

# Flow Around an Airfoil

By Michael A. Traetow

## Purpose

The purpose of this simulation is to calculate and examine, using computational fluid dynamics, the surface pressure distribution and the lift force acting on the airfoil and compare to experimental fluid dynamics data.

## Simulation Design

The simulation will be performed on the commercial CFD code Fluent. The simulation will be modeled after the laboratory set-up of experiment number three and therefore will use the same clark-y airfoil geometry as well as the same flow parameters, such as fluid density, viscosity, angle of attack, and free stream velocity. By modeling the simulation after the experimental set-up, comparisons between the two test methods can be made.

## Step 1: Preliminary Inputs

The inputs, which will be taken from experimental lab #3, that are required for this simulation are the following:

- Air dynamic viscosity
- Air density
- Free stream air velocity
- Angle of attack

For the case of the airfoil, the geometry is quite complex and it is rather difficult to create a grid that adequately models the airfoil as well as the surrounding fluid flow regions. For this reason, the grid will be provided to you. In order to retrieve the grid that will be used, you will use the ftp commands as described below. In addition to the grid, you will also have to retrieve other files that will be used in the simulation. The purpose of these files will be explained at the time of their use. The following is a step by step procedure on how to ftp the desired files.

- 1) At the command prompt type *ftp iihr.uiowa.edu*.
- 2) Name: *anonymous*
- 3) Password: *<your user id>*
- 4) You will now have to change the directory to that of which the files are in. *cd incoming*
- 5) *cd iihr*
- 6) *cd 57\_020*
- 7) *get clarky.grd*. This is the grid file that will be used for the airfoil simulation.
- 8) *get clarky.input* This file allows you to run the solver in the background. This will be explained in more detail later.

- 9) *get cp.xls* This spreadsheet, already created for you, will calculate  $C_p$  and  $C_L$  upon the insertion of your data.
- 10) *get cp-ref.xls*. This is all of the reference data that will be used in this lab.
- 11) *get cfd2.README* This file explains the equations that were used to generate “clarky.xls” in addition to various other equations that will be used throughout the simulation.

You are now ready to enter Fluent, this is done by typing fluent at the command prompt of the computer you are working at (WTA accounts only). If you are working from and ICAEN machine, type the following:

```
l-ecn000% /usr/local/apps/fluent/bin/fluent
```

As you may remember from CFD lab #1, the first step was to allocate memory. This same step will be done in this simulation, however since the grid is being read in, the allocation step will have to be performed after reading the grid.

## Step 2: Grid Generation

As stated earlier, a pre-made grid will be used for this simulation, which will essentially eliminate the actual steps of generating a grid. However, in place of generating the grid you will use this step to read the existing grid into Fluent. This is done as follows

- 1) **FILE-READ-GRID**. This command will display the files that you will have to choose from.
- 2) Select **Clarky.grd** from the appropriate menu.
- 3) A window will appear that reads “Current units for length are in M. Apply unit conversion?” To this you will answer **NO**.
- 4) To make sure that the grid was transferred and read into FLUENT correctly, you will want to view the grid using **DISPLAY GRID LIVE DISPLAY**.
- 5) It is required that you include this grid in the memo so you will want to save it to a file. This is done by following the “Saving a GRAPH/PLOT” steps in the attached section of this handout.

At this time the memory will be allocated for the simulation. This is done as follows:

- 1) Go to the **DEFINE** menu located in the command bar at the top of the FLUENT window. Choose **ALLOCATE**.
- 2) From this menu, select **k-e/RNG Turbulence** model option in the “Create Space for” section. Select **OK**.

Since the domain and the wall boundary conditions have already been specified and input into FLUENT along with the grid, you will not have to define the cells as in the pipe flow. You will however need to specify the turbulence model.

- 1) **DEFINE MODELS**.
- 2) Click on the arrow in the “Turbulence Model” section and from this menu choose the **K-Epsilon** model.
- 3) **APPLY CLOSE**.

### Step 3: Flow Parameters

- 1) **DEFINE-BC s**. It is here where you are able to set the free stream velocity of the air moving through the wind tunnel.
- 2) Select **INLET 1** from the “Active Zones” section followed by the **VELOCITY INLET** in the “inlet type” window. **SET**.
- 3) A window titled “Velocity Inlet Boundary Conditions”. In this window you are able to set the free stream velocity in the X and Y directions. In order to simulate the conditions of experiment #3, determine what the free stream velocity of the air, and also the angle of attack that the experiment was performed at.
- 4) For the U-velocity, enter in the value of the free stream such that  $U\text{-velocity} = U_{fs} * \cos(\text{angle of attack})$ . This is simply “rotating” the free stream velocity by the angle of attack rather than rotating the foil itself. As you will see, the results are identical.
- 5) In the same manner, enter the V-velocity =  $U_{fs} * \sin(\text{angle of attack})$ .
- 6) In the “Turbulence Parameters” section set the **Turb. Intensity** to **0.5** and the **Charc. Length** to **0.001**. The reason for setting the turbulence intensity at this value is that the flow is uniform (very little turbulence, therefore small turbulence intensity). Likewise for the characteristic length (size of the eddies), it is very small since it is uniform flow.
- 7) **APPLY CLOSE**.
- 8) From the *setup-1* command prompt go to the *Physical Constants* menu.
- 9) To input the density of the fluid, choose *Density* from this menu. You will be asked if you would like to employ the ideal gas law, you do not so answer no. Enter in the value that was used in Experiment #3.
- 10) Type *Viscosity* to set the desired dynamic viscosity. Please note the units of the viscosity requested.
- 11) Return to the main menu by typing *Quit* at the prompt until you reach the “main menu” command prompt.
- 12) *Expert Solution Parameter Convergence Criteria*.
- 13) Set the Min. Residual Sum to a value of  $1 e^{-5}$ .
- 14) Exit this window by typing *done* at the bottom line. Return to the main menu.
- 15) Finally you will have to make an initial guess for the velocity. *Patch*.
- 16) Continue to press return until a menu appears that allows you to select the *U-Velocity*.
- 17) Enter an initial guess for the velocity in the U-direction. This will be the same as was input into Fluent for incoming flow.
- 18) *V-Velocity*. Enter in the initial guess for the V-Velocity which will be the same as you input for the incoming flow.
- 19) Return to the main menu.

### Step 4: Solve

- 1) **SOVL MONITERS RESIDUALS**
- 2) From the “Residuals” window, select Plot from the options section.

- 3) **APPLY CLOSE**
- 4) **SOLVE ITERATE**
- 5) In the “number of iterations” window enter the desired number of iterations (100). Because of the complex grid, the iterations will take much longer than that of the pipe flow. It is recommended that you run the program in the background once you have set Fluent up to solve the problem. This can be done by following the steps in the next section on how to run Fluent in the background.
- 6) In order to use the background feature you must first initialize the solver. This is why you set the number of iterations to only 100. Click on the **Iterate** button to begin.
- 7) After the initial 100 iterations have been performed, go to **FILE WRITE CASE & DATA**. Save the file as **CLARKY**. The must be in capital letters.

### Running the Flow Solver in the Background

- 1) In an xterm window, check to make sure that you have the following three files:
  - CLARKY.CAS
  - CLARKY.DAT
  - clarky.input
- 2) At the WTA machines type, at the command prompt type  

```
% nohup fluent g <clarky.input>& clarky.out& return
```

 At an ICAEN machine type  

```
% nohup /usr/local/apps/fluent/bin/fluent g <clarky.input>& clarky.out& return
```

 These commands simply start the solver in the background.
- 3) In order to check to see if the job is running type  

```
% more clarky.out
```

 If the solver is running, you will be able to see the added iterations in this file.
- 4) The results of the background running will be saved to CLARKY.DATANEW
- 5) In order to view the results, enter Fluent followed by **FILE READ CASE CLARKY.CAS**
- 6) This command is followed by **FILE READ DATA CLARKY.DATANEW**.
- 7) From here FLUENT can be manipulated as normal.

### Step 5: Post Processing

- 1) You may view the contours and profiles by using the **DISPLAY** menu in FLUENT.
- 2) For this lab, a plot of the pressure on the top and bottom surface will be needed. This is done by going to **PLOT-XY**.
- 3) The default in FLUENT is set to static pressure which is what you want.
- 4) Change the sweep line direction from “I” to ‘J’ by clicking on the “J” circle.

- 5) You will want to choose the level at which to evaluate the foil in the J-direction. For the ClarkY, the upper surface will be row 59 in the grid. In the respective section put J=59.
- 6) The pressure will be evaluated only at the points that are on the wing. In order to select this range enter in the **I-min cell 61 (leading edge)**, and for the **I max = 141 (trailing edge)**. Display this plot in the graphics window and view the results. If the results seem feasible, write this data to an Excel file.
- 7) Repeat the above two steps except change the surface to the bottom surface. This will be done by changing the 59 in step five to 62 (the lower surface). Repeat step six for this row of the grid.
- 8) The results should then be entered into the spreadsheet given in order to calculate  $C_p$  and  $C_L$ .

From this point on, it is easiest to do the remaining post processing in Excel. This concludes the use of FLUENT for Lab #3.

### Analysis

- 1) Construct a plot comparing the  $C_p$  values calculated using EFD and CFD (including uncertainty bands for the experimental data).
- 2) Compare the values of  $C_L$  (including uncertainty bands for the experimental data) that were calculated using EFD and CFD.
- 3) A plot of the ClarkY grid used in this CFD simulation

### Discussion

- 1) Key values ( $C_L$ ,  $C_p$  and percent error between CFD and EFD for each).
- 2) Discuss how large the percent error is compared to the uncertainty of experiment #3.
- 3) Reasons for the differences between CFD and EFD.

### Report Format

The report for CFD Lab #2 should be short and concise. This report will use the format of a technical memo which highlights the purpose, results and discussion, and the conclusions and recommendations that can be drawn from this lab. This memo should be a **maximum** of two pages text and an additional page for figures and tables.

## **Additional Fluent Commands**

By Michael A. Traetow

### **Saving a FLUENT File**

- 1) **FILE WRITE CASE & DATA**
- 2) Enter the <file name>.cas and click on **OK**.

### **Saving a Graph/Plot in FLUENT**

- 1) **FILE HARDCOPY**
- 2) If you will be printing this graph make sure that the **postscript** option is selected.
- 3) Click on **Save**.
- 4) Enter the <file name>.ps.
- 5) This can be accessed and printed while in the directory in which it was saved.

### **Saving XY plot data in FLUENT**

This option is used when you will be making graphs outside of FLUENT (in Excel).

- 1) **PLOT XY PLOT**
- 2) In the “Options” section choose **write to file**.
- 3) Select the data that you would like to have saved in a data file from the “Y Axis Function” section.
- 4) Click on **WRITE**.
- 5) Enter the <file name>.xls.
- 6) **OK**.

FLUENT has many options available to the user. The best way to become familiar with FLUENT, like any program, is to experiment with it's capabilities. If you ever have any questions, please do not hesitate to ask.