FLUENT - Unsteady Flow Past a Cylinder

This page has been moved to https://courses.ansys.com/index.php/courses/unsteady-flow-past-a-cylinder/
Click in the link above if you are not automatically redirected in 10 seconds.

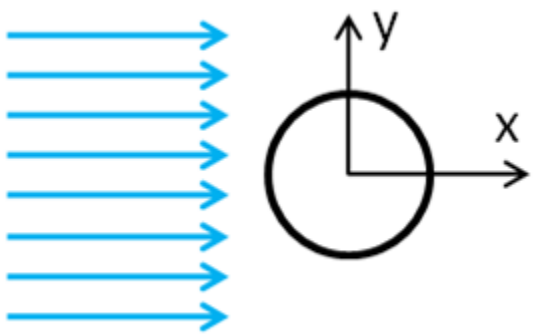
Authors: John Singleton and Rajesh Bhaskaran, Cornell University

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

Unsteady Flow Past a Cylinder

Created using ANSYS 13.0

Problem Specification



Consider the unsteady state case of a fluid flowing past a cylinder, as illustrated above. For this tutorial we will use a Reynolds Number of 120. In order to simplify the computation, the diameter of the cylinder is set to 1 m, the x component of the velocity is set to 1 m/s and the density of the fluid is set to 1 kg /m^3. Thus, the dynamic viscosity must be set to 8.333x10^-3 kg/m*s in order to obtain the desired Reynolds number.

Compared to the steady case, the unsteady case includes an additional time-derivative term in the Navier-Stokes equations:

$$\frac{\partial \vec{u}}{\partial t} + \rho(\vec{u} \cdot \nabla)\vec{u} = -\nabla p + \mu \nabla^2 \vec{u} \qquad (1)$$

The methods implemented by FLUENT to solve a time dependent system are very similar to those used in a steady-state case. In this case, the domain and boundary conditions will be the same as the Steady Flow Past a Cylinder. However, because this is a transient system, initial conditions at t=0 are required. To solve the system, we need to input the desired time range and time step into FLUENT. The program will then compute a solution for the first time step, iterating until convergence or a limit of iterations is reached, then will proceed to the next time step, "marching" through time until the end time is reached.

Go to Step 1: Pre-Analysis & Start-Up

Go to all FLUENT Learning Modules