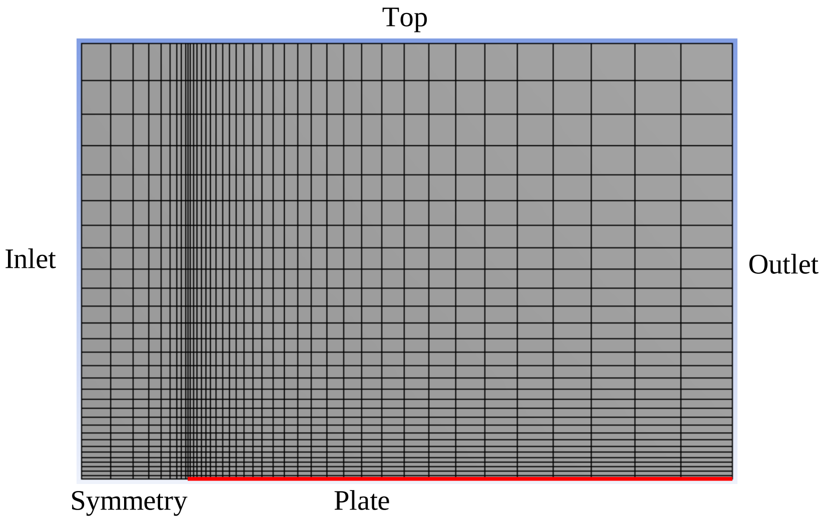
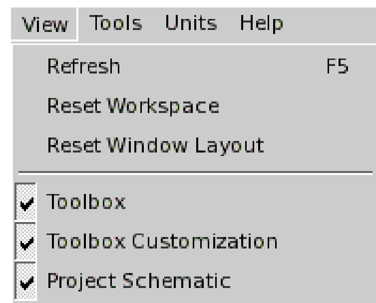
ANSYS Workbench Tutorial – Boundary Layer on a Flat Plate

Authors: Scott Richards, John M. Cimbala, and Keith Martin, Penn State University
Latest revision: 18 May 2010

Introduction

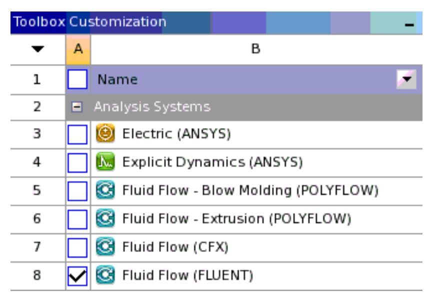
This tutorial provides instructions for creating a simple two-dimensional mesh that is used to simulate the boundary layer on a flat plate. The final mesh is shown below, along with labels for the edges.



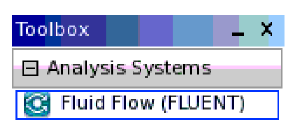


Set Workbench options for a new FLUENT project

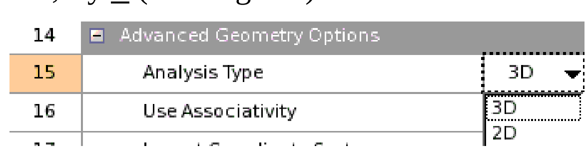
1. This tutorial assumes that ANSYS Workbench is running but no projects are open. If it is not running, open it before proceeding to the next step.
2. Under *View*, make sure that "Toolbox", "Toolbox Customization", and "Project Schematic" all have a check mark next to them [some versions may have a different look than what is shown here].
   - ✚ Check marks can be inserted by placing the cursor over the menu item and LMB (click the left mouse button).
   - 💡 Placing a check mark next to a menu item opens its window. If you close the window, the corresponding check mark will be removed from the *View* menu.



3. In *Toolbox Customization* under *Analysis Systems*, verify that "Fluid Flow (FLUENT)" has a check mark next to it.
   - 💡 You may need to stretch Column B to see the entries.
   - ✚ If Column A is not shown in *Toolbox Customization*, **1** located to the left of "Name".
   - ✚ If the box to the left of "Fluid Flow (FLUENT)" is unchecked, LMB in the box next to "Fluid Flow (FLUENT)".
   - 💡 To de-clutter your Workbench workspace, close or minimize the *Toolbox Customization*; it is not needed after this step.



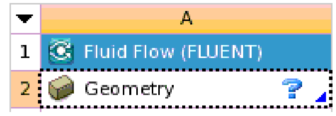
4. In the *Toolbox*, not *Toolbox Customization*, Fluid Flow (FLUENT) and hold the LMB to drag it into the box that will appear in *Project Schematic*. If everything works correctly, there will now be a *Fluid Flow (FLUENT)* template in *Project Schematic*.
   - ✚ If there are no Analysis Systems visible in the *Toolbox*, try + (clicking "+") to the left of *Analysis Systems* under the *Toolbox*.



5. R-Geometry-Properties. In the new window under *Advanced Geometry Options*, and to the right of option *Analysis Type*, 3D and then 2D from the

drop-down menu. This limits the analysis to two dimensions.

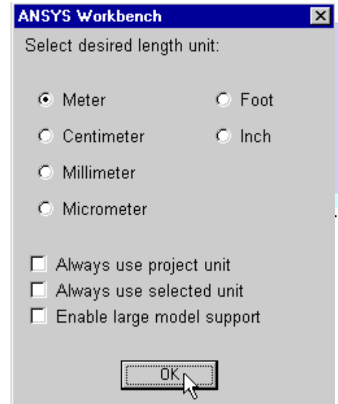
6. **<u>Geometry</u>** (double click “Geometry”) in the newly created *Fluid Flow (FLUENT)* template to open *ANSYS DesignModeler*.



Create required edges in DesignModeler (DM)

1. At startup DM will prompt the user to select a desired length unit. <u>LMB</u> on the circle next to “Meter”, then <u>OK</u>.

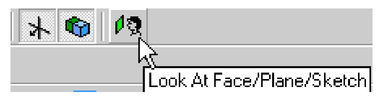
   If this prompt does not appear, it means that your version of DM was set to always use a certain length unit. Although your length unit may not be meters, you can continue because the instructions are the same for any system of units.



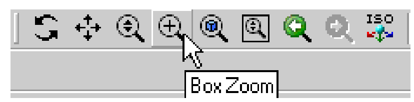
2. In *Tree Outline*, <u>XYPlane</u>. Then in *Display Toolbar* (near the top), <u>Look at Face/Plane/Sketch</u> .

   The name of a toolbar button will be displayed if you hover the mouse cursor over the button.

3. <u>Box Zoom</u> from the *Display Toolbar* . Use the LMB to drag a box from the 3rd quadrant, near the origin, to the approximate coordinates (1,1). The resulting view of the first quadrant will be perfect for constructing the sketch for this tutorial.



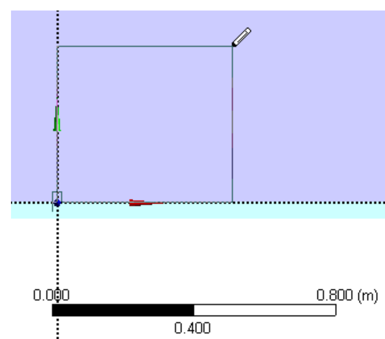
   If the desired view wasn’t obtained, you can draw another box to zoom in further or you can <u>Look at Face/Plane/Sketch</u> to redo this step from the beginning.



   The coordinates of your cursor are sometimes displayed at the bottom right corner of *Graphics*.

4. *<u>Sketching</u>* which is located near the bottom of *Tree Outline*. This will bring up the *Draw* tab of *Sketching Toolboxes*.

5. <u>Rectangle</u> and hover over the origin. <u>LMB</u> when a “P” is displayed. Move the cursor to make a rectangle in the 1st quadrant that is about .5 m by .5 m. <u>LMB</u> to place the other corner of the rectangle.

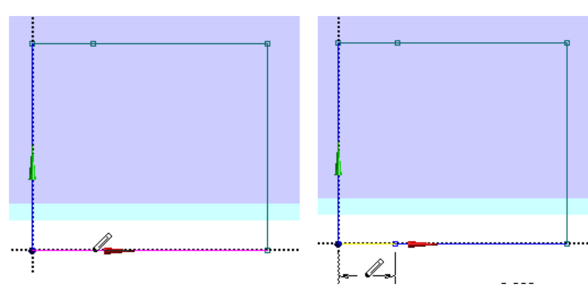


   The “P” means that one of corners of the rectangle will be constrained to the origin.

   Note: The location of the 2nd corner of the rectangle isn’t critical because you will supply dimensions for the length and height of the rectangle later.

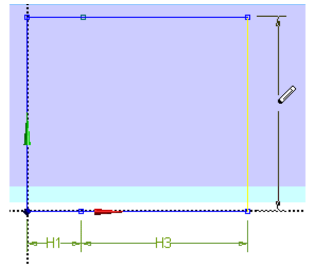
   If you make a mistake, <Esc> to exit the selected drawing tool or <u>Undo</u> which is located above *Sketching Toolboxes*.

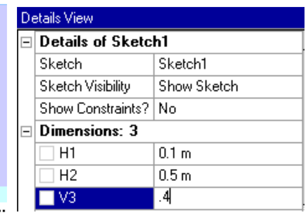
Split and dimension the edges



1. In *Sketching Toolboxes* *<u>Modify</u>*-<u>Split</u>. About ¼ of the way from the left edge, <u>LMB</u> on the top edge and then again in the same location on the bottom edge.
2. In *Sketching Toolboxes*, *<u>Dimensions</u>* [when *Dimensions* opens, the default is “General”, which is what we want], and <u>LMB</u> on the short section of the bottom edge. Then, move the cursor below the bottom edge and <u>LMB</u> again to place the dimension at the cursor’s location (see figures above).
3. <u>LMB</u> on the long section of the bottom edge. Move the cursor below this edge and <u>LMB</u> again to place the dimension. Repeat this process to dimension the right edge of the rectangle.

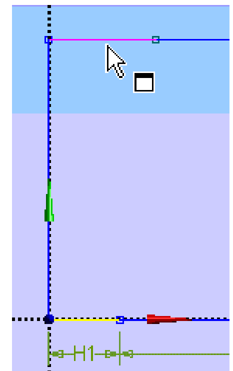
4. In the *Dimensions* section of *Details View*, the first dimension listed corresponds to the length of the symmetry boundary. Set this value to “0.1” m. The second dimension corresponds to the length of the plate; set it to “.5” m. The last dimension corresponds to the rectangle height; set it to “.4” m.





   - If you mess up at any point, Undo .

