Workshop 6
Vortex Shedding

Introduction to ANSYS
CFX
Workshop Description:
Set up a transient simulation of vortex shedding behind a cylinder and compare the predicted Strouhal number with experimental data. The cylinder has a diameter of 2m and the type of shedding will be a Kármán vortex street.

Learning Aims:
This workshop introduces several new skills:
• Preparing a simulation for transient analysis
• Learning how to post-process a transient simulation, including performing an FFT

Results from a completed run are provided for the purposes of post-processing
Reynolds Number Effects

\[ Re = \frac{\rho V d}{\mu} \]

- **Re < 5**: Creeping flow (no separation)
- **5-15 < Re < 40**: A pair of stable vortices in the wake
- **40 < Re < 150**: Laminar (Kármán) vortex street
- **150 < Re < 3 \times 10^5**: Laminar boundary layer up to the separation point and turbulent wake
- **3 \times 10^5 < Re < 3.5 \times 10^6**: Boundary layer transition to turbulent
- **Re > 3.5 \times 10^6**: Turbulent vortex street - the separation is narrower than in the laminar case
Mesh Import

• Launch Workbench and drag and drop a CFX component system into the project page.

• Open CFX-Pre and import the ICEM CFD mesh called F10_S10_B15_Hex010.cfx5 using Units of m. It can be found in workshop_input_files\WS_06_Vortex Shedding.

• For some mesh formats it is important to know the units used to generate the mesh.

• Save the project into your workspace as Vortex.wbpj.
Define Simulation Type

The first step is to change the Analysis Type to Transient:

• Edit the Analysis Type object in the Outline tree
• Set the Analysis Type option to Transient
• Set the Total Time to 100 [s]
• Set Timesteps to 0.1 [s] and click OK
New Material

Laminar vortex street occurs at $40 < \text{Re} < 150$. The physical properties of our fluid will be defined so that $\text{Re} = 100$ for a velocity of $20 \text{ m/s}$ around a cylinder with a diameter of $2 \text{ m}$.

- Create the following expressions:
  - $\text{FlowVelocity} = 20 \ [\text{m/s}]$
  - $\text{FluidDensity} = 1 \ [\text{kg/m}^3]$
  - $\text{Re} = 100$
  - $\text{FluidViscosity}$ for a viscosity based on above expressions and a diameter of $2 \ [\text{m}]$. Remember:

$$Re = \frac{\rho V d}{\mu}$$

- Create a new Material called $\text{MyFluid}$ using the expressions $\text{FluidDensity}$ and $\text{FluidViscosity}$ to define its physical properties.
Default Domain

- Open the Default Domain and select MyFluid as the Material for Fluid 1
- Flow is laminar
- No thermal effects
Boundary Conditions – Walls & Symmetry

- **LeftWall, Free Slip Wall on LEFT**
- **Sym1, Symmetry on SYM1**
- **Cylinder, No Slip Wall on CYLINDER**
- **Sym2, Symmetry on SYM2**
- **RightWall, Free Slip Wall on RIGHT**

**Create the boundaries shown**

There is no shear stress at a free-slip wall. It does not create a velocity profile. Examples of its use are:

- To represent the free surface of a liquid in a single-phase model
- Artificially to limit the extent of a domain for an external flow, as here
Boundary Conditions – Inlet & Outlet

- Create the following flow boundaries:
  - An *Inlet* called Inlet located on *IN* with *Normal Speed* = *FlowVelocity*
  - An *Outlet* called Outlet located on *OUT* with *Average Static Pressure* = 0 [Pa]
To accelerate the generation of vortices, we start with an asymmetric velocity field downstream of the cylinder as shown below. The offset is 10 degrees. Regular cyclical variations will develop as the run proceeds and so the initial conditions do not need to be completely physical.

- Initial relative pressure = 0 [Pa]
- Z-component of velocity = 0 [m/s] since flow is two-dimensional
- Settings for the other two velocity components are described on the next slide.
Initial Conditions

• Create the following expressions to define the initial velocity:
  – InitialVx for X-component. Equal to the inlet velocity
  – InitialFlowAngle for the offset of 10 degrees
  – InitialVy for Y-component.
    • upstream of centre of it is 0 [m/s]
    • downstream it is FlowVelocity\cdot\tan(InitialFlowAngle)
    • centre of cylinder is at X = 0 [m]
    • use an \textit{if()}, seen previously, or a \textit{step()} function.
      – \textit{step}(x) = 1 if \(x > 0\)
      – \textit{step}(x) = 0 if \(x < 0\)
      – \(x\) is dimensionless
Under *Solver Control > Basic Settings* set the following parameters:

- Min. Coeff. Loops = 1
- Max. Coeff. Loops = 5
- Residual Type = MAX
- Residual Target = 1E-3

These parameters together with the *Timestep* are the key numerical inputs for a transient calculations.
Output Control

• Under Output Control > Trn Results perform the following steps:
  Insert new Transient Results
  Option = Selected Variables
  Output Variable List = Pressure, Velocity, Velocity u, Velocity v, Velocity w
  Timestep Interval = 5

• On the Monitors tab switch on Monitor Objects and Monitor Coefficient Loop Convergence. It will then be possible to monitor the convergence within each time step.
Output Control - Monitors

Vortex shedding will lead to variations in the lift and drag forces on the cylinder. We will monitor the changes in the values of the coefficients of drag and lift with time.

