Supersonic Flow Over a Wedge - Physics Setup

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

Physics Setup

Note that the look and arrangement of the menu setup in FLUENT has changed a bit since this tutorial was created. If something has changed in the menu setup from the description below, you can often find the equivalent options by poking around in the menus.

Update the Project and Open FLUENT

Before we open FLUENT, we need to right-click on the Mesh cell and choose Update. You should see a checkmark next to Mesh. Double-click Setup

4 Setup to open FLUENT.

Initial Settings

Double-Click Setup in the Workbench Project Page.

When the *FLUENT Launcher* appears, choose "Double Precision" under "Options" and then click *OK* as shown below. The Double Precision option is used to select the double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision, but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

E Fluent Launcher (Setting Edit C	nly)
ANSYS	Fluent Launcher
Dimension	Options Double Precision Processing Options Serial Parallel
	ancel <u>H</u> elp •

Problem Setup - General

Now, FLUENT should open. If you are running FLUENT on Apps on Demand, you may need to hit the Maximize button (highlighted with the arrow in the snapshot below) to scroll down the tree in the Outline View.

				3	-/	×
Pa	arallel Desig	🗘 🔶 🔺 🔍 Quick Sea				ANSYS
ices	Mesh Models	Turbo Model		Adapt		
h	💋 Dynamic Mesh	Enable	Refi	ine / Coarse	en	
rset	💢 Mixing Planes	ổ Turbo Topology				Surface
		D Turbo Create	ooo Mor	e	*	-
		Mesh				×

We will begin setting up some options for the solver. In the left hand window (the *Outline* window), under *Problem Setup*, select *General* by double clicking. The only option we need to change here is the type of solver. In the *Solver* window, select *Density-Based*. It is also a good idea to click the "Check" button under the Mesh options, as this will check if the mesh was created and transferred correctly.

Models

In the outline window, double-click *Models*. We will need to utilize the energy equation in order to solve this simulation. Under *Models* highlight *Energy* - *Off* and click *Edit...* Now, the *Energy* window will launch. Check the box next to *Energy Equation* and hit OK. Doing this turns on the energy equation.

We also need to change the type of viscosity model. Select Viscous - Laminar and click Edit.... Choose the Inviscid option and press OK.

Models
Models
Multiphase - Off Energy - On
Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Acoustics - Off
Edit

Materials

In the Outline window, double-click Materials. In the Materials window, highlight Fluid > Air, and click Create/Edit.... this will launch the Create/Edit Materials window; here we can specify the properties of the fluid. Set the Density to Ideal Gas, the Specific Heat to 1006.43, and the Molecular Weight to 28.966. When you have updated these fields, press Change/Create. Then click Close.

Create/Edit Materials		×
Name	Material Type	Order Materials by
air	fluid	Name
Chemical Formula	ELLENT Eluid Materials	Chemical Formula
	air	FLUENT Database
	Mixture	User-Defined Database
	none	*
Properties		
Density (kg/m3)	ideal-gas 🗸 Edit	
Co (Specific Heat) (ilko.k)		
cp (specific field) (j/kg·k)	constant Edit	
	1006.43	
Molecular Weight (kg/kgmol)	constant	
	28,966	
	•	
	Change/Create Delete Close Help	

Boundary Conditions

In the Outline window, double-click Boundary Conditions. We will now specify each boundary condition for the simulation.

Farfield

In the Boundary Conditions window, select farfield. Use the drop-down menu to change the Type to pressure-far-field. You will be asked to confirm the change, and do so by pressing OK. (Note: In newer versions, this step is not needed.) Next, a dialogue box will open with some parameters we need to specify. Change the Gauge Pressure (Pascal) to 101325, and Mach Number to 3.

Pressure Far-Field	×		
Zone Name			
Momentum Thermal Radiation Species UDS DPN	1		
Gauge Pressure (pascal) 101325	constant 🔹		
Mach Number 3	constant 💌		
X-Component of Flow Direction 1	constant 👻		
Y-Component of Flow Direction	constant 🔻		
OK Cancel Help			

Also, select the Thermal tab, and ensure that the temperature correctly defaulted to 300 K. When you are finished, press OK.

Wedge

In the Boundary Conditions window, select wedge. Use the drop-down menu to change the Type to wall.

Symmetry

In the Boundary Conditions window, select symmetry. Use the drop-down menu to change the Type to symmetry.

Operating Conditions

In the Boundary Conditions window, select the Operating Conditions button. Change the Gauge Pressure to 0. Then press OK

Operat	ing Conditions	×
Pressure		Gravity
	Operating Pressure (pascal)	Gravity
Reference	e Pressure Location	
X (m)	P	
Y (m)	° e	
Z (m)	0 P	
	OK Cancel Help	>

It is important to check the operating conditions. When setting the density in materials to *Ideal Gas*, FLUENT calculates the density using the absolute pressure. However, the pressure we specify is the gauge pressure, not the absolute pressure. FLUENT will use the absolute pressure to compute the density, therefore if we do not set the operating pressure to 0 our density will be incorrect for the flow field.

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules