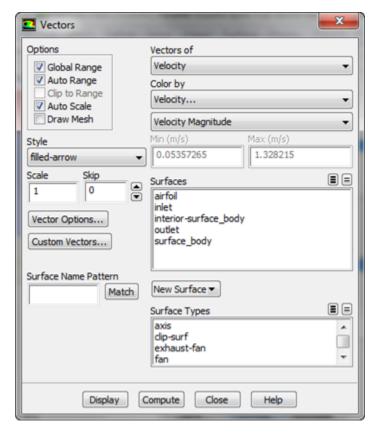
Flow over an Airfoil - Numerical Results

Author: Benjamin Mullen, 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

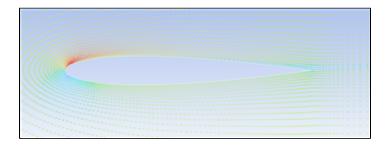
Numerical Results

Velocity

First, we will look at the velocity vectors of the solution to see if they make intuitive sense. To plot the velocity vectors, go to **Results > Graphics and Animations**. In the Graphics and Animations Window, select **Vectors** and click **Set Up...**. This will bring up the Vectors Menu.



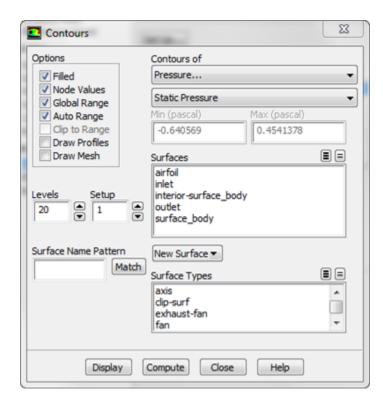
Make sure the settings of the menu match the figure above: namely *Vectors of > Velocity*, *Color by > Velocity*, and set the second box as *Velocity Magnitude*. To see the velocity vectors, press *Display*.



Pressure Contours

To view the pressure contours over the entire mesh, go to Results > Graphics and Animations again, and in the Graphics and Animations Window, select Contours.

Click Set Up... to bring up the Contours Menu. Check the box next to Filled. Under Contours Of, ensure that the two boxes that are selected are Pressure. .. and Static Pressure.



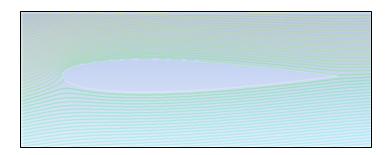
Once these parameters are set, press Display to see the pressure contours.

Streamlines

To view the streamlines, keep the *Contours* window open, and change the *Contours Of* box to *Velocity*, and the box below to *Stream Function*. Change *Levels* to 100. Also, uncheck the box marked *Auto Range*, and set *Min(kg/s)* to 13.11, and *Max(kg/s)* to 14.16

Contou	rs			23
Options		Contours of		
Filed		Velocity		-
Vode V		Stream Function		•
Auto R	-	Min (kg/s)	Max (kg/s)	
Clip to	Range	13.11	14.16	
Draw P	Profiles Mesh	Surfaces		
Levels Setup 100 • 1 •		airfoil inlet interior-surface_body outlet surface_body	,	
Surface Nar	me Pattern Match	New Surface 🔻		
		Surface Types		
		axis dip-surf exhaust-fan fan		-
	Display	Compute Close	Help	

To view the streamlines, press Display



Pressure Coefficient

Next, we will plot the pressure coefficient along the surface of the airfoil. Click on *Results > Plots* to open up the *Plots* Window. Under **Plots**, select *XY Plot*, and click *Set Up...*. In the window that pops up, change the settings *Y-Axis Function > Pressure*, and change the second box to *Pressure Coefficient*. Ensure *X-Axis Function > Direction Vector*. Under **Surfaces**, select *airfoil*. See the figure below for help.

Solution XY Plot	-	
Options V Node Values V Position on X Axis Position on Y Axis Write to File Order Points File Data		Y Axis Function Pressure Coefficient X Axis Function Direction Vector Surfaces Surfaces Surface interior-surface_body outlet surface_body
Plot	Free Data	New Surface Curves Close Help

When all the settings are correct, press *Plot* to plot the data to the command window. To save the data to a text file, check the box next to *Write to File*. You'll notice that the *Plot* button has been replaced by a button marked *Write...*, click it. Change the file type to *All Files* and save the file name as Pressu re_Coefficient.txt

Coefficients of Lift and Drag

To find the Coefficients of Lift and Drag, click **Results > Reports** to bring up the **Reports** Window. In the **Reports** Window, select **Forces** and click **Set Up...**. This will bring up the **Force Reports** menu

E Force Reports	metologi	23			
Options	Direction Vector X I Y O Z	Wall Zones 🗐 🚍			
Wall Name Pattern Match Save Output Parameter					
Print Write Close Help					

We need to set the parameters so drag across the airfoil (keep in mind, which is at an angle) will be displayed. In the *Force Reports* window change the *Direction Vector* such that X > .9945 and Y > .1045. Click *Print* to print the drag coefficient to the command window. To print the lift coefficient, in the *Force Reports* window change the *Direction Vector* such that X > .9945. Again, press *Print*.

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules