SIMULATION OF FLOW AROUND BLUFF BODY BENCHMARK VALIDATION USING ABAQUS CAE

Prof. S.V.Pawar[1], Prof. S.L. Patekar[2]
[1,2]Mech. Dept. Bhivarabai Sawant College of Engineering and Research, Narhe, Pune.

Abstract

The simulation of flow around bluff bodies and, in which unsteady nature and vortex shedding of flow are commonly found, using computational fluid dynamics (CFD). Various turbulence models have been tested to develop understanding and proper modeling techniques for the flow around such bodies. The major part of the work discusses flow around bluff bodies ranging from a simple circular cylinder, a square cylinder to rectangular sections with various aspect ratios (1:2 to 1:8).

The Paper concentrates on modeling flow characteristics around bluff bodies to investigate the impact of fluid flow on them. The thesis combines investigation and discussion of the vortex shedding nature on the flow around bluff bodies, in which the simulations are done using advanced modeling techniques on high performance computing system which is ABAQUS CAE.

Index Terms: CFD, ABAQUS, CATIA, ANSYS.

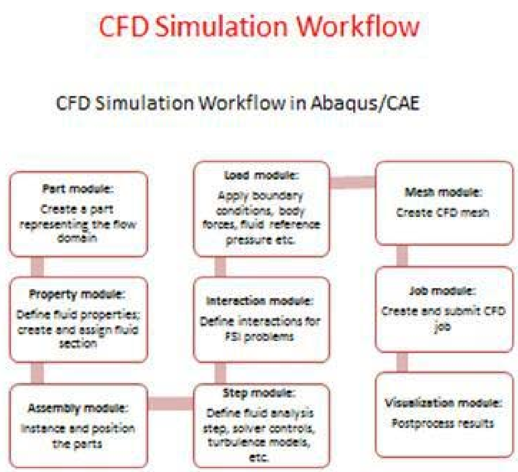
I.INTRODUCTION

The topic of this Seminar is the simulation of flow around bluff bodies using computational fluid dynamics (CFD). CFD calculates numerical solutions to the equations governing fluid flow. Bluff bodies are structures with shapes that significantly disturb the flow around them, as opposed to flow around a streamlined body. Examples of bluff bodies include circular cylinders, square cylinders and rectangular cylinders. To appropriately simulate these flow characteristics, turbulence models in CFD are used. Turbulence models are equations that account for turbulence of flow based on some assumptions, since it is computationally not practical to thoroughly represent all the physical characteristics of a flow using the current available computer power. The simulations start with the application of various turbulence models to flow around bluff bodies, starting from the basic circular cylinder to square ones followed by rectangular cylindrical bodies with increasing aspect ratio. Advantages and disadvantages of each turbulence model are identified based on comparative studies with experimental results. Modeling of the more complex flow around a bridge deck section is then carried out based on the findings from the study of the flow around bluff bodies.

The purpose of this Seminar is to illustrate the setup and solution of an unsteady flow past a circular cylinder and to study the vortex shedding process. Flow past a circular cylinder is one of the classical problems of fluid mechanics. The geometry suggests a steady and symmetric flow pattern. For lower value of Reynolds number, the Flow is steady and symmetric. Any disturbance introduced at the inlet gets damped by the viscous forces. As the Reynolds number is increased, the disturbance at the upstream Flow cannot be damped. This leads to a very important periodic phenomenon downstream of the cylinder, known as `vortex shedding'.

The present work aims to develop proficient and advanced CAD modeling methodologies for the simulation of flow around bluff bodies using CFD ABAQUS

II. DIFFERENT TYPES OF MODULES IN CFD



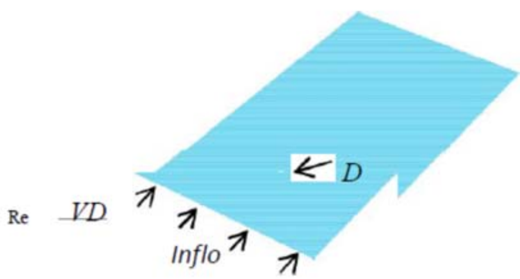
III. CASE STUDY

Flow around a rigid circular cylinder is often used as a CFD benchmark case

a) Characteristic length scale: D (cylinder diameter)

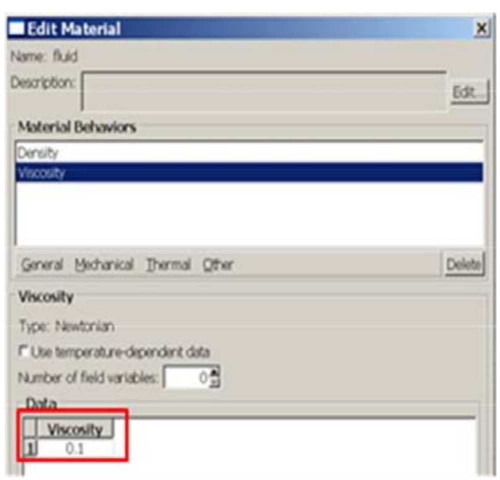
b) Model the flow as 3-dimensional but with one element through the thickness and symmetry boundary conditions on the front and back faces to enforce 2-dimensional conditions

c) V inlet =0.1 m/sec D = 0.1 m

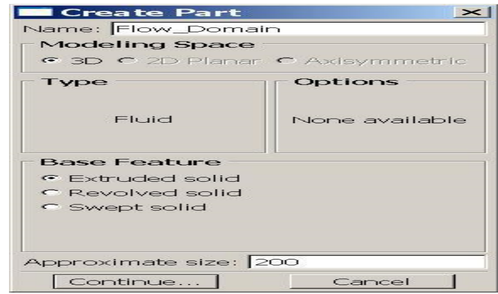


IV. METHODOLOGY

1. Create a "CFD" model in Abaqus/CAE



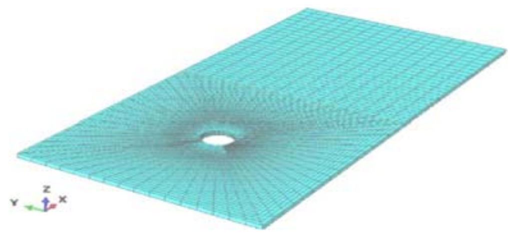
2. Define a part representing the flow domain



3. Generate the mesh

a) Hexahedral (FC3D8) and tetrahedral (FC3D4) element types are available

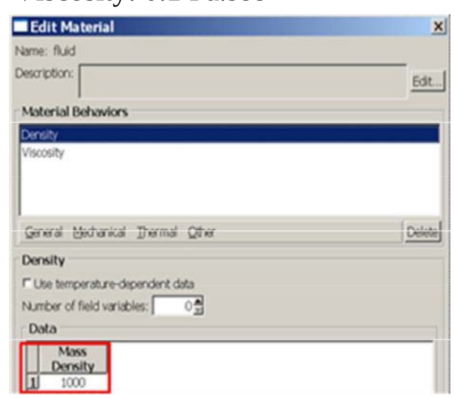
b) No mixed meshes are allowed



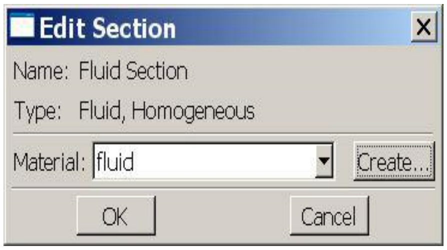
4. Define fluid material properties Newtonian fluid

a) Density: 1000 kg/m3

b) Viscosity: 0.1 Pa.sec



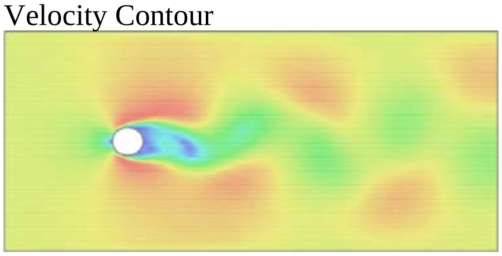
5. Create and assign a fluid section and instance the part



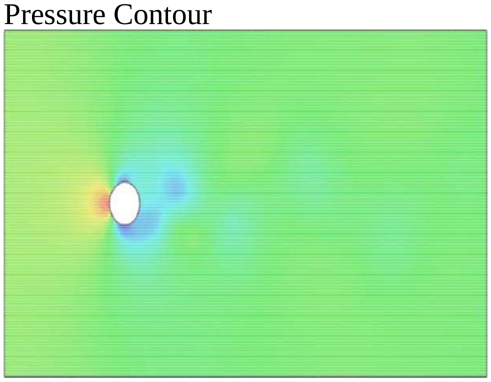
6. Define boundary conditions
i. Flow inlet Vx = 0.1, Vy = 0, Vz = 0
ii. Flow outlet, p = 0
iii. Wall (no-slip, no-penetration),Vx = 0, Vy = 0, Vz = 0
iv. Far field Vx = 0.1, Vy = 0, Vz = 0

V. RESULTS

Velocity Contour



Pressure Contour



VI. CONCLUSION

CFD is Better Tool to Evaluate the Industrial Problem like Turbine Flow Analysis, Heat Exchanger Analysis, and Flow Simulation around Chimney. Above Problem is a Benchmark Problem for Solver Validation of any CFD Software, Like Fluent, CFX etc. It has Ability to simulate real conditions; CFD provides the ability to theoretically simulate any physical condition. Also simulations can be executed in a short period of time.

REFERENCES

[1] Kai Fan Liaw, “Simulation of Flow around Bluff Bodies and Bridge Deck Sections using CFD”, the University of Nottingham, June 2005.

[2] ADPAC sample case: Vortex shedding over a circular cylinder in cross flow, www.grc.nasa.gov/WWW/5900/5940/code/adpac/sample. “

[3] Al-Jamal H, Dalton C, Vortex induced vibrations using Large Eddy Simulation at a moderate Reynolds number, Journal of Fluids and Structures, 2004.

[4] Blazek J, Computational Fluid Dynamics: Principles and Applications, Elsevier Science Ltd, Oxford England,2001.