Computational Fluid Dynamics

Lecture 6

by
Dr. A. Nurye
Faculty of Mechanical Engineering
nurye@ump.edu.my

Stages of CFD Analysis

- Aims
  - The aim of this chapter is to introduce students the main stages of CFD analysis

- Expected Outcomes: At the end of this lecture, students should be able to understand
  - elements of CFD analysis
  - how to generate computational grid

- References

1) J. Tu, G.H. Yeoh, C. Liu, Computational Fluid Dynamics : A Practical Approach, Elsevier, 1st Edition, 2013.
2) C.T. Shaw, Using Computational Fluid Dynamics, Prentice Hall, 1992

Contents

Stages of CFD analysis

- Pre-processing
- Solving
- Post-processing

Introduction

- There are many free and commercial CFD packages available.
- Popular non-commercial CFD solvers include OpenFOAM (http://www.openfoam.com/) CodeSaturne (http://code-saturne.org/cms/)
- An excellent web portal for all things CFD is http://www.cfd-online.com/.
- For this course, the commercial software called Ansys Fluent will be used for the lab exercise and projects.

- Some of the commercial CFD codes include :

Ansys CFD,

Comsol Multiphysics

Star-CD,

CFDRC, CFX/AEA, etc.

CFD solution includes three major processes

- Geometry creation
- Material properties selection
- Model selection
- Setting of initial and boundary conditions
- Mesh generation
- Specification of numerical parameters
- Flow solution
- Results visualization and analysis
- Uncertainty assessment

Pre-processing

Solving

Post-processing

Pre-processing

- This stage is the first step in CFD analysis. It involves

  I. definition and creation of the geometry of the flow region;

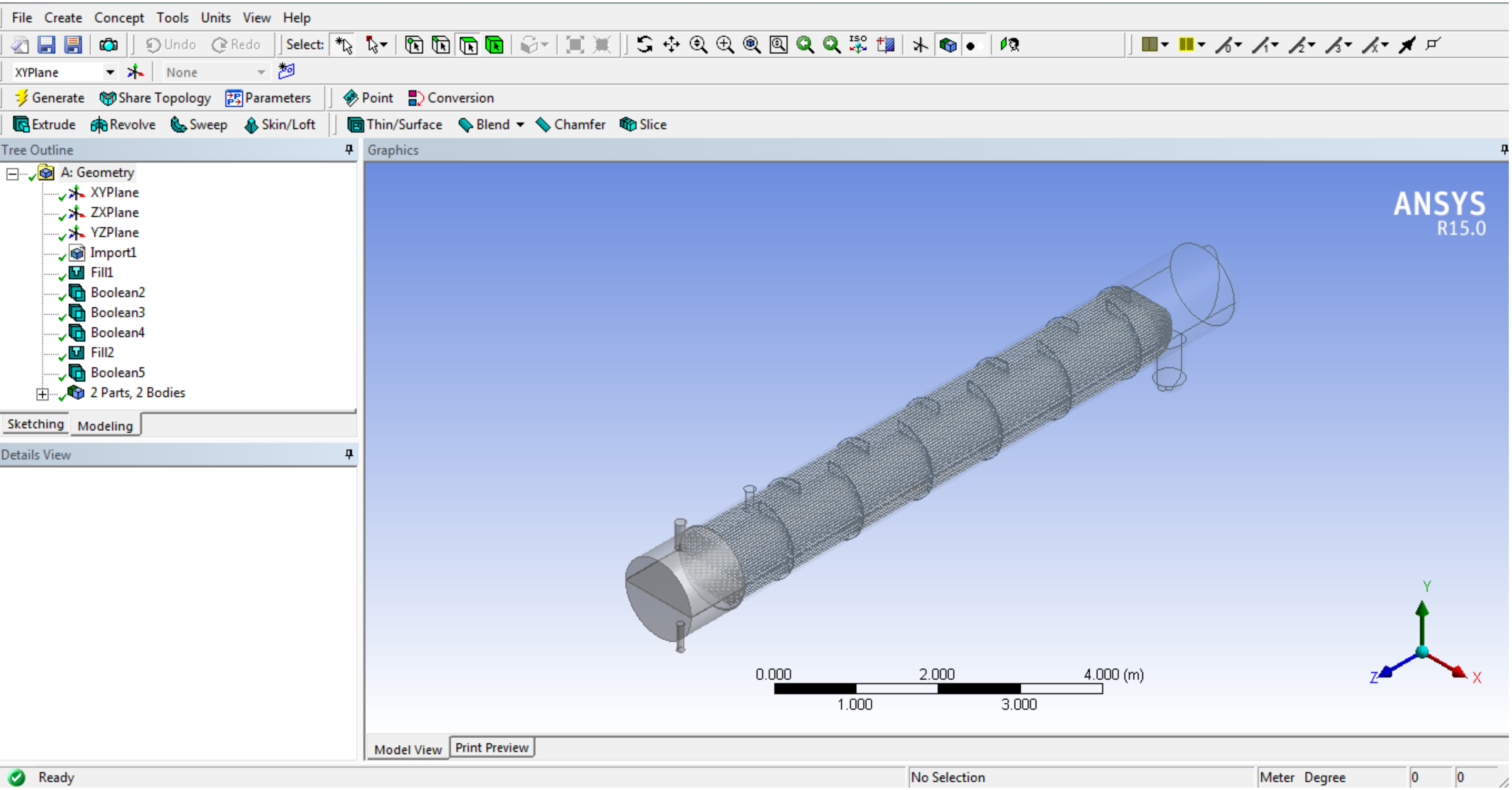
  II. mesh generation;

  III. specifying the physics (equations to be solved) and fluid properties;

  IV. specifying the boundary conditions

I. Creating the computational domain (geometry)

- It is representation of the real problem that is to be solve.
- Before creating/importing the computational domain, necessary assumptions and simplifications need to be done in relation to the intended analysis.
- Simple geometries can be easily created by few geometric parameters (e.g. circular pipe) using Ansys Workbench.
- Complex geometries can be created CAD/CAE or Solidworks or other similar software and importing the geometry (e.g. shell and tube heat exchanger) into Ansys Fluent.



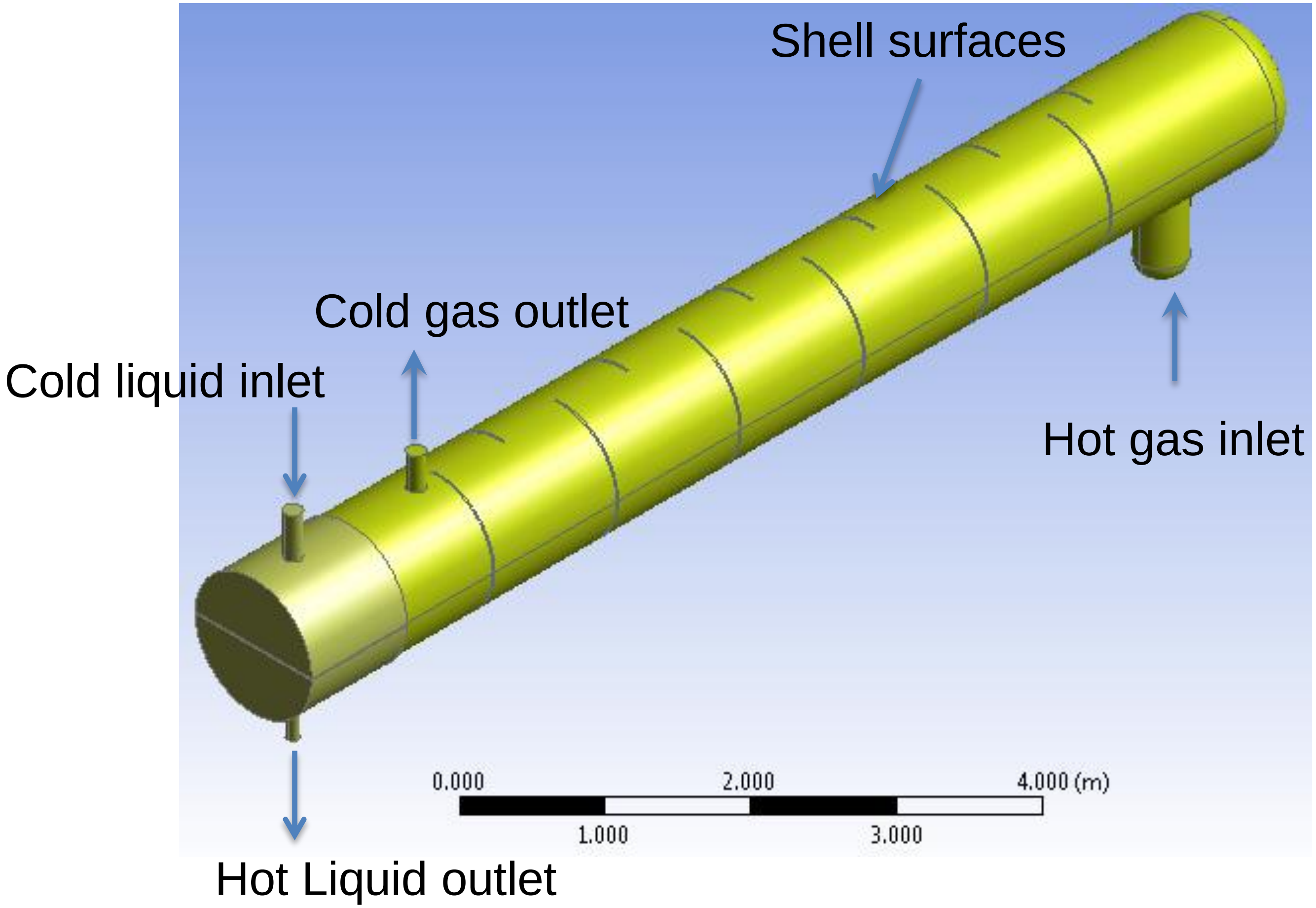
A typical Ansys Fluent Geometry: DesignModeler

Material properties and Model selection

- For a given problem, you will need to:
  - Select appropriate physical models.
    - Turbulence, combustion, multiphase, etc.
  - Define material properties.
    - Fluid
    - Solid
    - Mixture
  - Prescribe operating conditions.
  - Prescribe boundary conditions at all boundary zones.
  - Provide an initial solution.
  - Set up solver controls.
  - Set up convergence monitors.

Initial and boundary conditions

- Initial condition should not affect the final solution, only convergence path, i.e. iteration numbers needed to get the converged solution.
- But more reasonable guess can speed up the convergence.
- Boundary conditions
  - No-slip or slip-free on the wall, periodic, inlet (velocity inlet, mass flow rate, constant pressure, etc.), outlet (constant pressure, velocity convective, buffer zone, zero-gradient), and non-reflecting (compressible flows, such as acoustics), etc.

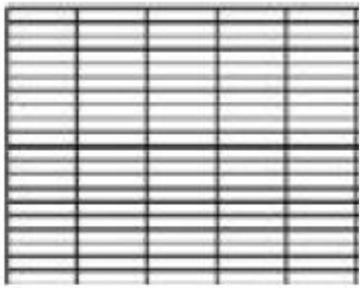


Heat exchanger model using Ansys Fluent

II. mesh generation

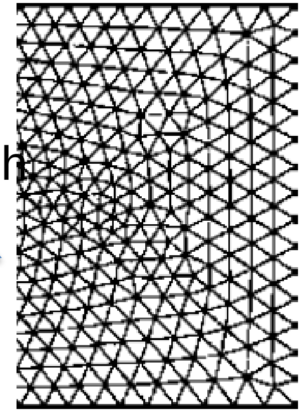
- Meshing is splitting of flow domains in to many smaller sub-domains
- The type of grid (mesh) used is different for 2D and 3D cases
  - ✓ Triangular or quadrilateral grids can be used in 2D meshing
  - ✓ Tetrahedral or hexahedral grids can be used in 3D meshing
- Depending on the discretization scheme and application of the simulation, either structured or unstructured mesh type can be used.

- In complex geometries unstructured grids can be generated faster than structured grids. However, the opposite is true for simple geometries.
- In terms of accuracy, structured meshes are more accurate for simpler problems. However, for more complex flows, the adaptivity facilitated by an unstructured grid may allow more accurate solutions.
- Structured meshes are advantageous in terms of calculation time as they take less time to calculate the problem compared to unstructured meshes.

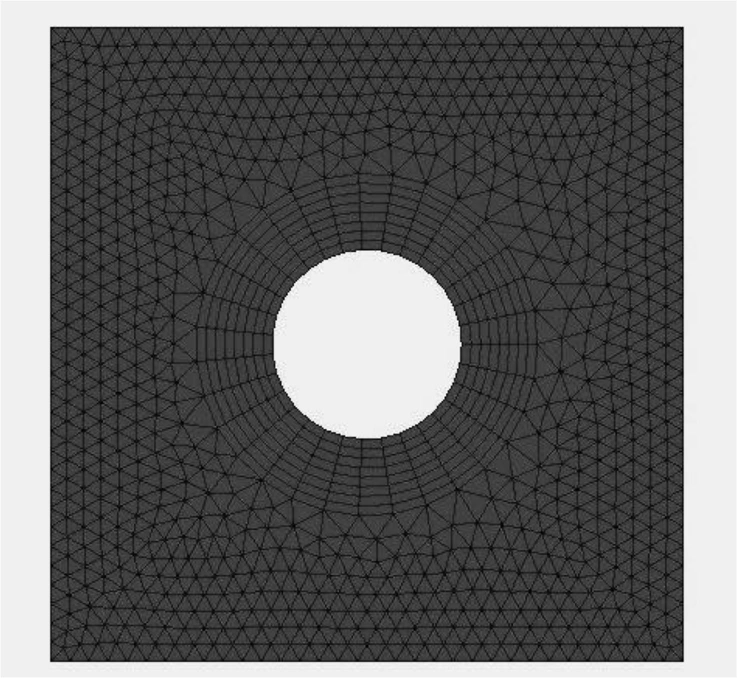


Example of structured Mesh

Example of unstructured Mesh

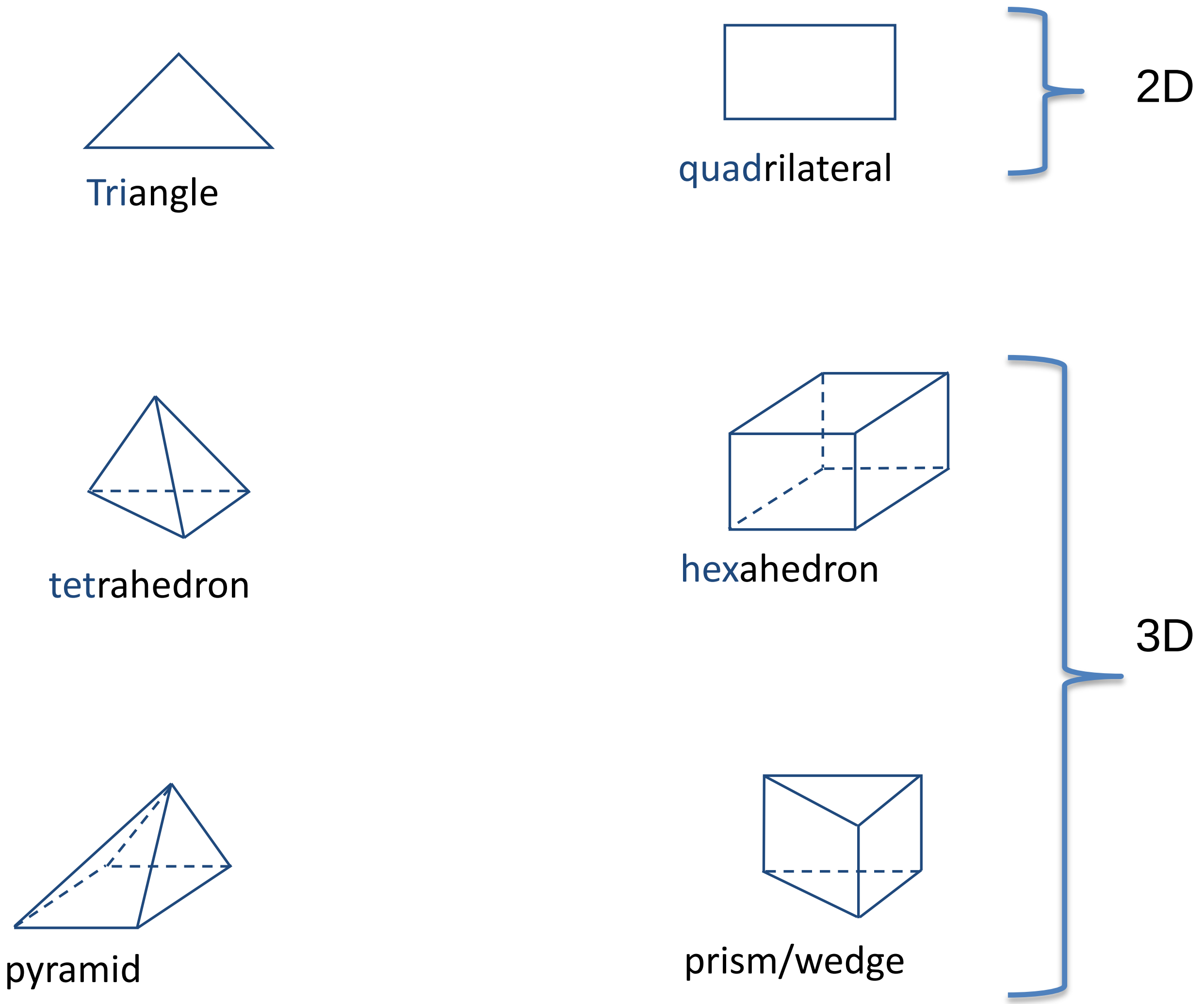


- In some case mixture of structured and unstructured girds called hybrid mesh can be used



https://commons.wikimedia.org/wiki/File:Unstructured_Grid.jpg

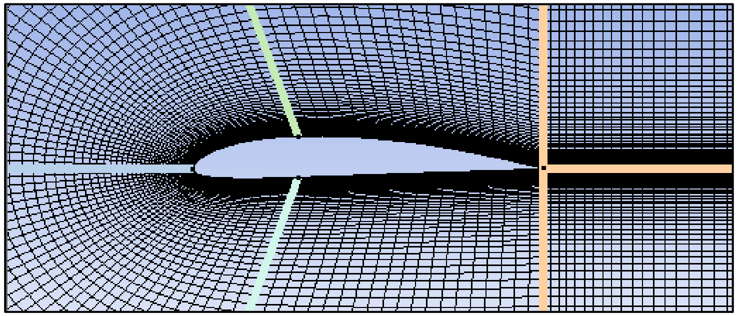
Types of grid elements



If you are not using commercial softwares, note that the following mesh properties are desired for accurate flow prediction:

i. Two adjacent grids should have the same node points

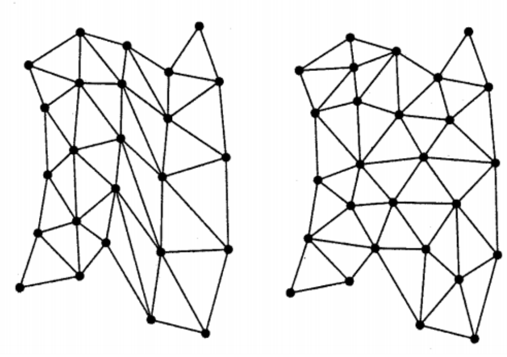
ii. Finer meshes must be applied near a fluid boundary layer and other similar boundaries.



A typical example of fine mesh near an airfoil walls generated by Ansys Fluent

iii. Select meshes than can be easily generated.

iv. Pay attention to the orthogonality of the mesh edges. Meshes with very small or very large angles may lead to inaccurate results, and affect the convergence.



Poor Mesh   Good Mesh

v. Try to minimize the number of meshes as much as possible.

Exercise

Following the step-by-step guidelines provided in Kalam, create geometry and mesh for the air flow through a circular tube.

After finishing this exercise, you should be able to understand the difference between

- sketching and modelling modes
- DesignModeller and Fluent-Mesh
- surface and volume Meshes

Dr. A. Nurye

Research interest:

- Computational Fluid Dynamics,
- Thermo-fluids,
- Multidisciplinary Numerical Modelling and Simulation

Contact:

Tel: +094246259

email: nurye@ump.edu.my