# **Unsteady Flow Past a Cylinder - Numerical Results**

Authors: John Singleton and Rajesh Bhaskaran, 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**

## **Examine Monitors**

If we look at our monitor of lift coefficient, we see how the lift coefficient changes with the flow time, becoming periodic due to the vortex shedding from the cylinder. We can use this plot to calculate the Strouhal number of the flow, which is a ratio of the unsteadiness in the flow to inertial forces in the flow field. We can calculate the Strouhal Number by calculating the frequency of the vortex shedding from our plot.

Unable to find DVI conversion log file.



Since this flow has D=U=1, the Strouhal Number is simply the frequency of vortex shedding. See the plot of Cl vs. Flow Time below. The timestep of .2 was used for this simulation.

## **Open CFD-Post**

We'll create a separate CFD-Post module, as this is the easiest way to load the results for this project.

On the left of the main project window, expand Component Systems and double-click Results.

Ξ	Component Systems
0	AUTODYN
Ð	CFX
Ì	Engineering Data
÷	External Connection
÷	External Data
0	Finite Element Modeler
•••	FLUENT
$\bigcirc$	Geometry
6	Mechanical APDL
6	Mechanical Model
	Mesh
X	Microsoft OfficeExcel
$\odot$	Results

Your project schematic window should now appear as below.

Ŧ	А	
1	G Fluid Flow (FLUENT)	
2	🕅 Geometry	× .
3	🍘 Mesh	× .
4	🍓 Setup	× .
5	🕼 Solution	× .
6	🥪 Results	× .
	Steady Flow	



Double click on the Results module that was just created to open CFD-Post.

## Results

Now, we need to load the results of our FLUENT simulation.

After opening CFD-Post, click the Load Results button in the upper left corner of the screen.

O C2 : Results - CFD-Post						
File	Edit	Session	Insert	Tools	Help	
<b>R</b>	<b>8</b> 8 -	4 🚺 🔟	90	1	Location	•

Next, browse to the location where you chose to save the FLUENT data files. Select the .cas file that is in this folder, which should be named "FFF-1-0001. cas", or similar. In the bottom right of this window, select Load complete history as: and Single Case. Finally, click Open.

💿 Load Results File						
Look in: 🔒 🚽 😋 📀 🗿 📑 🖽 🗏	Case options					
My Computer FFF-1-0001.cas FFF-1	☑ Keep current cases loaded					
Documents Documents	Open in new view					
labuser E FFF-1-0003.cdat FFF-1	Edit case names					
Goodmode     FFF-1-0005.cdat      FFF-1     FFF-1-0006.cdat      FFF-1	Additional actions					
FFF-1-0007.cdat  FFF-1 FFF-1-0008.cdat  FFF-1	Clear user state before loading					
EFFF-1-0009.cdat E FFF-1	Maintain camera position					
FFF-1-0010.cdat E FFF-1 FFF-1-0011.cdat E FFF-1	Load particle track data					
E FFF-1-0012.cdat E FFF-1	CFX run history and multi-configuration options					
E FFF-1-0014.cdat E FFF-1	Coad only the last results					
FFF-1-0015.cdat E FFF-1	Output Load complete history as:					
	A single case					
File name: FFF-1-0001.cas Open	Separate cases					
Files of type: All Readable Files (re 🗸 Cancel						

Click **OK** in the popup window if one appears.

#### Load Timesteps

Click Tools > Time Step Selector to open the Time Step Selector.



Select the first time step, and click Apply. Leave the Time Step Selector window open, but continue to the next step.

#### **Create Vorticity Contour**

Now, let's insert a contour of vorticity, in order to animate it.

While leaving the Time Step Selector window open, click Insert > Contour. Name it "Vorticity Contour".

Under Details of Vorticity Contour, select symmetry 1 from Locations.

Next, ensure that Variable is set to Vorticity.

Change Range to User Specified. Set the Min to 0.01 s^-1 and Max to 2 s^-1.

Enter 25 for Number of Contours. You should now see the following:

## Details of Vorticity Contour

Geometry	Labels Render View
Domains	All Domains 💌 📖
Locations	symmetry 1 🔹 🛄
Variable	Vorticity 🔹 📖
Range	User Specified 👻
Min	0.01 [s^-1]
Max	2 [s^-1]
Boundary Da	ta 🔘 Hybrid 🔘 Conservative
Color Scale	Linear 🗸
Color Map	Default (Rainbow)
# of Contours	3 25
Clip to Ra	nge
Apply	Reset

## Click **Apply** to create the contour.

Next, let's set up the view we would like for the animation. You can see that we are currently viewing the 2D surface from a 3D, isometric perspective. To fix this, click the Z-axis in the axes triad in the lower corner.



Now let's zoom in to the are of interest. Select the zoom box tool from the upper toolbar.



Using the zoom box tool, click and drag a box that roughly encompasses the area shown below to zoom in on it.



Now we're ready to animate the vorticity contour over this zoomed-in area.

## **Create Animation**

Return to the Time Step Selector Window, which should still be open. Click the Animate Timesteps button.

💿 Tir	mestep	Selector			? ×	
FFF	1 000 1					
Curr	ent Time	estep: 1				
#	Step	Solver Step	Time [s]	Туре	~	
1	1	1	0.2	Full		
2	2	2	0.4	Full		
3	3	3	0.6	Full	×	
4	4	4	0.8	Full		
5	5	5	1	Full		
6	6	6	1.2	Full		
7	7	7	1.4	Full		
8	8	8	1.6	Full		
9	9	9	1.8	Full		
10	10	10	2	Full		
11	11	11	2.2	Full		
12	12	12	2.4	Full		
13	13	13	2.6	Full		
14	14	14	2.8	Full	-	
La.e.		45	-			
Apply Reset Close						

Select Keyframe Animation, and click the insert new keyframe button, to animate, in this case 400. Your Animation window should look like this:

2

Change the number of frames to equal the number of data files we saved

0	Ar	imation		-	<u> </u>	X
	0	Quick Anir	nation	Ke	yframe Anii	mation
	#	Keyframe		Frame #	Frames	
	1	Keyframe	lo1	1	400	-
						2
						3
						0
						×
#	≠ of	Frames	400			* *
	μ	K I F	• [		►F ►	K
			F: 1	K: 1		*
						- •
						Close

Keeping the Animation window open, click back to the Time Step Selector window. Select time step #400, and click **Apply**. The Vorticity Contour on the right half of your screen should now have changed. Click back to the Animation window, and insert another new keyframe. This time, leave the number of frames set to 10.

We're now ready to set up the saving options for the animation. Click the arrow in the bottom right of the window to expand the options.

💿 An	imation				? X
Ô	Quick Animatio	n	Ker	yframe An	imation
#	Keyframe		Frame #	Frames	
1	KeyframeNo1		1	400	
2	KeyframeNo2		402	10	1 🎦 🗌
					1
					$\sim$
					^ ⊥
# of	Frames 10				* *
H	K F •			F	K M
	► <b>F</b> :	402	K: 2		*
0	Doop		⊚ Bo	unce	
Rep	eat	1		[	
	Save Movie	to	p/FFF 10	001.mp4	2
For	mat MF	EG4			•
	Animate Carr	era		Option	ıs
					Close

Check the box labeled Save Movie, and use the folder icon to set the desired file location and type.

Next, maximize your CFD-Post window, and click the play button in the Animation window to create the animation! Your video should turn out similar to the one below.

## Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules