

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](#)

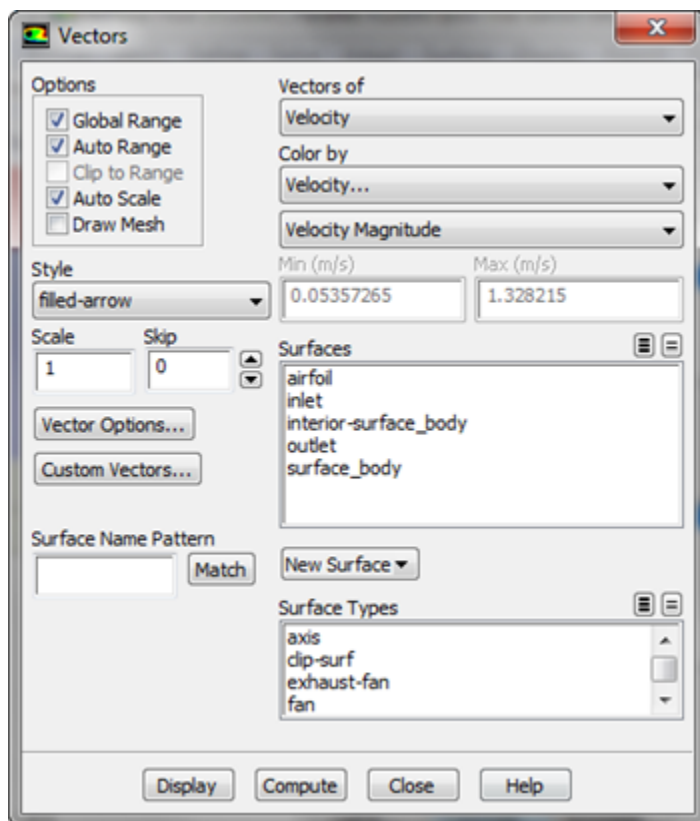
[Exercises](#)

[Comments](#)

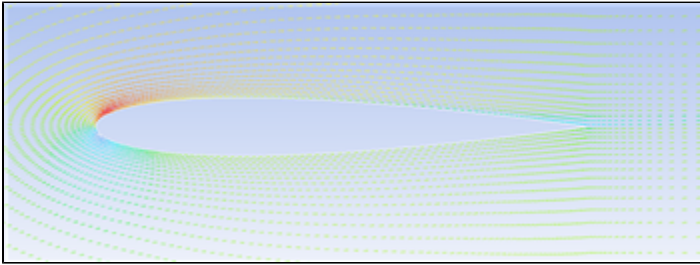
Numerical Results

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*.

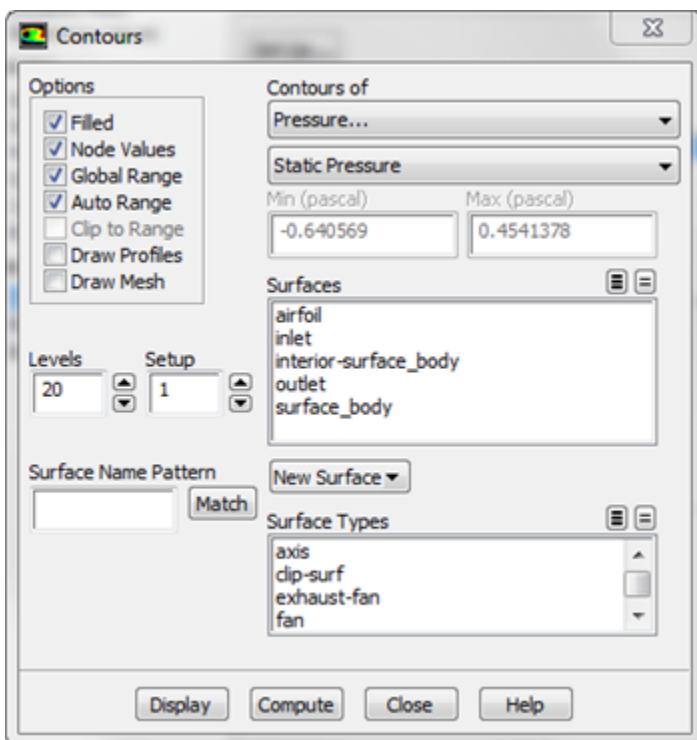


Make sure the settings of the menu match the figure above: namely **Vectors of > Velocity**, **Color by > Velocity**, and set the second box as **Velocity Magnitude**. To see the velocity vectors, press **Display**.



Pressure Contours

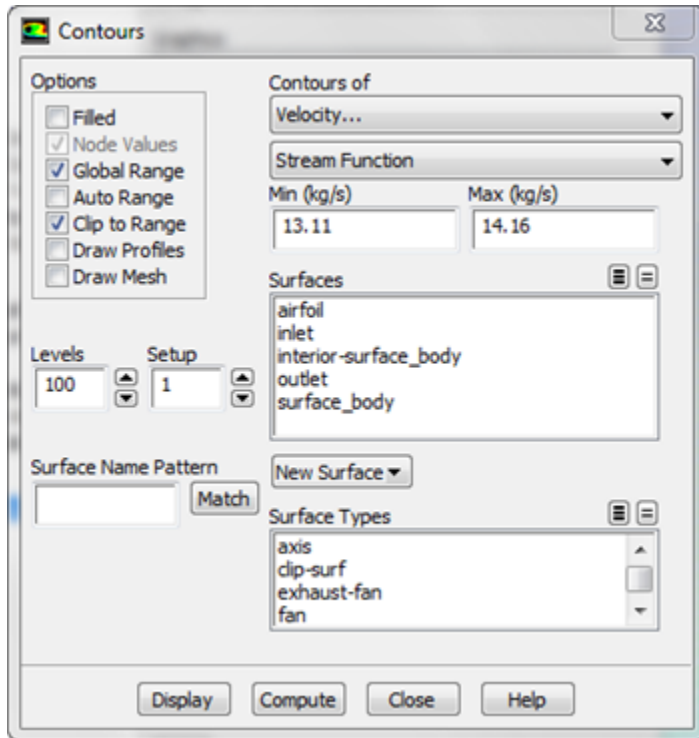
To view the pressure contours over the entire mesh, go to **Results > Graphics and Animations** again, and in the *Graphics and Animations* Window, select **Contours**. Click **Set Up...** to bring up the *Contours* Menu. Check the box next to **Filled**. Under *Contours Of*, ensure that the two boxes that are selected are **Pressure...** and **Static Pressure**.



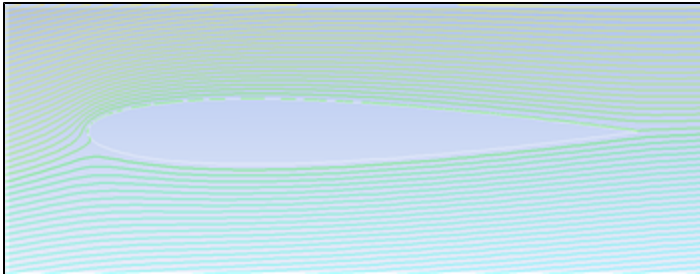
Once these parameters are set, press **Display** to see the pressure contours.

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

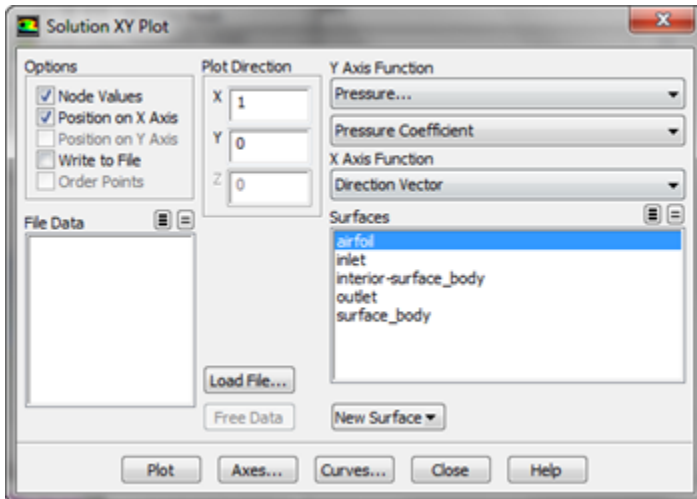


To view the streamlines, press **Display**



Pressure Coefficient

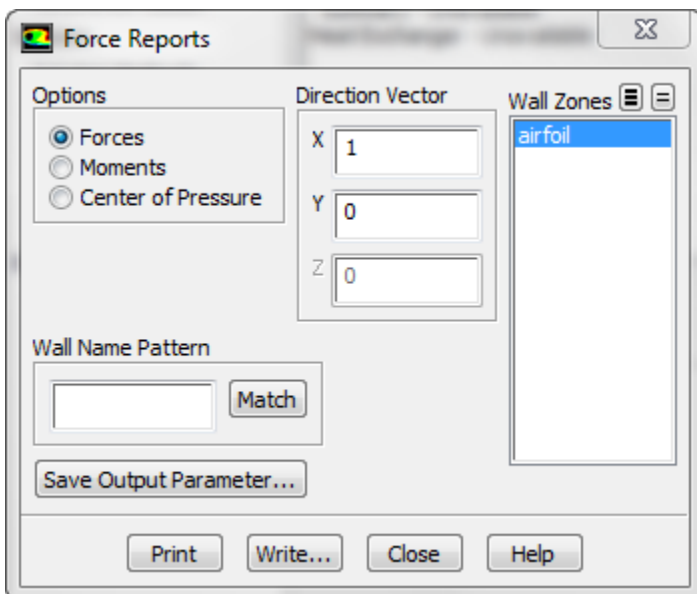
Next, we will plot the pressure coefficient along the surface of the airfoil. Click on **Results > Plots** to open up the *Plots* Window. Under **Plots**, select **XY Plot**, and click **Set Up....** In the window that pops up, change the settings **Y-Axis Function > Pressure**, and change the second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



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 `Pressure_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



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 $X > -.1045$ and $Y > .9945$. Again, press **Print**.

Go to Step 7: Verification & Validation

[Go to all FLUENT Learning Modules](#)