Old (4/1/2020) Flat Plate Boundary Layer - Physics Setup

Author: Rajesh Bhaskaran, Cornell University Problem Specification

- 1. Pre-Analysis & Start-Up
- 2. Geometry
- 3. Mesh
- 4. Model Setup
- 5. Numerical Solution
- 6. Post Processing
- 7. Verification & Validation
- 8. Part II: Flat Plate Convection

Frequently Asked Questions

Exercises

Comments

Physics Setup

Physical Setup

Your current Workbench Project Page should look comparable to the following image. Regardless of whether you downloaded the mesh and geometry files or if you created them yourself, you should have checkmarks to the right of Geometry and Mesh.



The question mark to the right of right of the Setup cell is normal and indicates that the Setup process has not yet been completed.

Launch Fluent

Double click on Setup in the Workbench Project Page which will bring up the FLUENT Launcher. When the FLUENT Launcher appears change the options to "Double Precision", 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 Only)	
ANSYS	FLUENT Launcher
Dimension 30 20 Display Options Display Mesh After Reading Embed Graphics Windows Workharch Color Scheme	Options Double Precision Use Job Scheduler Processing Options Secial Parallel
Do not show this panel again	Show More >>
<u>Ok</u>	ancel Help •

https://confluence.cornell.edu/download/attachments/118475240/DoublePrecision_Full.png

Twiddle your thumbs a bit while the FLUENT interface starts up. This is where we'll specify the governing equations and boundary conditions for our boundary-value problem. On the left-hand side of the FLUENT interface, we see various items listed under Problem Setup. We will work from top to bottom of the Problem Setup items to setup the physics of our boundary-value problem. On the right hand side, we have the Graphics pane and, below that, the Co mmand pane.

Check and Display Mesh

A:FlatPlate FLUENT [2d, dp, pbns, lam] [ANSYS Academic Teaching Define File Mesh Solve Adapt Surface Display Report Check (+)۲ Info • Quality Prob Size Polyhedra • Memory Usage Mo Merge... Zones Ma Ph Separate) Partitions Ce Fuse.

First, the mesh will be checked to verify that it has been properly imported from Workbench. In order to obtain the statistics about the mesh (Click) Mesh > Info > Size, as shown in the image below.

Then, you should obtain the following output in the Command pane.

Level	Cells	Faces	Nodes	Partitions
0	3000	6110	3111	1

1 cell zone, 5 face zones.

The mesh that was created earlier has 3,000 elements(50 x 60). Note that in FLUENT elements are called cells. The output states that there are 3,000 cells, which is a good sign. Next, FLUENT will be asked to check the mesh for errors. In order to carry out the mesh checking procedure (Click) Mesh > Check as shown in the image below.

💶 A:	FlatPlat	te FLUEN	T [2d, d	o, pbns,
File	Mesh	Define	Solve	Adapt
	(Check		
	I	nfo	~	+
Probl Ge	F	olyhedra	1	+
Mo Ma	I	Merge		

You should see no errors in the Command pane. Now, that the mesh has been verified, the mesh display options will be discussed. In order to bring up the display options (Click) General > Mesh > Display as shown in the image below.

A:FlatPlate FLUENT [20	d, dp, pbns, lam] [ANS)	/S Academic Teaching Advanc
File Mesh Define So	lve Adapt Surface	Display Report Parallel
) 📸 🕶 🔜 👻 🔞 🔘	ି 🌆 🔍 🖉 🥒] 洗 閒 ▾ □ ▾
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	General Mesh Scale Display Solver	Check Report Quality
Dynamic Mesh Reference Values	Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative

The previous step should cause the Mesh Display window to open, as shown below. Note that the Named Selections created in the meshing steps now appear.

💶 Mesh Displa	y		×
Options Nodes Edges Faces Partitions	Edge Type a All Feature Outline	Surfaces far_field inlet interior-surface_body outlet plate	
Shrink Factor	Feature Angle]	
Surface Name Pa	Match	New Surface Surface Types axis dip-surf exhaust-fan fan	
	Display Colo	rs Close Help	

https://confluence.cornell.edu/download/attachments/118771050/MeshDispMenu_Full.png

You should have all the surfaces shown in the above snapshot. Clicking on a surface name in the Mesh Display menu will toggle between select and unselect. Clicking Display will show all the currently selected surface entities in the graphics pane. Unselect all surfaces and then select each one in turn to see which part of the domain or boundary the particular surface entity corresponds to (you will need to zoom in/out and translate the model as you do this). For instance, if you select far_field, inlet, and plate and then click Display you should then obtain the following output in the graphics window.

Now, make sure all 5 items under Surfaces are selected. The button next to Surfaces selects all of the boundaries while the button deselects all of the boundaries at once. Once, all the 5 boundaries have been selected click Display, then close the Mesh Display window. The rectangle displayed in the graphics window corresponds to our solution domain. Some of the operations available in the graphics window to interrogate the geometry and mesh are:

Translation: The model can be translated in any direction by holding down the Left Mouse Button and then moving the mouse in the desired direction.

Zoom In: Hold down the Middle Mouse Button and drag a box from the Upper Left Hand Corner to the Lower Right Hand Corner over the area you want to zoom in on.

Zoom Out: Hold down the Middle Mouse Button and drag a box anywhere from the Lower Right Hand Corner to the Upper Left Hand Corner.

Use these operations to zoom in and interrogate the mesh.

Define Solver Properties

In this section the various solver properties will be specified in order to obtain the proper solution for the flat plate boundary layer flow. First, the Energy Model parameters will be specified. In order to open the Energy Model Options Models > Energy-Off > Edit..., as shown in the following image.

) 🐸 🕶 🔛 💌 🔘 🗍	S ⊕ Q Q ∥ Q X + □ +
Problem Setup	Models
General Materials Phases Cel Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Models Multiphase - Off Ensrgy-Off Yiscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Edt

For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining the temperature distribution. We will not deal with temperature in this example. So leave the Energy Equation set to off and click Cancel to exit the menu. Next, the Viscous Model parameters will be specified. In order to open the Viscous Model Options Models > Viscous - Laminar > Edit..., as shown below.



https://confluence.cornell.edu/download/attachments/118771050/ViscModel_Full.png

By default, the Viscous Model options are set to laminar, so no changes are needed. Click Cancel to exit the menu.

Define Material Properties

Now, the properties of the fluid that is being modeled will be specified. The properties of the fluid were specified in the Problem Specification section. In order to create a new fluid (Click) Materials > Fluid > Create/Edit... as shown in the image below.

Problem Setup General Models Intervisit Phases Cell Zone Conditions Boundary Conditions Meth Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Intislization Calculation Activities Run Calculation	File Mesh Define So	S 🔂 @ @ / @ 次 III ▼ 🗆 ▼
Results Graphics and Animations Plots Reports	Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Mesh Toterfaces Dynamic Mesh Mesh Toterfaces Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Materials Materials Plad aluminum aluminum

daust.	Material Type		Order Materials by
8ir	Rud		Name Ordenning formula
henical Formula	PLUENT Plud Materials		FLUENT Database
	Minture none		User-Defined Database
Density (kg/m3) Viscosity (kg/m-s)	constant	1	
	terdi	Ξ.	

In the Create/Edit Materials menu set the Density to 1kg/m^3 (constant) and set the Viscosity to 1e-4 kg/(ms) (constant) as shown in the image below.

https://confluence.cornell.edu/download/attachments/118771050/Mat1_Full.png

Click Change/Create. Close the window.

Define Boundary Conditions

At this point the boundary conditions for the four Named Selections will be specified. The boundary condition for the inlet will be specified first.

Inlet Boundary Condition

In order to start the process (Click) Boundary Conditions > inlet > Edit... as shown in the following image.

A:FlatPlate FLUENT [20	d, dp, pbns, lam] [ANSYS Academic Teaching Advanced
File Mesh Define 30	Call and the surface Display Report Parallel V
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods	S S Q Q A Q C TH - C
Solution Prethods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	
Graphics and Animations Plots Reports	Edt

Note that the Boundary Condition Type should have been automatically set to velocity-inlet. FLUENT guesses boundary conditions based on the label of the named selections. Now, the velocity at the inlet will be specified. In the Velocity Inlet menu set the Velocity Specification Method to Components, and set the X-Velocity (m/s) to 1 m/s, as shown below.

Velocity Inlet		X	
Zone Name			
met			
Momentum Thermal Radia	tion Species DPM M	ltiphase UDS	
Velocity Specification Method	Components	•	
Reference Frame	Absolute	•	
X-Velocity (m/s)	1	constant 💌	
Y-Velocity (m/s)	0	constant 👻	
	OK Cancel Help		

https://confluence.cornell.edu/download/attachments/118771050/SetInletBC_Full.png

Then, click OK to close the Velocity Inlet menu.

Outlet Boundary Condition

First, (Click) Boundary Conditions > outlet > Edit... as shown in the image below.

File Mesh Define So	live Adapt Surface Display Report Parallel V 등 🔂 ⓒ ① 🌶 🔍 🔍 八 🏾
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Dynamic Mesh Mesh Inter Fraces Dynamic Mesh Reference Values Solution Methods Solution Intellization Calculation Intellization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Boundary Conditions Zore Type Netion surface_body plate Phase Type D Type D Type D Tope Type D Tope Type D Tope Type D Tope Type D Tope Type D Tope Tope Type D Tope Tope Tope Type D Tope

Note that the Boundary Condition Type should have been automatically set to pressure-outlet. For the outlet boundary condition the gauge pressure needs to be set to zero. The default gauge pressure is zero, thus no changes need to be made. Close the Pressure-Outlet menu.

Plate Boundary Condition

A:FlatPlate FLUENT [20	I, dp, pbns, Iam] [ANSYS Academic Teaching Advanced
File Mesh Define So	ve Adapt Surface Display Report Parallel Vi
) 📸 • 🖬 • 📾 🛛	중 🔁 @ 🍳 🥒 🍭 🏷 🖷 - 🗖
Problem Setup	Boundary Conditions
General	Zone
Models	far field
Materials	inlet
Phases	interior-surface_body
Cell Zone Conditions	plate
Mesh Interfaces	prove
Dynamic Mesh	
Reference Values	
Solution	
Solution Methods	
Solution Controls	
Monitors	
Solution Initialization	
Calculation Activities	
Run Calculation	
Results	
Graphics and Animations	Phase Type ID
Plots	mixture v wall v 8
hepot to	
	Copy Profiles
	Parameters Operating Conditions
	Display Mesh

https://confluence.cornell.edu/download/attachments/118771050/WallBCC_Full.png

If the Boundary Condition Type is not set to wall, then set it to wall. We will use the default setting for the wall boundary condition, thus no changes are needed.

Far-Field Boundary Condition

For the far_field, set the Boundary Condition Type to symmetry, as shown below.

File Mesh Define So	lve Adapt Surface Display Report Parallel V
😂 • 🖬 • 🗃	요면 🖉 🍭 大 🖷 - 🗖
Problem Setup	Boundary Conditions
General Models Materials Phases Cel Zone Conditions Datadasi Conditions Mash Interfaces Dynamic Neish Reference Values Solution Nethoda Solution Controls Monitors Solution Initialization Calculation AntiVities Run Calculation	Zone fine field interior-surface_body outlet plate
Carphics and Animations Plots Reports	Phose Type ID moture v and v 6 Edit C enhand v 6 Parameters C enhand fan Deplay Meth C enhand fan Deplay Meth C enhand fan presure viet presure viet presure viet

https://confluence.cornell.edu/download/attachments/118771050/symmbc_Full.png

In the first dialog box click yes and in the second dialog box leave the name as is and click OK. The symmetry boundary conditions sets the velocities normal to the boundary equal to zero.

At this point save your work in the FLUENT Window by clicking the save button,

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules