ENE 505 – Applied Computational Fluid Dynamics in Renewable Energy Technologies

WEEK 12: TUTORIALS

<u>TUTORIALS:</u>

- 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*.