

# Computational Fluid Dynamics

## Lecture 7

by  
Dr. A. Nurye  
Faculty of Mechanical Engineering  
[nurye@ump.edu.my](mailto:nurye@ump.edu.my)

# Setting the boundary conditions and solver

- 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
  - how to apply boundary conditions
  - how to set the solver
- 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

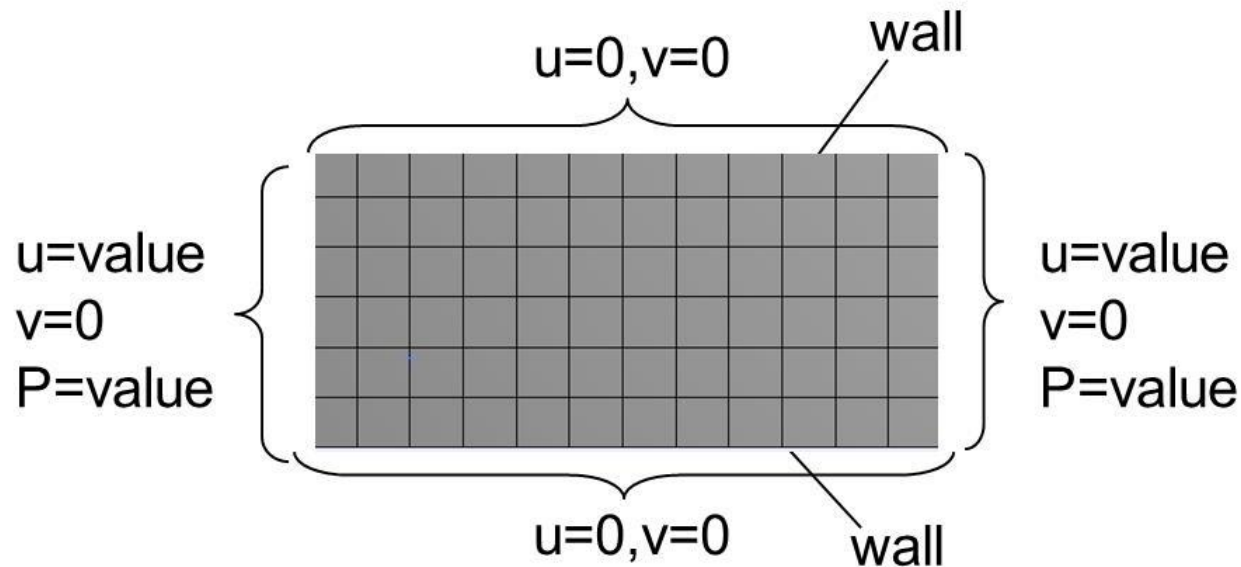
# Specifying Fluid Properties

- Different fluids have different thermophysical properties. To calculate the thermo-fluid solutions, those properties should be specified to the solver.
- The most common thermophysical properties required to solve any thermofluid problem using CFD are
  - ✓ density,
  - ✓ viscosity,
  - ✓ thermal conductivity and
  - ✓ specific heat capacity.
- ✓ These properties may be constant or varying with temperature or pressure

### iii. How to apply boundary conditions

- One of the most important step in CFD calculation is applying appropriate boundary conditions (BC).
- It involves identifying the location of the boundaries and providing information at the boundaries.
- When applying BCs, make sure that they are not under defined or over defined.
- If the BCs are under defined, the generated data may not be sufficient.
- On the other hand, if the BCs are over defined, they may produce conditions which are physically impossible.

- Consider, model of 2D air flow through a circular pipe as shown below



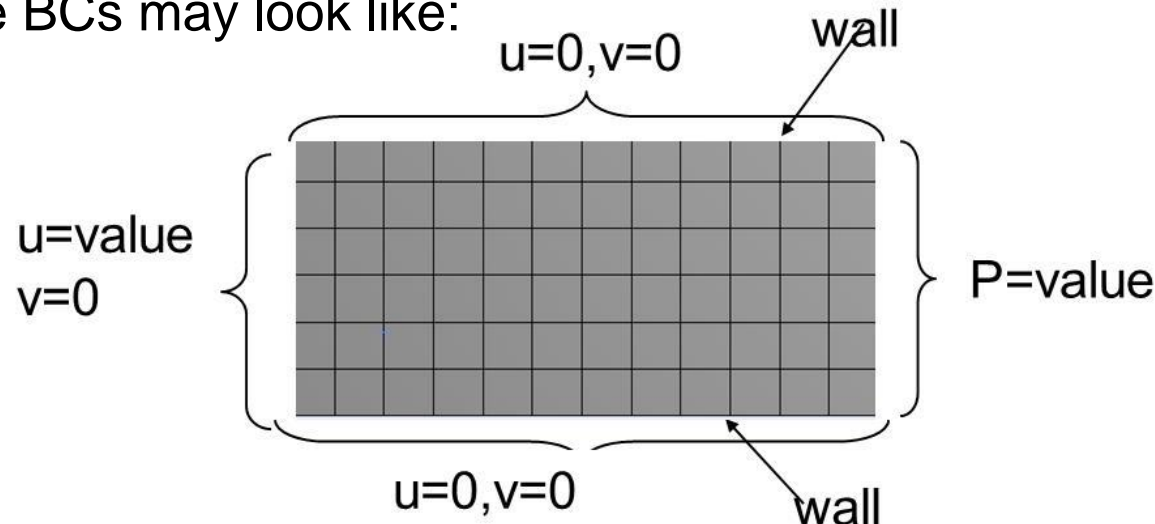
The above model is considered as over-defined because the velocity and pressure are specified at inlet and outlet.

Thus, values may not satisfy the continuity and momentum equations.

Some of the factors that affect the determination of boundary conditions depends on the nature of the problem to be solved

- steady or unsteady flow,
- incompressible or compressible,
- inviscid or viscous

Consider the previous example of air flow through a 2D circular pipe for inviscid, steady, and incompressible flow, solved by pressure based method. The BCs may look like:



Some of the common boundary types in Fluent are the following

- Inlet and outlet boundaries: these boundary types include
  - ✓ Velocity.
  - ✓ Pressure boundaries and others.
- Wall, symmetry, periodic and axis boundaries.
- Internal face boundaries.
- Interface boundaries: when two different zones share same boundary

All the values of the flow/thermal variables at the boundaries can be prescribed as

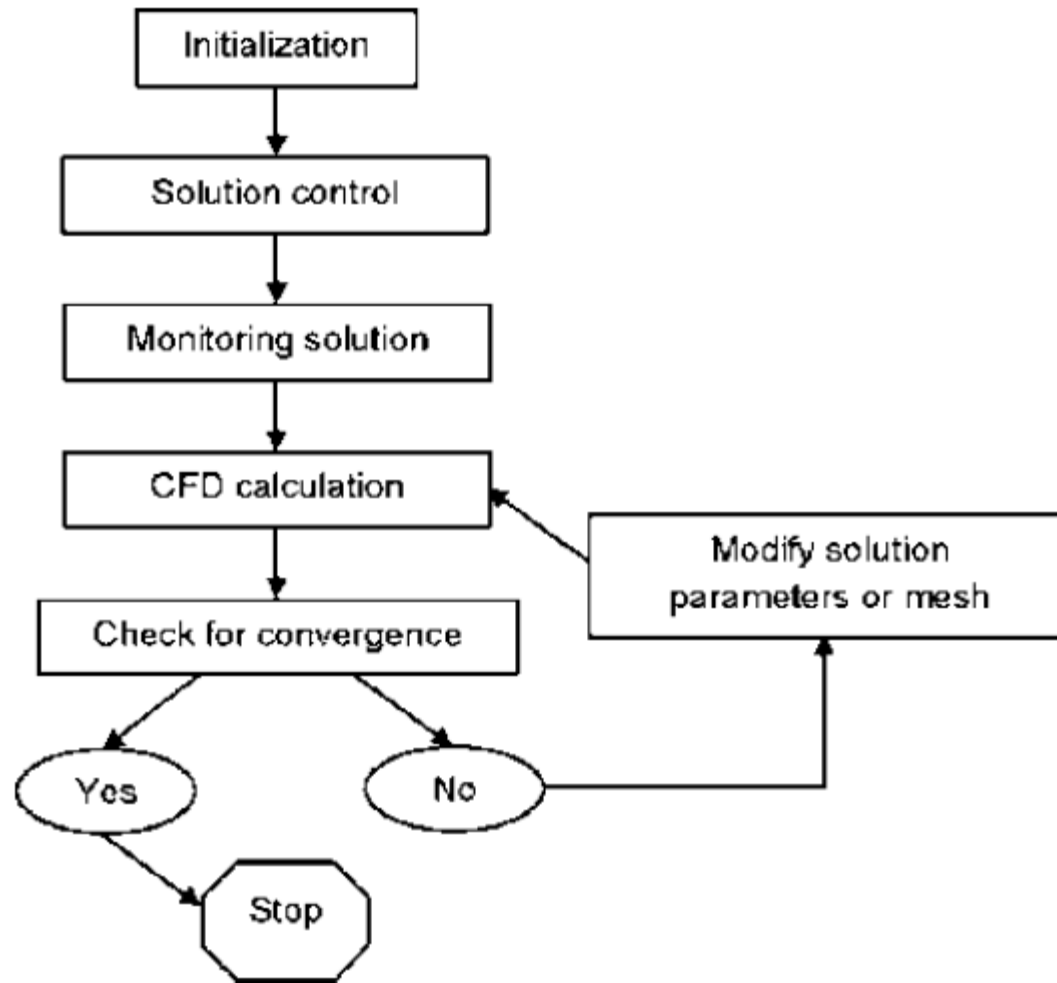
- constant values (Dirichlet boundary condition). For instance, constant inlet velocity  $v(x)=\text{constant}$
- gradient normal to the boundary of a variable (Neumann boundary condition)
- Mixed boundary conditions (Dirichlet +Neumann)



## iv. CFD Setup and Flow Solution

- In Ansys Fluent, this stage involves defining and setting the
  - I. Iteration procedure;
  - II. solution controls and monitors;
  - III. running the calculation

# Solution procedures



An overview of a solution procedure [1]

## Iteration process

- The solution is initialized by using guessed values of flow and thermal variables at the beginning of the solution process.
- Then the continuity, momentum, and energy governing equations will be used to calculate the accurate approximations.
- These approximate values will then be used as initial guess for the next iteration.
- The process is repeated many times until the final solutions are very small (less than the give tolerance values). This is called **Iteration process**.

- The iteration process can be controlled by
- i. using relaxation parameters (for steady state problems) :-
    - In this case, the variables determined during the new iteration will be scaled so that the solution used in the next iteration is not very different from the values used at the beginning of the new iteration.
- $$u_{new} = \lambda u_{calc} + (1 - \lambda) u_{prev}$$
- $u_{cal}$  = Velocity calculated at the end of the iteration
- $u_{prev}$  = Velocity calculated at the beginning of the iteration
- $\lambda$  = Under – relaxation factor ( $0 < \lambda < 1$ )
- ii. time dependent solution scheme (for both steady and unsteady flow problems)

## Solution Methods

- As the pressure field is unknown, additional scheme is required for calculating pressure.
- The scheme can be
  - ✓ SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) – Guess and correction

Normally the pressure equation are not available, thus, at positions near the boundary, boundary conditions are not required for the pressure.

Instead, linear extrapolation will be used to calculate the values of pressure at the boundaries from the two near-boundary node pressures.

- Other Improved versions of SIMPLE than can speed up convergence include:
  - ✓ SIMPLER (SIMPLE Revised).
  - ✓ SIMPLEC (SIMPLE Consistent).
  - ✓ PISO (Pressure Implicit with Splitting of Operators).
- Which scheme make the solution converges fastest depends on the type of problem being solved.

# Dr. A. Nurye

## Research interest:

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

## Contact:

Tel: +094246259

email: [nurye@ump.edu.my](mailto:nurye@ump.edu.my)