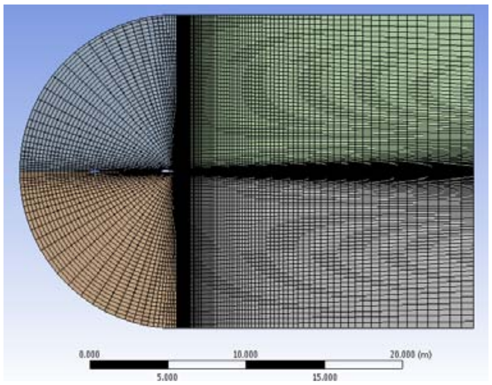
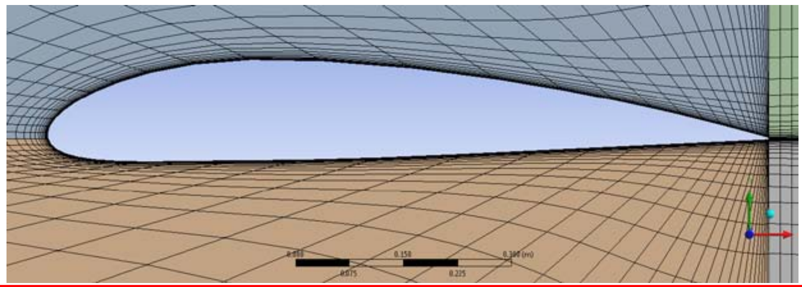
ANSYS Workbench Tutorial – Flow Over an Airfoil

Authors: Scott Richards, Keith Martin, and John M. Cimbala, Penn State University
Latest revision: 17 January 2011

Introduction

This tutorial provides instructions for creating a fluid volume and mesh around a NACA 7420 airfoil and for analyzing the flow in FLUENT. It also shows how to use multiple fluid bodies and edge sizing to create a "c-mesh". The entire meshed fluid field and a portion of the mesh near the airfoil are shown below.

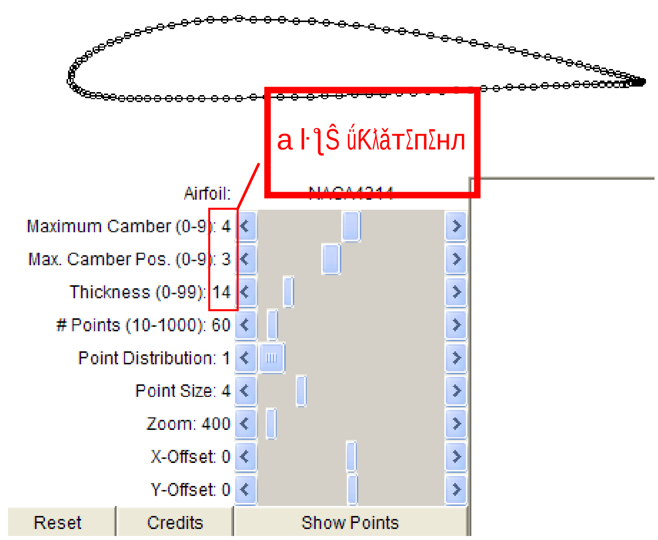




This tutorial was developed at Penn State University: it is excellent and is adapted for the airfoil modelling we are doing in the Courework Module

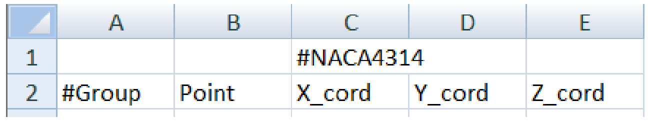
Generate data points that define the airfoil

1. Open an internet browser and go the the NACA 4 Digits Series Profile Generator at http://www.ppart.de/aerodynamics/profiles/NACA4.html .
   - On some computers, Java may be disabled and the website won't display properly. Check your browser's settings if you are having trouble viewing the website.
   - This site enables users to generate any standard NACA 4-digit 2D airfoil.



2. Adjust the top three sliders to create a non-symmetric 7420 airfoil.
3. Change *# Points* to "60" and *Point Size* to "4" so that the points are clearly visible.
   - If too many points are used to define the airfoil, DesignModeler won't be able to create the profile of the airfoil because the distance between adjacent points is too small.
4. Show Points, then highlight everything that appears in the window to the right of the sliders and copy it to the clipboard.
   - In Windows <Ctrl> + <A> can be used to highlight the data points, then <Ctrl> + <C> can be used to copy the selection to the clipboard.
5. Open Microsoft Excel.
   - A simple text editing program (such as *vi* in *linux*) can also be used to format the airfoil data. Use the following steps as a guidline to create a compatible file using a text editor:
     - Paste in the data (<Ctrl> + <V> in Windows machines).
     - Create lines of data with five fields separated by spaces or tabs, as described below.
     - Eliminate lines of data that have identical coordinates.
     - Precede comments with "#".
     - Save the data as a simple text file.
6. LMB in cell C1 to select it, then paste the data from the clipboard (<Ctrl> + <V>).

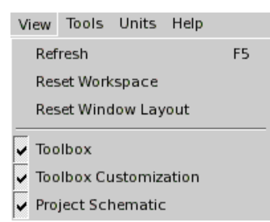
7. LMB on the “Data” tab. The data should still be highlighted. Text to Columns.
8. In *Convert Text to Columns*, Next. Select “Space” under *Delimiters*.
9. Next, then Finish to separate the data into two columns.
10. Insert a blank row between “NACA7420” and the airfoil data.
    - Rows can be inserted by RMB on a row number, then Insert. The new row will be inserted above the row on which you clicked.



11. In row 2, type column labels that match the ones shown to the right.
    - **LMB** on cell A2 and type “#Group” then <Tab>, type “Point” <Tab>, type “X_cord” <Tab>, type “Y_cord” <Tab> and type “Z_cord”.
12. Insert a “#” before “NACA7420” in cell C1.
    - The “#” is used to denote a comment in DesignModeler coordinate files.
    - The first two rows of this excel file help keep the data organized, but they are not required.
13. In the first line of data, change the X coordinate to “1” and the Y coordinate to “0”.
    - The first data point should be located at the trailing edge of the airfoil, but due to resolution and accuracy limits it is not exactly (1,0). Now scale everthing to make the chord 80mm.
14. Enter “1” in cell A3 and A4, and fill through the rest of column A.
    - The first column in the spreadsheet denotes the group number for each set of points.
15. Starting in cell B3, enter “1” and increase to “2” in cell B4. Fill the rest of the column in numerical order.
16. In column B of the last row of data, change the point number to “0”.
    - For closed curves, the last point number must be “0” in DesignModeler.
17. In cells E3, E4, etc. through the rest of the rows, enter “0” for the Z coordinate of each point.
18. Office Button-Save As. After selecting an appropriate folder, name the file “7420_Airfoil_Profile”. Change the *Save as type* to “Excel Workbook”. Save-OK-Yes.
    - Saving the file as an Excel Workbok isn’t necessary, but is done so that the file can be easily modified at a later time in the event that the import into DM is unsuccessful.
19. Repeat the procedure from the previous step, but change the *Save as type* to “Text (Tab delimited)”. Answer “Yes” (or “OK”) to any warnings that may appear; not to worry about these messages.
20. Close Excel.

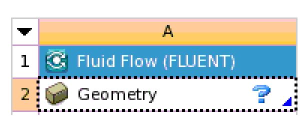
Create a FLUENT template in the *Project Schematic* window

1. This tutorial assumes that ANSYS Workbench is running but no projects are open.
2. Under *View* make sure that “Toolbox”, “Toolbox Customization” and “Project Schematic” all have check marks next to them.
    - Check marks can be inserted by placing the cursor over the menu item and LMB.



3. In *Toolbox Customization* under *Analysis Systems*, verify that “Fluid Flow (FLUENT)” has a check mark next to it.
    - If column “A” is not shown in *Toolbox Customization*, **1** located to the left of “Name” in *Toolbox Customization.*
    - To de-clutter your Workbench workspace, close or minimize *Toolbox Customization*; it is not needed after this step.
4. In *Toolbox*, not *Toolbox Customization*, Fluid Flow (FLUENT) and hold the LMB to drag it into the box that will appear in *Project Schematic*.
    - If there are no Analysis Systems visible in the *Toolbox*, try + (clicking “+”) next to *Analysis Systems* in *Toolbox.*

Use the data points to create the airfoil in DesignModeler (DM)



1. **Geometry** [bold underline means to double click] in the *Fluid Flow (Fluent)* template to open ANSYS *DesignModeler*.
2. At startup DM will prompt the user to select the desired length unit. LMB

on the circle next to "Meter", then OK.