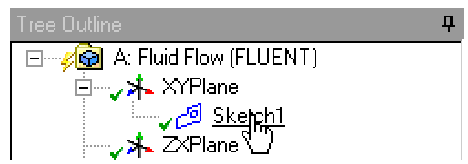
5. In *Sketching Toolboxes*, *Constraints*-Equal Length. LMB on the short section of the rectangle’s bottom edge that will serve as the symmetry boundary and then again on the corresponding short section of the top edge.



   - To view applied constraints, in *Details View*, **No**, which is located to the right of *Show Constraints*. In the present case, two of the edges (line segments) should have an “Equal Length” constraint listed.
   - Constraining one of the corners of the rectangle to the origin, specifying the three linear dimensions, and constraining the short sections of the top and bottom edges to be equal length are necessary steps to fully define the sketch.

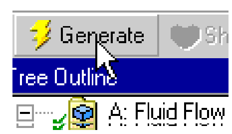
Create a surface from the sketch

1. *Concept* (in DM’s main menu bar)-Surfaces from Sketches then Sketch 1 from *Tree Outline* under *XYPlane*, and Apply in the *Details View* to set “Sketch 1” as the *Base Object*.

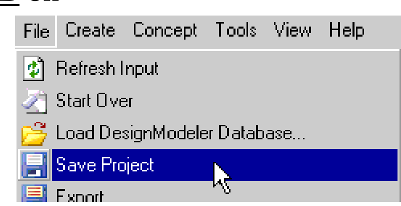


   - If no sketches are visible, + (click “+”) to the left of “XYPlane” in *Tree Outline*.
   - If the sketch that will be used as the *Base Object* wasn’t the first sketch to be created, DM will assign it a number other than “1”.
   - The sketch must be turned into a surface because ANSYS Workbench does not allow sketches to be meshed.
   - The *Base Object* refers to the sketch that will be made into a surface.

2. The default settings are acceptable so Generate from the toolbar near the top to create the surface; the surface should turn gray.

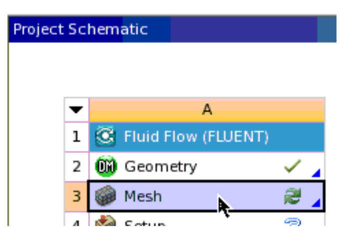


   - If DM displays a red “!” next to “Extrude1” in *Tree Outline*, RMB on “Extrude1” then Show Errors or Warnings to find out the reason why the extrusion couldn’t be generated.

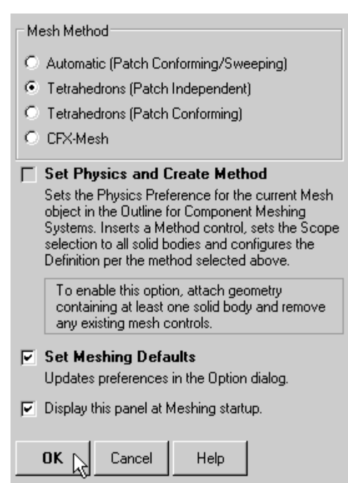


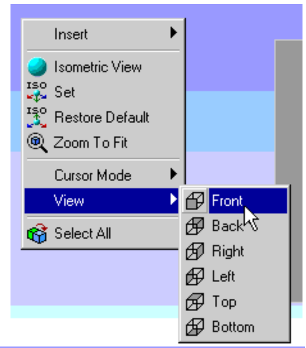
3. In the upper left corner of the *DM*, File-Save Project. After selecting an appropriate folder to save the project in, enter “FlatPlate” for the *File Name*, Save. Close the *DM* window.
   - Back in *Project Schematic*, “Geometry” should now be checked, indicating it is complete.

Set meshing options and label the edges



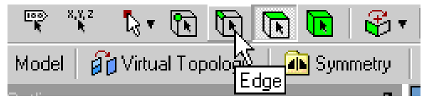
1. Open *ANSYS Meshing* by **Mesh** in *Project Schematic*.
2. From the *Meshing Options* window that opens, LMB on the circle next to “Tetrahedrons (Patch Independent)” to set the *Mesh Method*, then OK.
   - Depending on the size of your window, you may need to scroll down to find the OK button.



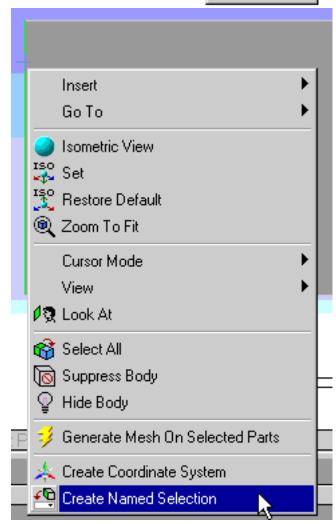


- The *Physics Preference* was automatically set to *CFD* when "Fluid Flow (FLUENT)" was chosen for the *Analysis System*. Thus, it is not necessary to specify a preference in *Meshing Options*.

3. From anywhere in the *Graphics* window, RMB-View-Front. This will orient the geometry so it will be easier to work with.
4. From the *Selection Toolbar*, which is located near the top of the *ANSYS Meshing* window, Edge to change the selection filter from "Face" to "Edge".

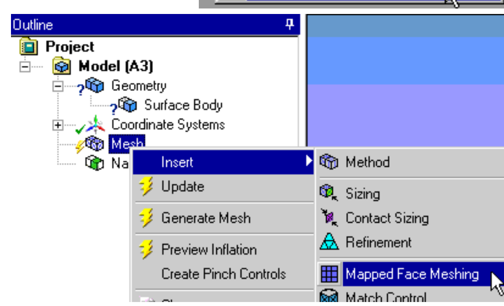


5. Place the cursor over the left edge of the rectangle. When the edge changes to a dotted red line, LMB to select it. The edge will turn green to indicate that it has been selected. Then RMB again in the same spot, Create Named Selection, and enter "inlet" as the name for that edge; OK.

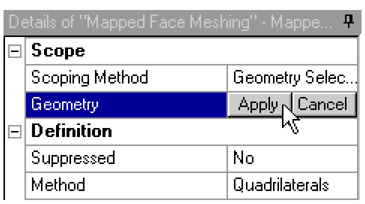


6. Repeat the process in the previous step to name the lower left edge "symmetry", the lower right edge "plate", and the right edge "outlet".
7. Name the two edges that form the top of the rectangle "top_left" and "top_right".
   - The two top edges could be combined into one named selection, but it would make sizing the grid more complicated.
8. In *Outline*, Named Selections to see a list of all the edges you have named. There should be a total of six.

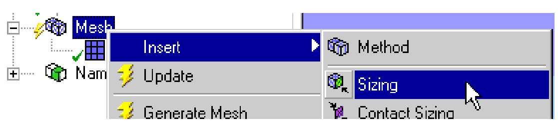
Insert a mapped face mesh and sizing controls



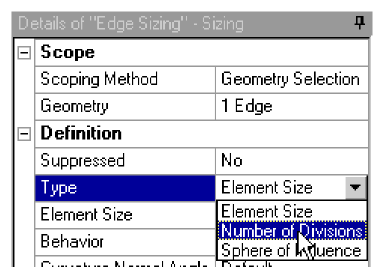
1. From *Outline*, R-Mesh-Insert-Mapped Face Meshing.
2. LMB on the front surface of the geometry. In *Details of "Mapped Face Meshing"*, Apply to make this surface the *Geometry* selection. Check that *Method* is set to "Quadrilaterals".



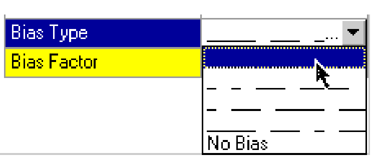
3. R-Mesh-Insert-Sizing. From the *Display Toolbar*, Edge, then LMB on the left edge that we had labeled "inlet" and Apply.



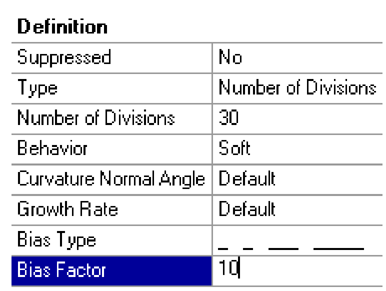
   - The labels disappear when you open the sizing window.



4. Change *Type* from "Element Size" to "Number of Divisions", then set *Number of Divisions* to "30".



5. For *Bias Type*, No Bias then LMB on the down arrow to display the available *Bias Types*. LMB the first option available in the drop down menu. Enter "10" for the value of *Bias Factor*.

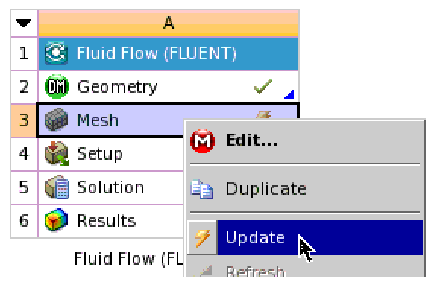


6. Repeat the procedure of the last three steps to put 30 divisions on the right edge that we labeled "outlet" with one exception: for *Bias Type*,

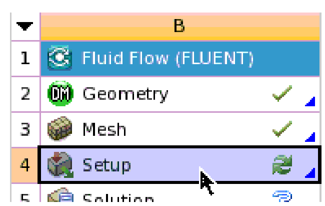
LMB the second option available from the drop down menu instead of the first, or else the bias will be in the wrong direction.

7. Put 10 divisions on the bottom left edge that we labeled "symmetry". Choose the first *Bias Type* from the drop-down menu such that a finer mesh is produced near the leading edge of the plate. Enter a *Bias Factor* of 15.
   - Try different Bias Type options to see what happens. If you are not careful, your mesh could have clustering around the wrong side of the edge.
8. Repeat the procedure from the previous step for the top left edge that we labeled "top_left", but use the second *Bias Type* on the drop-down menu.
   - Choose *Bias Type* for each edge such that clustering occurs near the plate's leading edge.
9. Lastly, put 30 divisions on the two remaining edges, "plate" and "top_right". Use a *Bias Factor* of 15 with the *Bias Type* that will give a finer mesh near the leading edge of the plate. At this point there should be a total of six *Edge Sizing* entries listed in *Outline*.
10. R-Mesh- Generate Mesh. This generates a mesh made up of rectangular elements that get progressively finer (the mesh is clustered) near the leading edge of the plate.
    - The intentional use of finer elements near the leading edge of the plate helps FLUENT resolve the boundary layer along the plate.
11. In the upper left corner of the *Meshing* window, File-Save Project then close *ANSYS Meshing*.

Update mesh and launch FLUENT



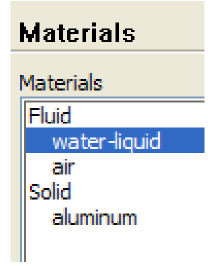
1. In the *Project Schematic* of ANSYS Workbench, R-Mesh from the analysis template, then Update. A check mark should now appear to the right of both *Geometry* and *Mesh*.
2. **Setup** from the *Project Schematic* to open *FLUENT Launcher*. In *FLUENT Launcher*, verify that the box next to "Double Precision" is selected, then OK.
   - If the box next to "Double Precision" is not checked, LMB in the box to select it before clicking OK.
   - Some older versions may give an error. If you get an error, close ANSYS Workbench, reopen it, and repeat Step 2.
3. The next screen will be the main *FLUENT Window* with your mesh in the *Graphics Window*.



Generate lines at *x* = 0.10 m and at *x* = 0.30 m

1. The boundary layer profile will be examined in detail at three *x* locations along the flat plate, namely at *x* = 0.10 m, 0.30 m, and 0.50 m. The last *x* location is the outlet of the domain, but lines need to be defined within FLUENT for the first two.
2. In the main *FLUENT* menu, Surface-Line/Rake. Type in the desired starting and ending *x* and *y* locations of the vertical line, i.e. a vertical line going from (0.1, 0) to (0.1, 0.4).
3. The *New Surface Name* should be assigned at this point. It is suggested that this line be called "profile0.10" or something descriptive of its intended purpose. Create to create the line.
4. Similarly, create a line at *x* = 0.30; a suggested label is "profile0.30". Close *Line/Rake Surface*.
5. To view these newly created lines, Display-Mesh. Under "Surfaces", LMB the default interior called "interior-surface_body" to unselect it. LMB the names of the newly created lines to select them instead. Display. The lines should be visible at the appropriate locations. If not, create them again more carefully.
6. Close the *Mesh Display* window.

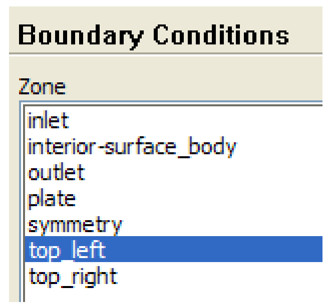
Define the fluid as liquid water



1. The default fluid is air, but we want to define the fluid as water. In the main *FLUENT* menu, Define-Materials-Create/Edit-FLUENT Database. Select water-liquid from the list of *FLUENT Database Materials*. Copy.
2. Write down the density and viscosity of liquid water. These properties are needed later to calculate Reynolds numbers, etc. Close.
3. Close the *Create/Edit Materials* window. *Caution*: This has added liquid water into the list of available fluids, but has not actually changed the fluid from air to water. This will be done next.

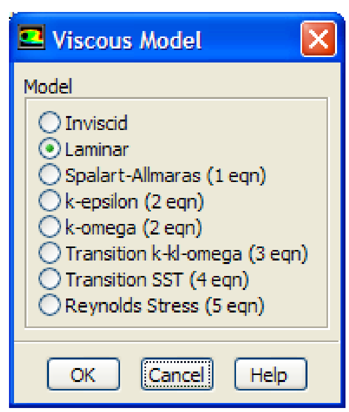
Define the cell zones and boundary conditions

1. In the main *FLUENT* menu, Define-Cell Zone Conditions. There should be only one zone, “surface_body”. Make sure *Type* is set as fluid. Edit. Select water-liquid instead of “air” as the material. OK.
2. Now the boundary conditions need to be specified. Previously, the boundary conditions were named, e.g., inlet, symmetry, etc., but actual values for inlet velocity, etc. were never defined. This must be done in FLUENT. In the main *FLUENT* menu, Define-Boundary Conditions.



3. The default boundary condition for the plate (stationary wall) is okay, so nothing needs to be done to it.
4. Likewise, the default boundary conditions for the symmetry plane (symmetry) and the outlet (pressure-outlet) are okay, so nothing needs to be done to them.
5. Select top_left, which is a wall by default. Edit. In the *Wall* window that pops up, Specified Shear. The default values of the specified shear are zero, which is what we want. We are specifying the top as a no-shear wall so that the fluid is constrained from flowing across the top boundary, but it can slip (it is a slip wall rather than a no-slip wall). OK.
6. We now copy this slip wall to the other top edge. Copy and select top_left as *From Boundary Zone*. Select top_right as *To Boundary Zones*. Copy-OK-Close.
7. Select inlet, which is the left side of the computational domain. Edit. In the *Velocity Inlet* window, change *Velocity Magnitude* to “0.1” m/s, and OK.

Set up some parameters and initialize



1. In the main *FLUENT* menu, Define-Models-Viscous - Laminar-Edit. Laminar flow is the default, so we really don't need to do anything here. Later on, however, you may need to specify turbulent flow calculations; this is where the turbulence models are specified in FLUENT, options of which are shown to the right. OK.
2. Now the convergence criteria need to be set. As the code iterates, residuals are calculated for each flow equation. Residuals represent a kind of average error in the solution – the smaller the residual, the more converged the solution. In the main *FLUENT* menu, Solve-Monitors-Residuals…. Edit. In the *Residual Monitors* window that pops up, make sure both Plot and Print to Console options are specified in the *Options* portion of the window. Here, “Print” refers to text printed in the main *FLUENT* window, and “Plot” causes the code to plot the residuals on the screen while the code is iterating.
3. Since there are three differential equations to be solved in a two-D incompressible laminar flow problem, there are three residuals to be monitored for convergence: continuity, *x*-velocity, and *y*-velocity. The default convergence criteria are 0.001 for all three of these. Experience has shown that

this value is generally not low enough for proper convergence. Change the *Convergence Criterion* for all three residuals from 0.001 to 1e-08 (highlight the number, and then type in the new value).

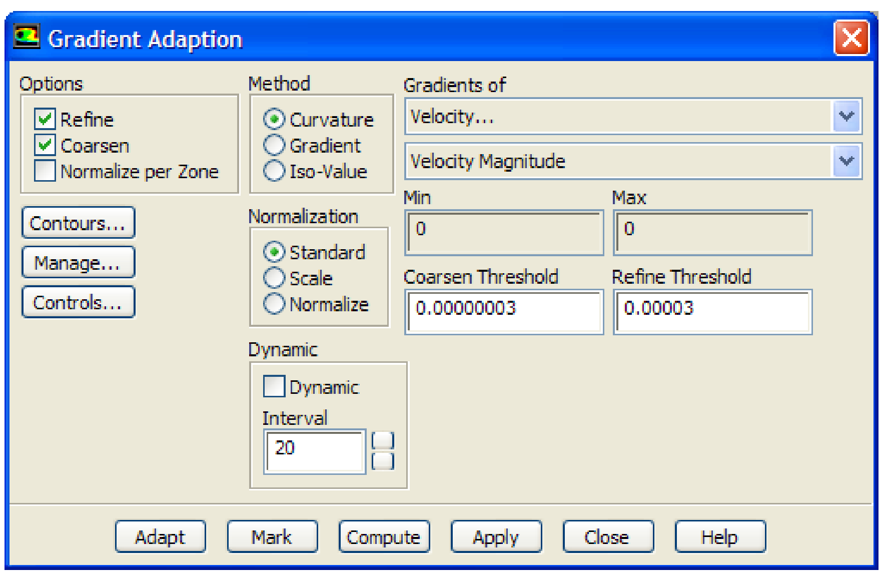
4. To apply the changes, OK, which will also close the *Residual Monitors* window.
5. In the main *FLUENT* menu, Solve-Initialization. The default initial values of velocity and gage pressure are all zero. These are good enough for this problem. Initialize.
6. At this point, and every so often, it is wise to save your work. In the main *FLUENT* menu, File-Save Project. Fluent writes two files in addition to the Workbench file: the *case* file (the grid plus all boundary conditions and other specified parameters) and the *data* file (the velocity and pressure fields calculated by the code).

Iterate towards a solution

1. In the main *FLUENT* menu, Solve-Run Calculation to open up the *Run Calculation* sub-window. Change *Number of Iterations* to 200, and Calculate. The main screen will list the residuals after every iteration, while the graphical display window will plot the residuals as a function of iteration number. The residuals may rise at first, but should slowly start to fall. It is normal for the residuals to fluctuate up and down. Do not be concerned if there are reverse flow warnings; these will disappear in time. When it is done or converged, OK.
2. At the end of these iterations, check to see how the solution is progressing. In the main *FLUENT* menu, Display-Graphics and Animations-Vectors-Set Up [the upper Set Up, not the lower one].
3. In the *Vectors* window that pops up, Display. The graphical display window will show the velocity vectors. Zoom in with the middle mouse [drag a box with MMB from lower left to upper right to zoom in on the box you select] to view the velocity field in more detail if desired. Is it starting to look like a boundary layer profile? Close the *Vectors* window.
4. 200 iterations are sufficient for the first round. Before iterating further, the mesh must be refined.

Refine the mesh and iterate some more

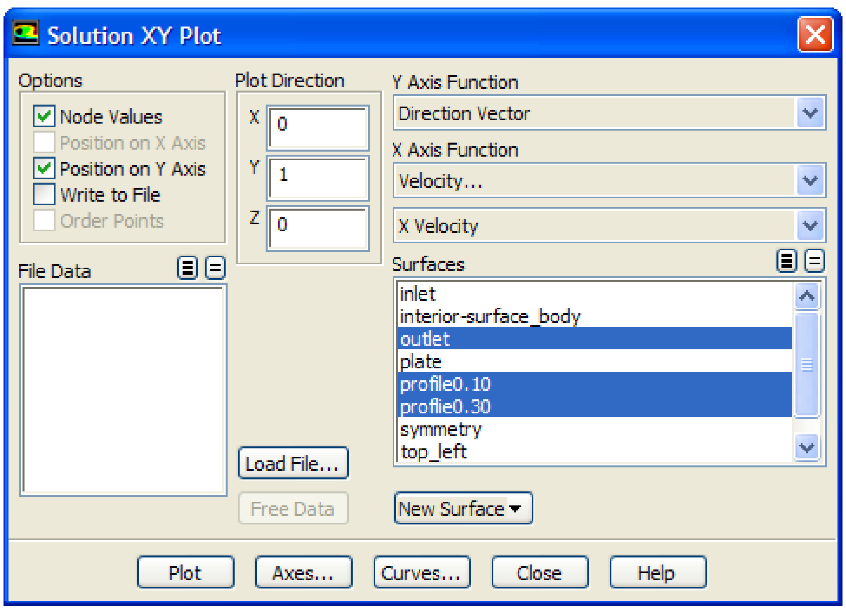
1. Our mesh is not tight enough near the wall to accurately resolve the boundary layer, especially near the front of the flat plate where the boundary layer is very thin. Fortunately, FLUENT has an “Adapt” feature that automatically adds grid points where needed for better resolution. There are several options for grid adaptation – we shall adapt by velocity gradient.
2. In the main *FLUENT* menu, Adapt-Gradient. In the new *Gradient Adaption* window, select Gradients of Velocity.
3. Select the Compute option. Minimum and maximum velocity gradients will appear in the window.
4. As a good rule of thumb, set the *Refine Threshold* to about 1/10 of the maximum gradient. Enter this values in the appropriate text box.
5. Mark. The main *FLUENT* window will display how many cells have been selected for refining and coarsening. The coarsening cells can be ignored since FLUENT is unable to coarsen the original grid – it can only refine the original grid.



6. *Optional*: If you want to see where the grid will be adapted, click Manage-Display. Areas destined for grid refinement will be highlighted. (You may need to zoom out to see this better.)
7. Back in the *Gradient Adaption* window, Adapt-Yes. The main *FLUENT* window will display some information about the grid adaptation.
8. The *Gradient Adaption* window can be closed at this point. Or, if you prefer – move it somewhere on the screen where it can be accessed again, since we will need it again later.
9. *Optional*: To see what the refined grid looks like, Display-Mesh from the main *FLUENT* menu, highlight interior-surface_body, and Display. (You may need to zoom in close to the wall to see more clearly.) You should see some new cells near the wall (especially near the leading edge of the flat plate) where velocity gradients are highest.
10. Solve-Run Calculation from the main *FLUENT* menu to re-open the *Run Calculation* sub-window. Change Number of Iterations to about 500, and Calculate-OK.
11. After the iterations, check to see how the solution is progressing. In the main *FLUENT* menu, Display-Graphics and Animations-Vectors-Set Up-Display. The graphical display window will show the velocity vectors. They should be closer together in regions where the mesh was refined.
12. Under *Surfaces*, select *only* inlet, profile0.10, profile0.30, and outlet. Display.
13. Zoom **in** (MMB lower left to upper right) or **out** (MMB upper right to lower left) and move (MMB where you want to center the view) as necessary to see all four velocity profiles.
14. Change the velocity vector scale to about 10 to see the profiles more clearly. Display. The growth of the boundary layer should be apparent. Close the *Vectors* window.

Examine the velocity profiles in detail

1. At this point, the velocity profile at three desired downstream locations (x = 0.10, 0.30, and 0.50 m) will be plotted and examined in detail.
2. In the main *FLUENT* window, on the left side under *Results*, Plots-XY Plot-Set Up. A window called *Solution XY Plot* will open. In this window, select (highlight) profile0.10 and profile0.30 from the list of surfaces. Also select outlet, which is at x = 0.50 m.
3. In the upper left corner of the window, turn ***off*** (uncheck) Position on X Axis, and turn ***on*** (check) Position on Y Axis. This will make the vertical axis the y position on the plot, as desired.
4. In the upper middle part of that window, set Plot Direction to X = 0 and Y = 1. This will make the y-coordinate position appear on the vertical axis, as desired for a standard velocity profile plot.

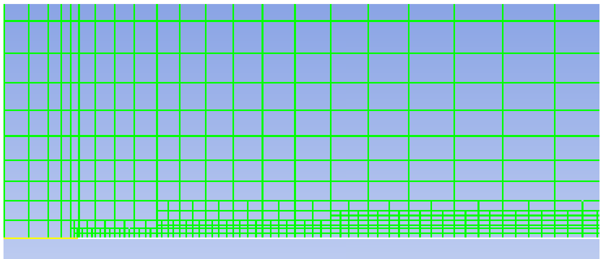


5. The upper right part of the window selects the variable to be plotted. The *Y Axis Function* will be set automatically to Direction Vector, and should be left alone. For the *X Axis Function*, select Velocity and (just below that) X-Velocity. Plot.
6. The boundary layer profiles should be there, but may be hard to see since the vertical axis extends all the way to the upper boundary of the computational domain. The axes limits can be changed as follows: Axes. Choose X if necessary (X should already be the default). ***Unselect*** Auto Range, and select Major Rules. Set *Range* from “0” to “0.11” m/s. Apply. (Nothing will happen to the plot yet, so don't panic.)

7. Similarly, choose the Y (vertical) axis. Unselect Auto Range, and select Major Rules for this axis. Set the range from "0" to "0.04" m. (Again, nothing will happen to the plot yet.)
8. To make the scale more readable, change the Number Format to *Type* = float and *Precision* = 3. Apply. Close.
9. Back in the *Solution XY Plot* window, Plot. Adjust the axes and/or number format as desired to obtain a nice plot of the boundary layer profiles. If done correctly, all three profiles should be visible on the plot, and the growth of the boundary layer with downstream distance should be apparent.
10. At this point, you should leave the *Solution XY Plot* window open, but move it to a convenient location since you will need it again. If you prefer, you can Close the *Solution XY Plot* window.
    - If you choose to close a window, you can always re-open it later when needed.

Iterate towards a final solution

1. At this point, the boundary layer should be reasonable, but more grid refinement and iteration may be required. As a general rule of thumb, if there are less than about 10 data points within the thickness of the boundary layer at the outlet profile location, the grid should be refined again.
2. Following the procedure outlined previously in the section called "**Refine the mesh and iterate some more**", refine the grid and re-iterate as necessary to obtain a final solution. Each time you adapt the grid, you must re-calculate the gradients (Compute), re-adjust the refine threshold (again, it should be set to about 1/10 of the maximum gradient), Mark, Adapt-Yes.
3. Iterate at least 300 to 500 iterations after each grid adaption. The residuals will rise dramatically after an adaption, but will decay as the solution adjusts itself to the newly refined grid.
4. *Caution*: Don't adapt too much, or the computations will take too much CPU time. Note that every time you refine the grid, the computer must calculate the flow field at more grid points, requiring longer for each successive iteration.
5. After refinement and iteration, look at the velocity profiles again, following the procedure discussed previously in the section called "**Examine the velocity profiles in detail**".
6. Adapt several times until there are at least 8 to 10 data points within the boundary layer at the outlet. A close-up view of a reasonable final mesh is shown to the right; notice the mesh refinement near the leading edge of the plate.



7. When finished adapting, run several hundred iterations until the residuals level off, or until the convergence criteria are reached.

Save your velocity profiles and your calculations

1. In the main *FLUENT* menu, File-Export-Case & Data. In the *Select File* window that pops up, name the file with an extension ".cas.gz". Make sure the *Write Binary Files* option is checked. OK to write the file. You might have to OK again to overwrite these files if they already exist.
    - The ".gz" at the end of the file name causes FLUENT to save the file in a condensed or "zipped" format, which saves disk space.
2. From the *Solution X-Y Plot* window, Plot.
3. Before saving the plot, your name and a short description should be added to the title. On the graphics window where the plot is visible, LMB just below the existing plot title ("X Velocity") in the bottom left of the plot. A cursor should appear. Type your name(s) on the plot title.
    - On some operating systems, you may have to RMB instead of LMB to get the text cursor.
4. In the main *FLUENT* menu, File-Save Picture, select TIFF, select Color if desired, and Save. Give a unique, descriptive name to the file (something with your name in it, like

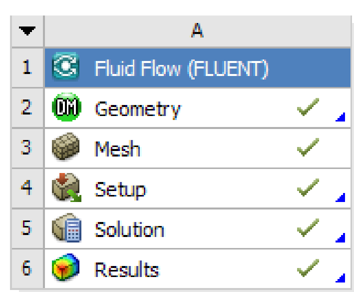
"laminar_BL_profiles_*Lastname_Firstname*.tif" is appropriate), OK, and Close. The tif file just generated can be inserted into a Microsoft Word document later.

5. The data points along the $x = 0.50$ m line will now be saved to an ASCII file for further analysis and comparison to predictions. From the *Solution XY Plot* window, select *only* the outlet line. Select Write to File, and Write. Name the file (suggestion: "laminar_BL_profile.txt"), and OK.
6. Close the *Solution XY Plot* window, and save your work: File-Save Project.

Calculate the drag force and drag coefficient on the body

1. In order to calculate drag coefficient correctly, the proper reference values for area, velocity, etc. need to be defined. In the main *FLUENT* menu, Report-Reference Values. A sub-window called *Reference Values* appears. The proper reference area to use is the planform area of the plate, defined as the area seen from above, in this case L times b, where L is the plate length (0.5 m) and b is the depth or span of the plate. In a 2-D calculation, FLUENT assumes unit depth (b = 1.0 m) for convenience so that drag force is per unit depth. *Depth* should already be set to 1.0 m by default.
2. Change *Area* to "0.5" m^2 (L times b), change *Velocity* to "0.1" m/s (the freestream velocity of the flow), and change *Length* to "0.5" m (L, the length of the flat plate).
3. Unfortunately, the reference values for density and viscosity are those of air, the default fluid, even though we defined our working fluid as liquid water. Change *Density* and *Viscosity* to their proper values. (These values for liquid water should have been written down previously, but can also be found from Define-Materials, select water-liquid, and Create/Edit.)
4. In the main *FLUENT* menu, Report-Result Reports-Forces-Set Up. In the window that pops up, under *Wall Zones*, select only the plate. You will need to enter the components of the force vector. These are the x and y components of a unit vector pointing in the direction of the desired force. Here, set the x-component to 1 and the y-component to 0, causing FLUENT to calculate the force (the drag force)acting on the wall in the x-direction, defined as parallel to the freestream direction.
   - The force is calculated only on the top of the plate since we have applied symmetry. An actual thin flat plate would have double the calculated drag (drag on both the top and bottom).
   - The calculated force is in units of N/m since this is a 2-D problem (force per unit span).
5. Print to perform the calculation. The results will be printed to the main *FLUENT* window.
   - You may need to widen that window and/or scroll to the right in order to see the results.
   - Notice that the force is broken into a pressure component and a viscous component. The pressure component is zero because the plate is parallel to the streamwise direction.
6. Write down the total drag force and drag coefficient on the flat plate.
7. To exit the *Force Reports* window, Close.

Save your calculations and exit FLUENT



Fluid Flow (FLUENT)

1. In the main *FLUENT* menu, File-Save Project. File-Export-Case & Data. OK. It is okay to overwrite the files, so OK again.
2. Exit FLUENT by File-Close FLUENT. Make sure the option to store the results is turned on. OK. This will return you to Workbench.
3. In Workbench, Update Project. After some calculations, check marks should appear on all components of the *Fluid Flow (FLUENT)* template in *Project Schematic*.
4. You are now finished with this tutorial. File-Exit-Yes (save the file).