

ENE 503 – Computational Fluid Dynamics

WEEK 12: TUTORIALS

TUTORIALS:

- Fluent Tutorial 1: Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow R18.0

- This tutorial illustrates using ANSYS Fluent fluid flow systems in ANSYS Workbench to set up

and

- To solve a three dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow.

In this tutorial, following below basic steps are accomplished one by one:

- ANSYS Workbench is launched*
- A fluid flow analysis is created using FLUENT in ANSYS Workbench*
- The elbow geometry is generated using ANSYS Design Modeller*
- The computational mesh is created using ANSYS Meshing*
- The CFD simulation is set up in FLUENT by the following below procedures:*
 - Material properties are set*
 - Boundary conditions are imposed*
 - The calculation is initiated with residual plotting*
 - Calculation is initiated by using pressure-based solver*
 - The flow and temperature fields are analysed by in CFD Post*
 - A Copy of the original FLUENT fluid flow analysis system is created in ANSYS Workbench*
 - The geometry is changed in ANSYS Design Modeller using the Duplicated system*
 - The computational mesh is regenerated*
 - A solution is recalculated in FLUENT*

- *The results of the two calculations are compared in CFD-Post.*

➤ **Fluent Tutorial 2: Parametric Analysis in ANSYS Workbench Using ANSYS Fluent R18.0**

- *This tutorial illustrates using ANSYS Workbench to set up and solve a three-dimensional turbulent fluid flow and heat transfer problem in an automotive heating, ventilation, and air conditioning (HVAC) duct system using an ANSYS Fluent fluid flow system.*

Following below procedures are accomplished here:

- *Constraints to the ANSYS DesignModeler input parameters are added*
- *ANSYS Fluent fluid flow analysis system in ANSYS Workbench is created*
- *CFD simulation in ANSYS Fluent is set up as below:*
 - *Material properties and boundary conditions are set up*
 - *Input parameters are defined*
 - *Output parameters in CFD-Post*
 - *Create additional design points in ANSYS Workbench.*
 - *Run multiple CFD simulations by updating the design points.*

➤ **Fluent Tutorial 3: Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow R18.0**

- *This tutorial illustrates the setup and solution of a three-dimensional turbulent fluid flow and heat transfer problem in a mixing elbow, using both the serial and the parallel version of ANSYS Fluent.*

➤ **Fluent Tutorial 4: Modeling Periodic Flow and Heat Transfer R18.0**

- *This tutorial illustrates how to set up and solve a periodic heat transfer problem, given a pre-generated mesh.*

➤ Fluent Tutorial 5: Modeling External Compressible Flow R18.0

- The purpose of this tutorial is to compute the turbulent flow past a transonic airfoil at a nonzero angle of attack, using the Spalart-Allmaras turbulence model.

➤ Fluent Tutorial 6: Modeling Transient Compressible Flow R18.0

- In this tutorial, ANSYS Fluent's density-based implicit solver is used to predict the time-dependent flow through a two-dimensional nozzle with a transient boundary condition implemented as a using a user-defined function (UDF).

➤ Fluent Tutorial 22: Post processing R18.0

- This tutorial demonstrates the post processing capabilities of ANSYS Fluent using a 3D model of a flat circuit board with a heat generating electronic chip mounted on it.

➤ Fluent Tutorial 23: Using the Adjoint Solver - 2D Laminar Flow Past a Cylinder R18.0

- This tutorial provides an example of how to use the adjoint solver to generate sensitivity data for flow past a circular cylinder, how to postprocess the results, and how to use the data to reduce drag by morphing the mesh.