✚ If this prompt does not appear, your version of DM was set to always use a certain unit of length. If this is the case, Tools-Options-Units, and set *Display Units Pop-up Window* to "Yes".

3. Concept-3D Curve. In *Details View*, **Not Selected**. Browse to where you saved the airfoil data, select the text file, and Open. Lastly, change *Merge Topology* from "No" to "Yes".
4. Generate [located in the menu above *Tree Outline*] to create the profile of the airfoil.
   ✚ If a "!" appeared next to "Curve1" in *Tree Outline*, R-Curve1-Show Errors or Warnings to display the reason why the 3D curve was not successful.
5. In the *Display Toolbar*, Look at Face/Plane/Sketch. This should also zoom in on the airfoil.
   If you want to find out the name of any toolbar button, hover over the button with the cursor to display its name.
6. Concept-Surfaces from Edges. LMB on the curve that was created in the previous step to select it. When it turns green to signify that it has been selected, Apply and Generate. The inside of the airfoil should now be shaded – it is a surface.

7. Create-Thin/Surface. Select the surface created in the previous step by LMB on it, then Apply in *Details View*.
8. Change *Direction* from "Inward" to "Outward".
9. LMB on the box to the left of *FD1, Thickness*. For *Parameter Name*, type "ExtrusionLength". OK.
10. In *Tree Outline*, R-"Thin1"-Rename. Name this body "Airfoil" and <Enter>. Generate.
11. In *Tree Outline*, LMB on "+" that is to the left of "2 Parts, 2 Bodies". R-"Solid"-Rename. Type "Airfoil" as the name for the solid and <Enter>.

Details View

Details of Thin1	
Thin/Surface	Thin1
Selection Type	Faces to Keep
Geometry	1 Face
Direction	Outward
D FD1, Thickness (>=0)	0.05 m

Create the fluid volume

1. Create-Primitives-Box.
2. In *Details View*, **From One Point and Diagonal** to change *Box Type* to "From Two Points".
3. Enter the coordinates for Point 1: X = "80"mm, Y = "-1" m, and Z = "0" m.
4. Enter the coordinates for Point 2: X = "2" m, Y = "1" m.
5. To set the value for *Point 2 Z Coordiante*, LMB in the box to the left of the label. For *Parameter Name*, type "ExtrusionLength", then OK. Generate.
   Using defined parameters to set dimensions makes changing dimensions easier later on.
6. Create-Primitives-Cylinder.
7. In *Details View*, set *Origin X Coordinate* = "80" mm. *Set FD10, Radius* = "1" m.
8. LMB in the box to the left of *FD8, Axis Z Coordinate*. For *Parameter Name*, type "ExtrusionLength", then OK. Generate.
9. Look at Face/Plane/Sketch.

Details View

Details of Box1	
Box	Box1
Base Plane	XYPlane
Operation	Add Material
Box Type	From Two Points
Point 1 Definition	Coordinates
FD3, Point 1 X Coordinate	1 m
FD4, Point 1 Y Coordinate	-10 m
FD5, Point 1 Z Coordinate	0 m
Point 2 Definition	Coordinates
FD6, Point 2 X Coordinate	20 m
FD7, Point 2 Y Coordinate	10 m
D FD8, Point 2 Z Coordinate	0.05 m
As Thin/Surface?	No

Details View

Details of Cylinder1	
Cylinder	Cylinder1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	1 m
FD4, Origin Y Coordinate	0 m
FD5, Origin Z Coordinate	0 m
Axis Definition	Components
FD6, Axis X Component	0 m
FD7, Axis Y Component	0 m
D FD8, Axis Z Component	0.05 m
FD10, Radius (>0)	10 m
As Thin/Surface?	No

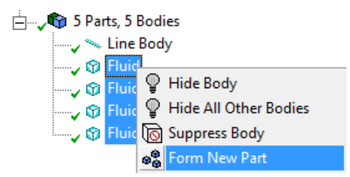
Remove airfoil from fluid volume

1. Create-Boolean.
2. Change *Operation* from "Unite" to "Subtract".
3. In *Tree Outline*, LMB on "+" that is to the left of "3 Parts, 3 Bodies" to display available bodies if necessary (this may already be selected).
4. For *Target Bodies*, Not Selected, choose "Solid" in *Tree Outline*, and Apply.

5. For *Tool Bodies*, Not Selected, choose "Airfoil" in *Tree Outline*, and Apply. Generate.
6. Tools-Freeze.
   - Freezing prevents the geometry from combining with geometry that is created after the freeze.
   - Rotate slightly by dragging the MMB; you should see a small thickness of the airfoil body.
7. In *Tree Outline*, LMB on "Solid" under "2 Parts, 2 Bodies". In *Details View*, change *Body* to "Fluid" and *Fluid/Solid* to Fluid.

Slice the fluid volume and exit DM

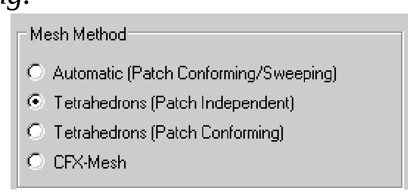
1. Create-Slice. Select "ZXPlane" from *Tree Outline*. Apply to assign the selected plane as the Base Plane. Generate.
   - Slicing the fluid volume into smaller volumes allows for greater control over mesh sizing.
2. New Plane. In *Details View*, set *Base Plane* as "YZPlane" (Select it in *Tree Outline* and Apply). Change *Transform 1 (RMB)* to "Offset Z". Set *FD1, Value 1* = "1"m. Generate.
3. Create-Slice. Select "Plane4" from *Tree Outline*. Apply to assign the selected plane as the Base Plane. Generate.
4. Simultaneously select the four volumes ("Fluid") under "5 Parts, 5 Bodies" in the *Tree Outline*. RMB-Form New Part.



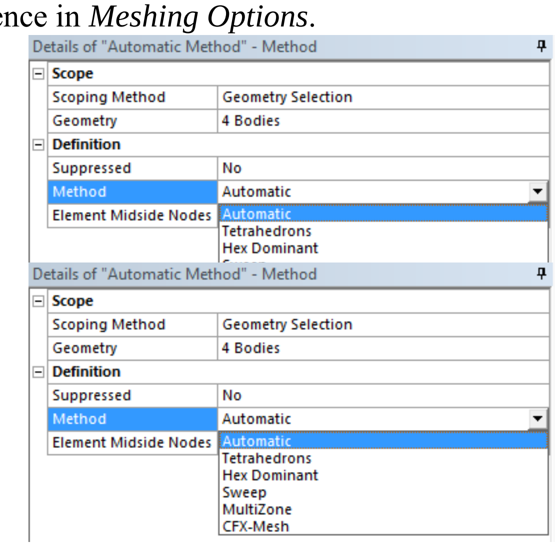
5. File-Save Project. After selecting an appropriate folder in which to save the project, enter "7420Airfoil" for the *File Name*, and Save.
6. Close DM.
   - In *Workbench*, "Geometry" should now have a check mark.

Set the meshing method

7. **Mesh** in the *Fluid Flow (Fluent)* template to open *ANSYS Meshing*.
8. From the *Meshing Options* window that opens, select "Tetrahedrons (Patch Independent)" to set the *Mesh Method*. OK.
   - 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*.

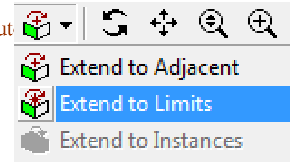


9. Units-Metric (m, kg, N, s, V, A).
10. In *Outline*, LMB on "Patch Independent".
11. Use the drop down menu to change *Method* from "Tetrahedrons" to "Automatic".

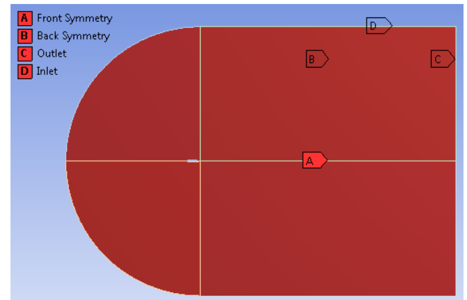


Label faces

1. In an empty area of *Graphics*, RMB-View-Front.
2. Group-select the four exposed faces. RMB-Create Named Selection. Enter "Front Symmetry" as the name for the group of faces. <Enter>.
   - Hold <Ctrl> while LMB to make multiple selections.
3. RMB-View-Back.
4. Group-select the four exposed faces. RMB-Create Named Selection. Enter "Back Symmetry" as the name for the group of faces. <Enter>.
5. RMB-View-Right.
6. Scroll the MMB to zoom in until two faces are visible.
7. Group-select the two exposed faces. RMB-Create Named Selection. Enter "Outlet" as the name for the group of faces. <Enter>.
8. RMB-View-Top.

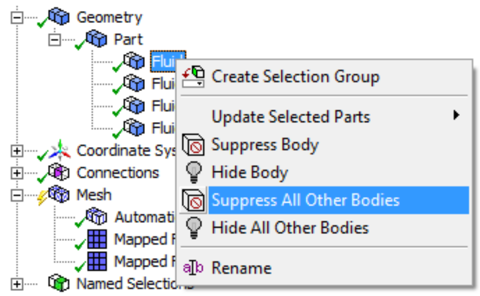


9. Zoom in until surfaces are visible. LMB on one of the exposed faces. LMB from the toolbar. Extend to Limits.
10. RMB-Create Named Selection. Enter “Inlet” as the name for the group of faces. <Enter>.
11. RMB-View-Front.
12. LMB Named Selections in the *Outline*. The labeled selections should look similar to the ones to the right.

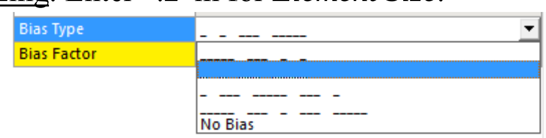


Insert edge sizing control

1. LMB “Mesh” in *Outline*.
2. LMB the rectangular face located in the top right. RMB-Insert-Mapped Face Meshing.
3. Repeat the previous step for the rectangular face located in the bottom-right.
4. In *Outline*, LMB on the “+” next to “Geometry”. LMB on the “+” next to “Part”. RMB on the first “Fluid” and Suppress All Other Bodies.
   - Edge sizing helps smooth the transition between the four separate fluid volumes and creates a refined mesh around the airfoil.

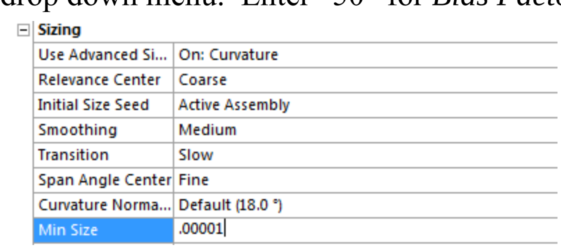


5. LMB “Mesh” from *Outline*.
6. Edge from the *Display Toolbar*.
7. LMB the left edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*.
8. LMB the right edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*. Choose the second *Bias Type* from the drop down menu. Enter “10” for *Bias Factor*.



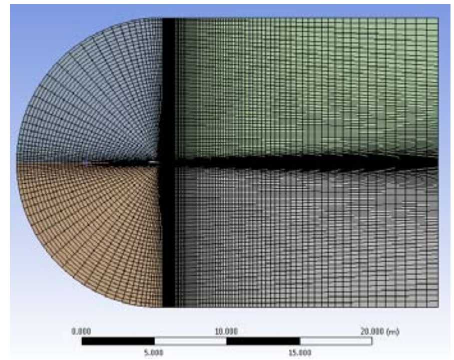
9. LMB the top edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*. Choose the first *Bias Type* from the drop down menu. Enter “50” for *Bias Factor*.
10. LMB the bottom edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*. Choose the first *Bias Type* from the drop down menu. Enter “50” for *Bias Factor*.
11. In *Outline*, RMB on the third “Fluid” under *Part* and Unsuppress All Bodies. RMB on the third “Fluid” and Suppress All Other Bodies.
12. LMB “Mesh” from *Outline*. Make sure Edge is the current selection filter.
13. LMB the left edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*.
14. LMB the right edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*. Choose the first *Bias Type* from the drop down menu. Enter “10” for *Bias Factor*.
15. LMB the top edge of the rectangle, then RMB-Insert-Sizing. Enter “.2”m for *Element Size*. Choose the second *Bias Type* from the drop down menu. Enter “50” for *Bias Factor*.

16. In *Outline*, RMB on the fourth “Fluid” under *Part* and Unsuppress All Bodies.
17. LMB “Mesh” from *Outline*. Make sure Edge is the current selection filter.
18. LMB the horizontal edge between the two pie-shaped faces. RMB-Insert-Sizing. Enter “.2”m for *Element Size*. Choose the second *Bias Type* from the drop down menu. Enter “50” for *Bias Factor*.



19. LMB “Mesh” from *Outline. In Details of “Mesh”*. LMB on the “+” next to “Sizing”. Change *Min Size* to “.00001”m.

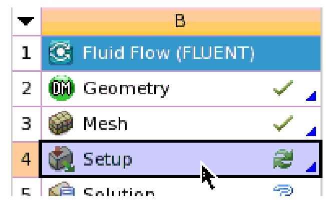
Create mesh



1. R-Mesh-Generate Mesh.
2. When completed, the mesh should look like the one to the right.
3. Zoom in and pan to view the mesh around the airfoil.
4. File-Save Project. Close *ANSYS Meshing* and return to ANSYS Workbench.

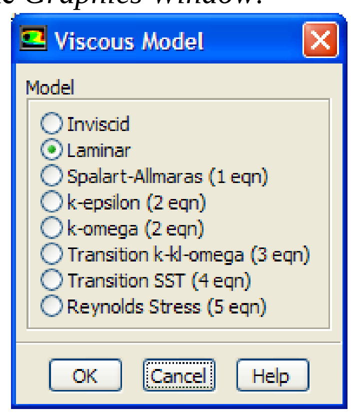
Update mesh and launch FLUENT

5. 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*.
6. **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 software versions may give an error. If you get an error, close ANSYS Workbench, reopen it, and repeat Step 2.
7. The next screen will be the main *FLUENT Window* with your mesh in the *Graphics Window*.

Set solver model



1. In the main *FLUENT* menu, Define-Models-Viscous - Laminar-Edit.

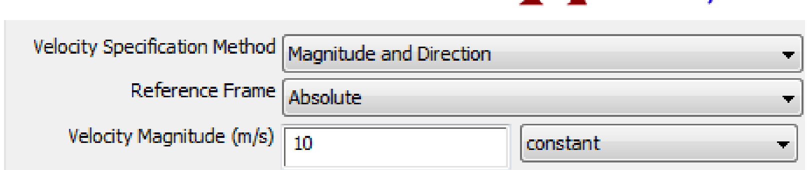
We (OXFORD) want Spalart-Almaras as the turbulence model so change the settings on the box on the right so that the correct check box is ticked.

Define the fluid as air

1. The default fluid is air, so you do not need to change anything in these boxes

Define the cell zones and boundary conditions

1. In the main *FLUENT* menu, Define-Cell Zone Conditions. There should be only one zone, "part-fluid".
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 wall (airfoil surface) is okay, so nothing needs to be done to it.
4. Likewise, the default boundary conditions for the symmetry planes and the outlet are okay, so nothing needs to be done to them.
5. Select inlet. Edit. Select "Magnitude and Direction" from



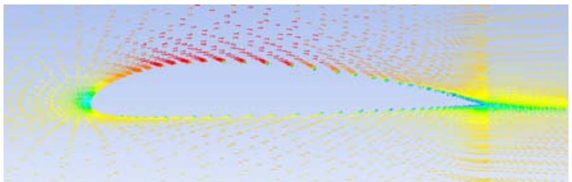
the drop down menu for *Velocity Specification Method*. Change *Velocity Magnitude* to the value calculated for a Reynolds number of 130,000, set your direction vectors and OK.

Set convergence criteria

1. In the main *FLUENT* menu, Solve-Monitors-Residuals…. Edit.
2. In the *Residual Monitors* window that opens, 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" graphs the residuals on the screen while the code is iterating.
   - 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.
3. Since there are four differential equations to be solved in a three-D incompressible laminar flow problem, there are four residuals to be monitored for convergence: continuity, *x*-velocity, *y*-velocity, and z-velocity. The default convergence criteria are 0.001 for all four 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 0.000001 (enter three additional zeroes).
4. To apply the changes, OK.
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. 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

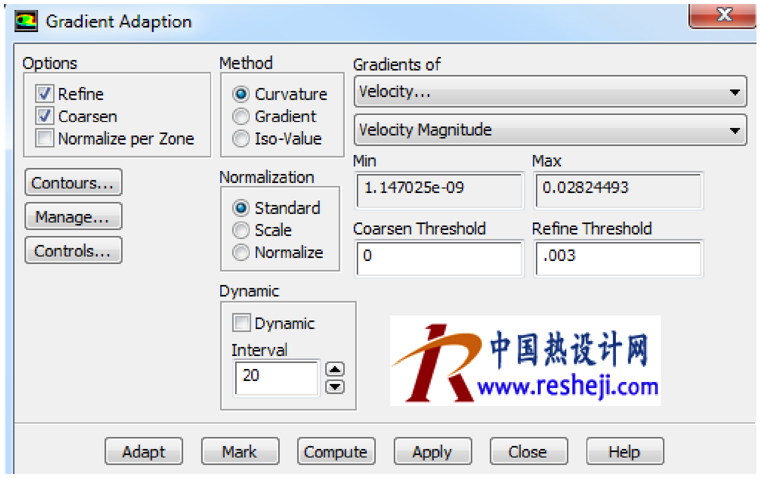
1. In the main *FLUENT* menu, Solve-Run Calculation to open up the *Run Calculation* sub-window. Change *Number of Iterations* to 200, and Calculate. The main screen will list the residuals after every iteration, while the graphical display window will plot the residuals as a function of iteration number.
   - 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.
2. When the solution is done or converged, OK.
3. 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].
4. In the *Vectors* window that opens, select "interior-part-fluid" under *Surfaces*. Display. Velocity vectors will be displayed.



5. Go get a better view of the vector field, rotate the vector field by LMB while dragging the cursor.
6. Zoom in on the airfoil to view the velocity field in more detail. It should look similar to the velocity field to the right. Close the *Vectors* window.
   - To zoom in, drag a box with the MMB from the lower left to upper right.
   - To zoom out, drag a box with the MMB from the upper right to the lower left.
7. The current mesh is sufficient for 200 iterations and initial calculations, but the mesh must be refined before iterating further.

Refine the mesh and iterate some more

1. Our mesh is not tight enough near the airfoil surface to properly resolve the boundary layer. Fortunately, FLUENT has a feature that automatically adds grid points where needed for better resolution. There are several options for grid adaptation – we will adapt by velocity gradient.



2. In the main *FLUENT* menu, Adapt-Gradient. In the new *Gradient Adaption* window, select Gradients of Velocity.
3. Compute. 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 value 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.
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.
9. Solve-Run Calculation from the main *FLUENT* menu to re-open the *Run Calculation* sub-window. Change Number of Iterations to 500, and Calculate-OK.
10. When the solution is done or converged, OK.
11. 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 velocity vectors. The vectors should be closer together in regions where the mesh was refined.
12. Close the *Vectors* window.
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 the velocity field.

Iterate towards a final solution

1. Following the procedure outlined previously in the section called "**Refine the mesh and iterate some more**", refine the grid and re-iterate as necessary (three times) to obtain a final solution. Each time you adapt the grid, you must re-calculate the gradients (Compute), re-adjust the refine threshold (set to about 1/10 of the maximum gradient), Mark, Adapt-Yes.
2. Calculate at least 400 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.
   - *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 successive iterations.
3. When finished adapting, run several hundred iterations until the residuals level off, or until the convergence criteria are reached.