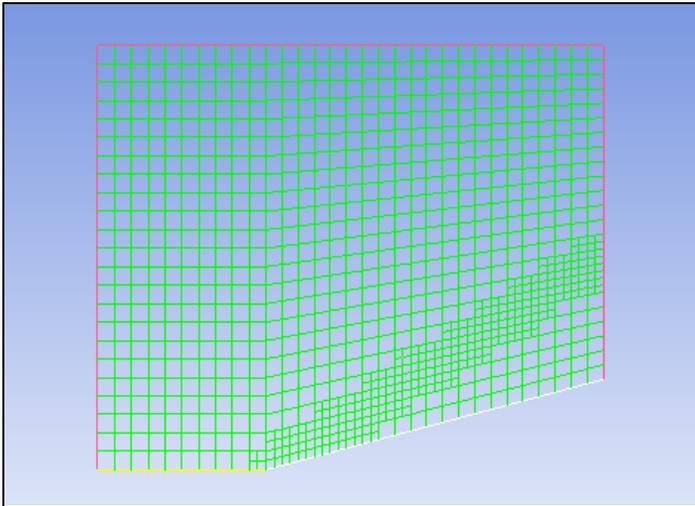


Our goal is to discover the areas of high gradient, and mark those areas for adaption. Using the contour graph we can then find the areas of high pressure gradient that we want to refine. In the *Contours* window, uncheck **Auto Range**. This will allow us to specify the range of gradients we can see in the graph. Change the **Min** value to **10000**. Then press **Display**. You will see that the pressure gradients above 10000 generally mark the area of high pressure gradient quite well. We will use 10000 as our threshold for refinement.



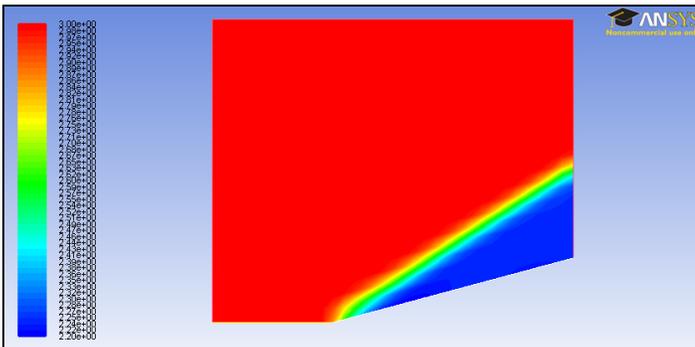
Return to the *Gradient Adaption* window, and enter **10000** into *Refine Threshold*. Then press **Mark**. The *print* window will inform us we have marked 70 elements for refinement. Next, press **Adapt**. You will be asked if you want to change the mesh. Press **Yes**.

It will seem like nothing has changed, but that is because we need to re-display the mesh in order to see the adaption. The new mesh should look something like this.

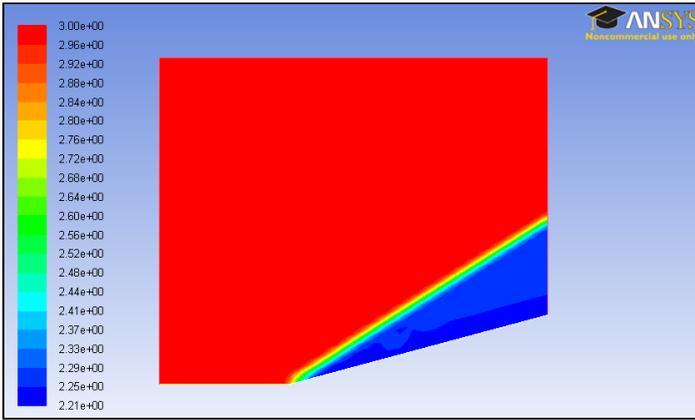


Notice that the area surrounding the shock was refined. Now, re-initialize the solution, (Solution Initialization > Compute From Farfield > Initialize), and rerun the solution (you will also need to increase the number of iterations – I recommend 150).

Now, once again, plot the contours of the mach number. Below is a comparison of the mach number results from the original mesh and the refined mesh.



Original Mesh



Refined Mesh

The most striking difference between the two results is the thickness of the shock. Notice that for the refined mesh, the shock is less thick than for the original mesh. This shows that the refined mesh is converging towards the real case.

### Comparison to Analytical Solution

In order to verify our simulation, we need to compare our results to either an analytical solution or an experiment. Below is a table comparing the values from the simulation with the calculations from the pre-analysis.

	Mach Number	Static Pressure (atm)	Shock Angle (degrees)
<b>Theory Value</b>	2.254	2.824	32.22
<b>FLUENT Solution</b>	2.243	2.803	34.99
<b>Percent Difference</b>	0.8%	0.7%	8.2%

As we can see from the table, we are getting fairly good matching between the computation and analytical approaches. From this we can build our trust in our simulation.

### Save Project

Save the project using *File > Save*. Copy *wedge.wbpj* and the associated *wedge\_files* folder to a flash drive. You will need both entities to resume the project.

Alternately, you can select *File > Archive* and save the project as one file called *wedge.wbpz*. When prompted, select the option to save *Result/Solution* also. You will then need to save only this file. This is also convenient to e-mail the project. Double-clicking on the *wedge.wbpz* file will resume the project.

[Continue to Exercises](#)

[Go to all FLUENT Learning Modules](#)