The coefficients take the form \( \frac{2F}{(A \rho v^2)} \)

- \( F \) is the force on the cylinder wall. For drag it is the stream-wise component (X) and for lift the transverse component (Y)
- \( \rho \) and \( v \) are the density and velocity of the fluid in the free stream; the fluid density is constant and the free-stream velocity can be defined using the inlet velocity
- \( A \) is the projected area of the cylinder. The diameter of the cylinder is 2 [m] and its length is 0.5 [m]

• Create expressions, \( C_{dCylinderExpression} \) and \( C_{lCylinderExpression} \), for the coefficients of drag and lift, respectively. The expressions are identical apart from the component of force. So you can duplicate the first and edit it to create the second

\[
\text{Details of } C_{dCylinderExpression} \\
\begin{array}{|c|}
\hline
\text{Definition} \\
\hline
\text{(force}_x\text{Cylinder}^2)/\text{(FluidDensity}^\text{FlowVelocity}^2\times0.5[m]^2\times2[m]) \\
\hline
\end{array}
\]

• Use the two expressions to define the monitors, \( CdCylinder \) and \( ClCylinder \)
Output Control – Monitor Points

• Create HighPpt & LowPpt to monitor Pressure on the upstream and downstream sides of the cylinder wall

• Create Monitor Points 1 – 7 to monitor Velocity in the wake at a series of points 2 [m] above the centre of the cylinder. The X coordinates are tabulated below

Note that the centre of the cylinder is at (0 [m], 0 [m], 0.25 [m])

You can speed up the creation of the 7 monitor points by duplicating points in the Outline Tree and/or using the Command Editor

<table>
<thead>
<tr>
<th>Monitor Point</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>X [m]</td>
<td>-2</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>6</td>
<td>8</td>
<td>28</td>
</tr>
</tbody>
</table>

Diagram showing the placement of monitor points.
Run Solver

We will examine the effect of changing the size of time step during this run.

- Save the project, open the CFX-Solver Manager and start the run.
- After about 50 time steps select Edit Run in Progress and save a back-up file when asked.
- When the editor window appears, expand the FLOW: object and then ANALYSIS TYPE: and TIME STEPS:. Double-click on Timesteps and change the value from 0.1 [s] to 0.01 [s].
- File > Save.
- Let the run complete ~500 time steps before stopping it (Interrupt Update in the Project Schematic).

NOTE: Edit Run in Progress has no effect on the CFX-Pre case file (.cfx) or the definition file (.def)
CFX-Solver Manager

As Monitor Coefficient Loop Convergence was active, we can see graphically how well the solution converged within a time step.

- Change residuals displayed from RMS to MAX - the target was a max of 1E-3.

To study the behaviour within a time step, we will look at short sections of the plot.

Right-click in the Momentum and Mass window and edit the Monitor Properties to display a Fixed range of 10 time steps just before the run was edited. Do the same for a period after the edit.
With the larger time step the solution did not converge after the 5th coefficient loop but when the size of the time step was reduced it converged on the 3rd loop.

• On Workspace > Workspace Properties > Global Plot Settings
  – switch off Plot Coefficient Loop Data
    • only values at the end of each time step are plotted
  – change Plot Data By to Simulation Time
• Create a new monitor (Workspace > New monitor)
  – plot one USER POINT, e.g. CdCylinder
  – Adjust the vertical range to show more clearly the effect of reducing the time step
CFX-Solver Manager

- View short sections of the plot to see how the reduction in time step size affects the shape of the curve
  - Easier if data plotted by time step (Monitor Properties > Range Settings)
  - Remember that 10 times as many time steps will be needed to monitor a given interval after the size of time step was reduced.
Post-Process Results

• A full set of results from a run covering a period of 20 s with a time step of 0.01 s is provided in the folder called Results. Drag and drop CFX_001.res into the Project Schematic and open CFD-Post.

• Plot contours of Velocity from 0 – 26 [m/s] on Sym1. Use 27 contour levels.
Keyframe Animation

- Using the Timestep Selector, load the first time step (#1) and open the Animation tool (Keep the Timestep Selector open.)

- Switch from Quick Animation to Keyframe Animation and create the first Keyframe.

- Load the last time step (#401) and create the second Keyframe. A minimum of two Keyframes are needed. They represent start and end states.

- Double-click on the first Keyframe to rewind. Change # of Frames to 399. This is the # of Frames between the first and second Keyframes. Here we have one frame for each transient file written.

- Switch on Save Movie and save in your working directory

- Toggle off Repeat forever and click Play. It will take 4 or 5 minutes to create the animation.
Keyframe vs. Quick Animation

We could have created a similar movie in a Quick Animation by simply animating Timesteps.

A Keyframe Animation offers greater control and more options. A set of Keyframes can represent a series of different states, e.g. camera position and object visibility. Everything that changes is interpolated over the frames between Keyframes.
Strouhal Number

• The contour plot and animation show the formation of vortices behind the cylinder
• The frequency of the vortices depends on the Reynolds number
• The Strouhal number is a dimensionless number that indicates the predominant frequency
• For flow past a cylinder it tends to be about 0.2 over a wide range of Reynolds number – for a Reynolds number of 100 it is between 0.16 and 0.17
• The Strouhal number is defined as a function of the frequency, diameter and velocity
• The frequency can be calculated through a Fast Fourier Transform of the monitoring points. We will use the chart tool in CFD-Post to perform an FFT on the velocity at Monitor Point 2

\[ St = \frac{f \cdot D}{U} \]
Export Plot Data

For the FFT we need to read monitor data into a chart in CFD-Post. One method is to export data from the CFX-Solver Manager

• Right-click on the Solution cell for the imported results and select Display Monitors to open the CFX-Solver Manager

• Switch off Plot Coefficient Loop Data and plot data by Simulation Time (Workspace Properties)

• Create a New Monitor to display the plot line for Monitor Point 2

• Right-click in the new monitor window and select Export Plot Data.... Save the data into your working directory.

The exported file contains a column for time step number as well as the two required for time and velocity. So it needs to be edited. An edited file is provided (Results/Monitor Point 2.csv)

• Close the CFX-Solver Manager
Chart - FFT

• Open CFD-Post and create a new Chart called myFFT

• General tab
  – Type = General XY - Transient
  – Fast Fourier Transform = on
  – Subtract mean = on (so that the amplitude is clearer)
  – Range input Data Min = 10 and Max = 20 (excludes early values when the vortices were not established)

• Data Series
  – Data Source > File = Monitor Point 2.csv

• X Axis tab
  – Min = 1 and Max = 5

• Y Axis tab
  – Variable = Velocity
  – Y Function = Magnitude

• Apply

• Reduce the range of the X Axis to 1.50 – 1.55 so that the frequency with the greatest Magnitude is clearer. It would also be possible to export the chart data to a csv file, which could be opened and sorted in a spreadsheet package to identify the frequency of greatest magnitude.

• Use this frequency, the diameter (2 m) and velocity (20 m s⁻¹) to calculate the Strouhal number
There is a beta feature that allows you to read monitor data directly into a chart in CFD-Post. Beta features must always be treated with caution as they have not been fully tested.

Select *Edit > Options* in CFD-Post for the option to *Enable Beta Features*.

Extra option labelled *Monitor Data (Beta)* then available for *Data Source* on *Data Series* tab of Chart.

With this option selected you can set the *Variable* for the *X Axis* to *Time* and select the required monitor point as the *Variable* to define the *Y Axis*. 
Optional Exercise 1 – Grid Refinement

The Strouhal number is about 0.1498 compared with an experimental value of 0.164. Runs on a series of finer grids (provided in the directory called GRIDS) can be used to study the discretisation error. As the grids are refined, the Strouhal number asymptotically approaches a grid-independent value.

There will probably not be sufficient time to run each case.

- Restart from the previous completed solution obtained on a coarser grid
  - In CFX-Pre increase the Total Time or switch to the Time per run
  - Set a shorter time period, e.g. 5 s; as the grid is finer, each time step takes longer
  - Make sure the Timestep size is 0.01s

- Import the monitor data directly into the chart using the beta feature.

- Change the range of input values for Time so that only good data from the current run are used.
  - To find a suitable range, create a monitor in the CFX-Solver Manager for Monitor Point 2, plotting data by Simulation Time.
Optional Exercise 2 – Transient Chart

We have seen two ways of using existing monitor point data to create a chart in CFD-Post. It is possible to create a chart without monitor data, provided that the required variable is available in the intermediate transient files.

• In CFD-Post open the res file provided with the workshop input files and create a point location at 2 [m], 2[m], 0.25 [m].

• Create a new chart
  – Type = XY - Transient or Sequence
  – On Data Series set Location = Point 1 for the Data Source
  – X Axis = Time and Y Axis = Velocity

• Add a second Data Series and bring in the data for Monitor Point 2.

• Chart a 1s time period.

Note that the first curve is less well resolved as the transient files were written at intervals of 5 time steps.
Optional Exercise 3 – Derived Variables

- In the CFX-Solver Manager, create and explore the options for Derived Variables
Summary

• You performed a transient simulation to study the laminar vortex shedding behind a cylinder.
  – You changed the size of time step during the run and noted its effect on the solution.

• The computed Strouhal number was compared with the experimental value and, optionally, the effect of grid refinement on the results was tested.

• A keyframe animation was created.

• You saw how to switch on Beta Features in ANSYS CFX and two different ways of importing Monitor Point data into CFD-Post.

• Two methods of charting a variable value against time were used.