TURBOMACHINERY AERODYNAMICS

Lect- 38

Prof. Bhaskar Roy, Prof. A M Pradeep
Department of Aerospace Engineering,
IIT Bombay

In this lecture...

- Computational Fluid Dynamics for turbomachinery
  - Introduction and overview
  - Grid generation
  - Boundary conditions

Computational Fluid Dynamics

- Computational Fluid Dynamics (CFD) is a powerful analytical tool.
- Is a third approach for analysis besides experimental approach and theoretical approach.
- CFD compliments theory and experiments and is not primarily intended to replace these.
- CFD is currently a commonly used research tool.
- CFD is an essential component of the design, analysis and optimization cycle.

Computational Fluid Dynamics

- There are various levels of CFD analysis
  - Simple Euler (potential flow) solutions
  - 2-D/axisymmetric Navier-Stokes solution
  - 3-D Navier-Stokes solution
    - Reynolds Averaged Navier-Stokes (RANS) and Unsteady RANS (URANS)
    - Large Eddy Simulation (LES)
    - Direct Numerical Simulation (DNS)
- CFD analysis could also be
  - Steady or unsteady
  - Incompressible or compressible
  - Laminar or turbulent
  - Internal or external flow

Computational Fluid Dynamics

- CFD involves solving the fundamental governing equations of fluid flow:
  - Conservation of mass
  - Conservation of momentum
  - Conservation of energy
  - Equation of state
  - Species conservation (reacting flows)

Computational Fluid Dynamics

- Steps in CFD solution
  - Setting up the domain
  - Discretisation of the domain in space and time (for unsteady solution)
  - Defining boundary conditions
  - Solving the appropriate governing equations for the domain on the discretised points
  - Post-processing and analysis of the converged solution.

Computational Fluid Dynamics

- Turbomachinery: complex shear flows
    - Shear layers on rotating surfaces
    - Shear layers developing on curved surfaces
    - Separated flows: shock-boundary layer interaction, corner separation...
    - Swirling flows and vortices
    - Interacting boundary layers

Computational Fluid Dynamics

- Challenges in turbomachinery CFD
  - Grid generation
    - Complex geometry
    - Rotating domain
  - Flow is 3-D, highly unsteady, rotating, and turbulent
    - Capturing the losses and other viscous effects
    - Turbulence modelling
  - Fluid-structure interactions

Computational Fluid Dynamics

- Types of simulations
  - 2D, quasi-3D, 3D
  - 2D
    - Conceptual design phase
    - Long blades/vanes (LP turbines)
    - Reasonable results
  - Quasi-3D
    - Area of flow path changes
    - Extra source terms for acceleration/deceleration or boundary layer growth

Computational Fluid Dynamics

- Types of simulations
  - 3D
    - True geometry required
    - Simulate secondary flows, shock locations
    - End wall boundary layers
  - Transient or stationery
    - Stationery simulations more common
    - Transient: flow unsteadiness, vortex shedding, wake interaction with rotors

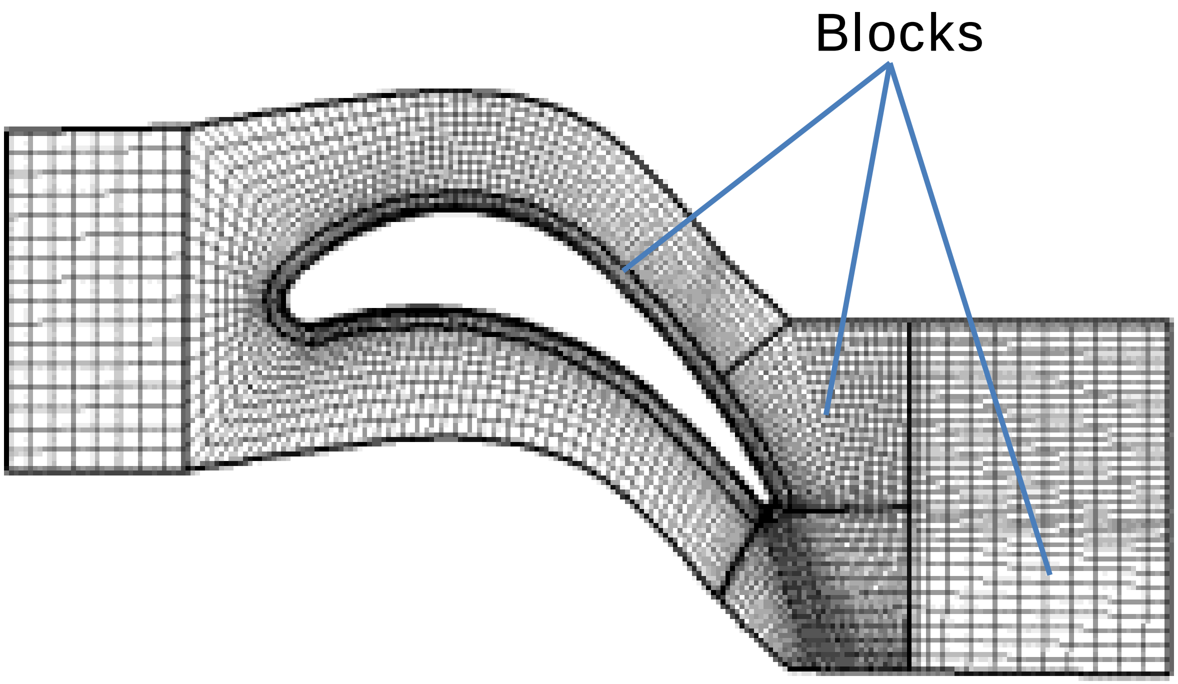
Computational Fluid Dynamics

- Solver
  - Euler
  - 3D NS
  - RANS, URANS
  - DES, DDES
  -
  -
  - LES
  -
  - DNS

Grid Generation

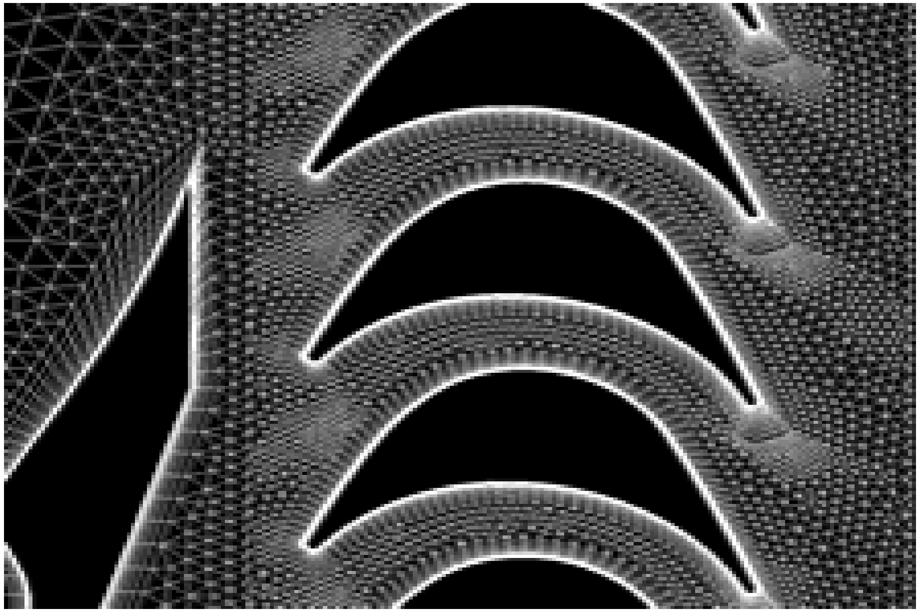
- Grid/mesh
  - Structured, unstructured and hybrid grids
- Structured grid
  - More suited for well-defined geometries
  - More difficult to generate
  - Easier to control near-wall clustering of cells
- Unstructured grid
  - Primarily intended for complex geometries
  - Easier to generate
  - Not much control over the near-wall clustering of cells
  - Easily automated

Grid Generation



Structured grid with multiple blocks

Grid Generation



Unstructured grid

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay

Grid Generation

- Multi-block structured grid
  - In order to generate structured grid over curved surfaces, multiple blocks need to be defined.
  - Interface of the blocks need to be carefully managed.
  - Grid topology needs to be appropriately defined.
  - The Grid topology also needs to account for the change in geometry of the blade from hub to tip.

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay

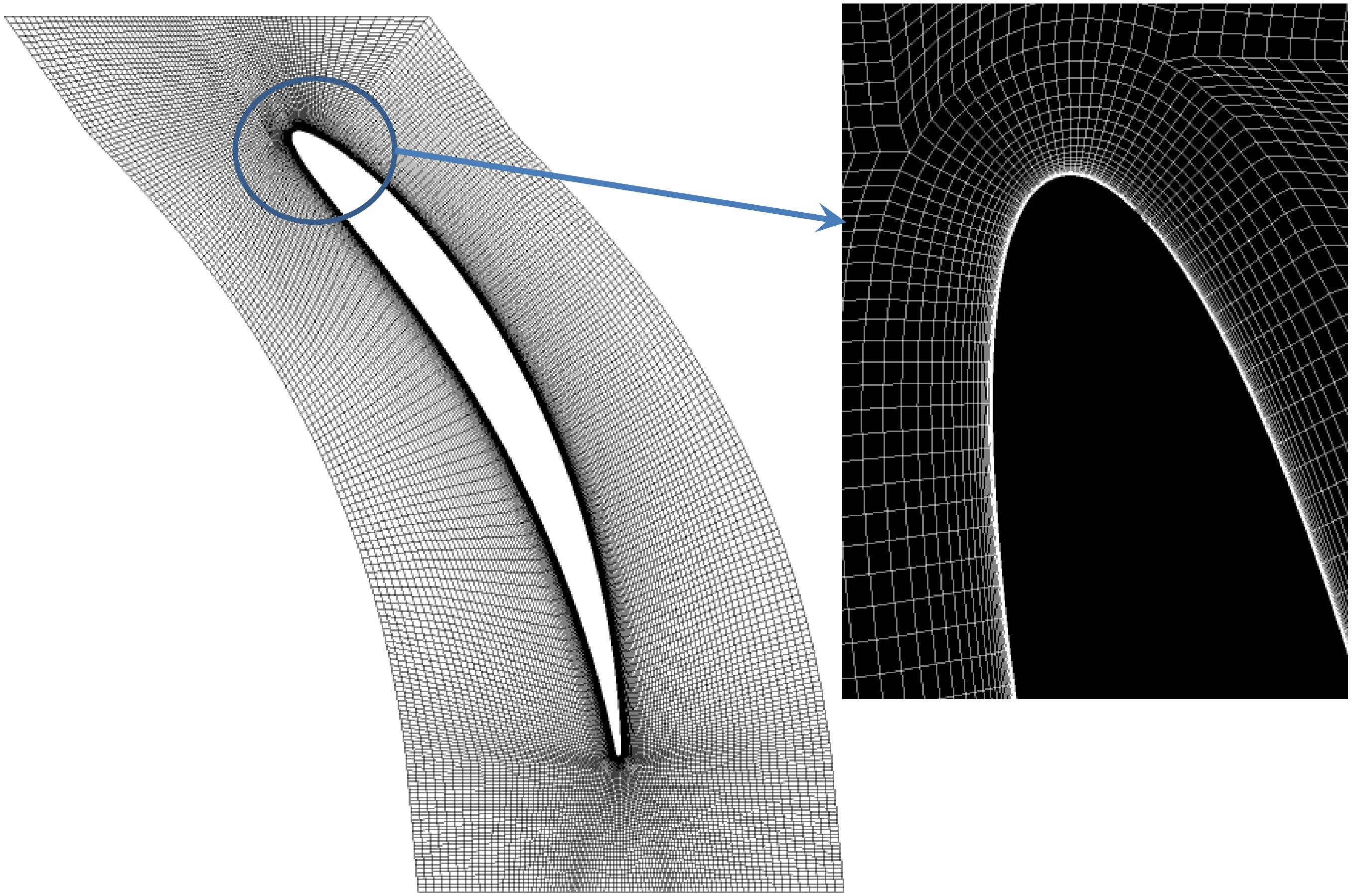
Grid Generation

- Topology
  - Is a structure off blocks that acts as a framework for placing mesh elements.
  - Blocks are laid out without gaps with shared edges and corners.
  - Blocks contain same number of elements along each side.
  - Is usually invariant from hub to tip.
  - Can be edited on 2-D layers from hub to tip sections.
  - Number of blocks will dictate the skewness of the grid elements.

Grid Generation

- Grid topology schemes
  - O-grid:
    - Usually used around the blade by forming a continuous loop around it
    - Yields excellent boundary layer resolution
    - gives good control over the y^+ values that needs to be tightly monitored
    - Provides near orthogonal elements on the blades

Grid Generation



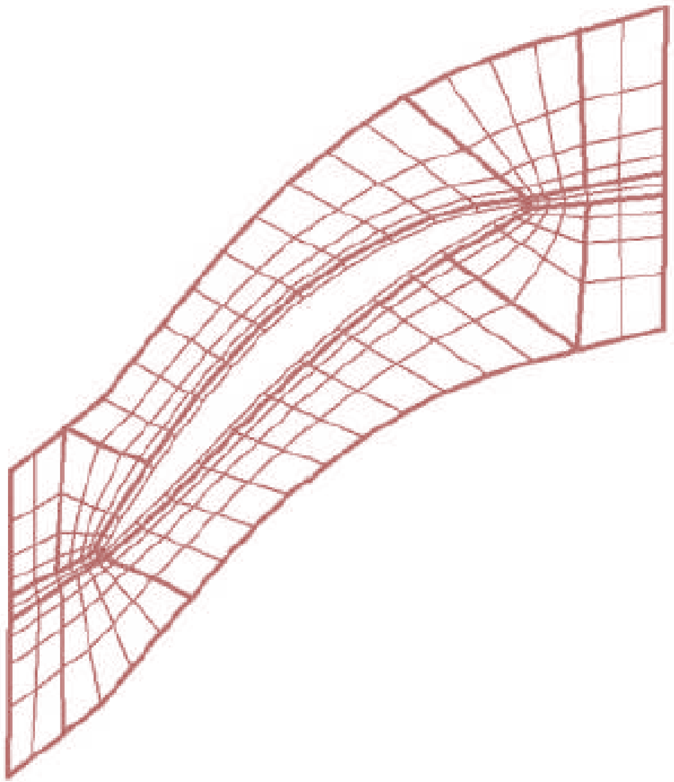
O-grid topology

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay

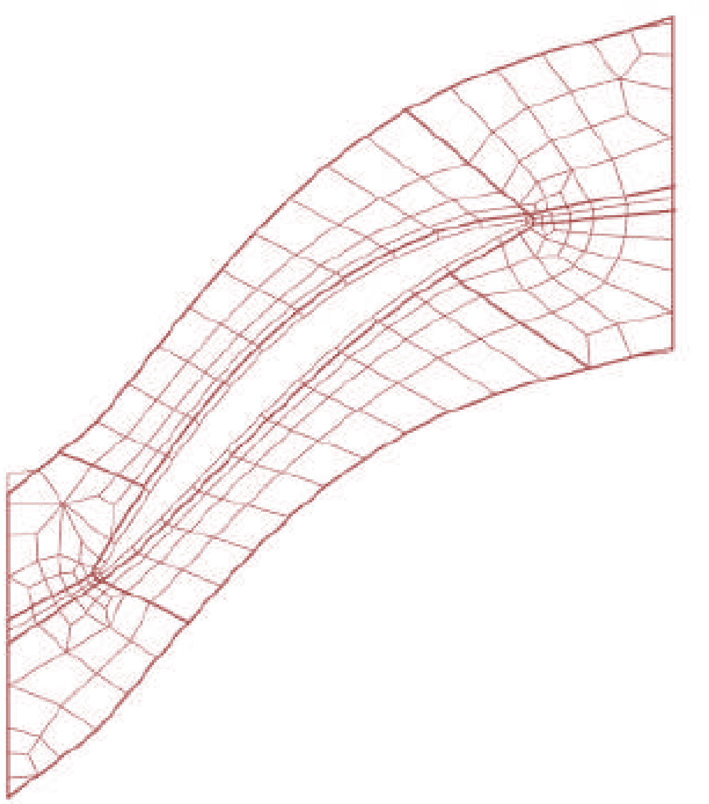
Grid Generation

- J-grid:
  - Usually used near leading and trailing edges
  - Wraps up in opposite directions at the leading and trailing edges
- H-grid:
  - Tends to complete the meshing by adding some blocks in an unstructured manner
  - The structured blocks extend from upstream of the LE, downstream of the TE and between the blades and the periodic surfaces

Grid Generation



J-grid topology

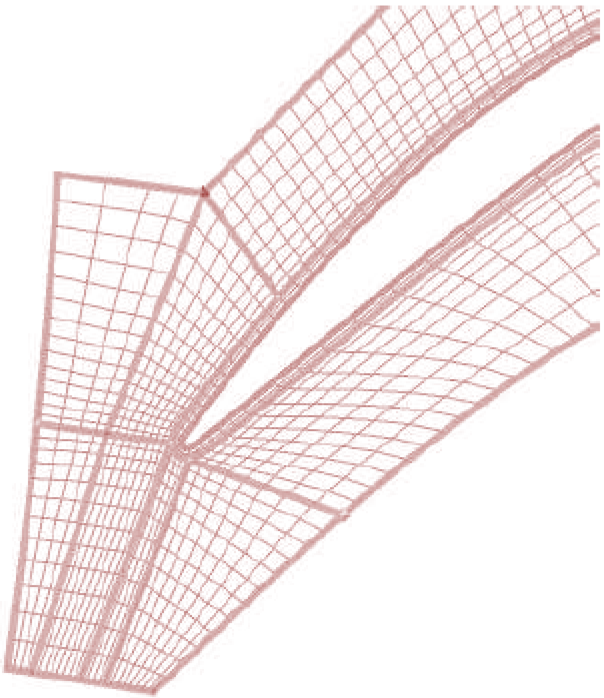


H-grid topology

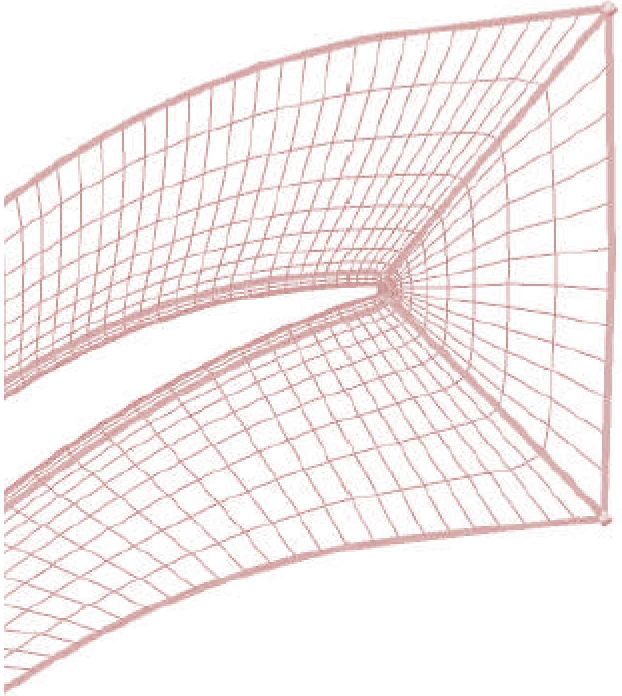
Grid Generation

- Other topology options include C-grid and L-grid.
- These are also often used at the leading and trailing edges.
- All the above grid topologies are used along with an O-grid for proper resolution of the boundary layer.
- Proper resolution of the leading and trailing edge radii are important.
- Establishing grid-independence or grid-insensitivity of the results is now a standard practice.

Grid Generation



L-grid topology



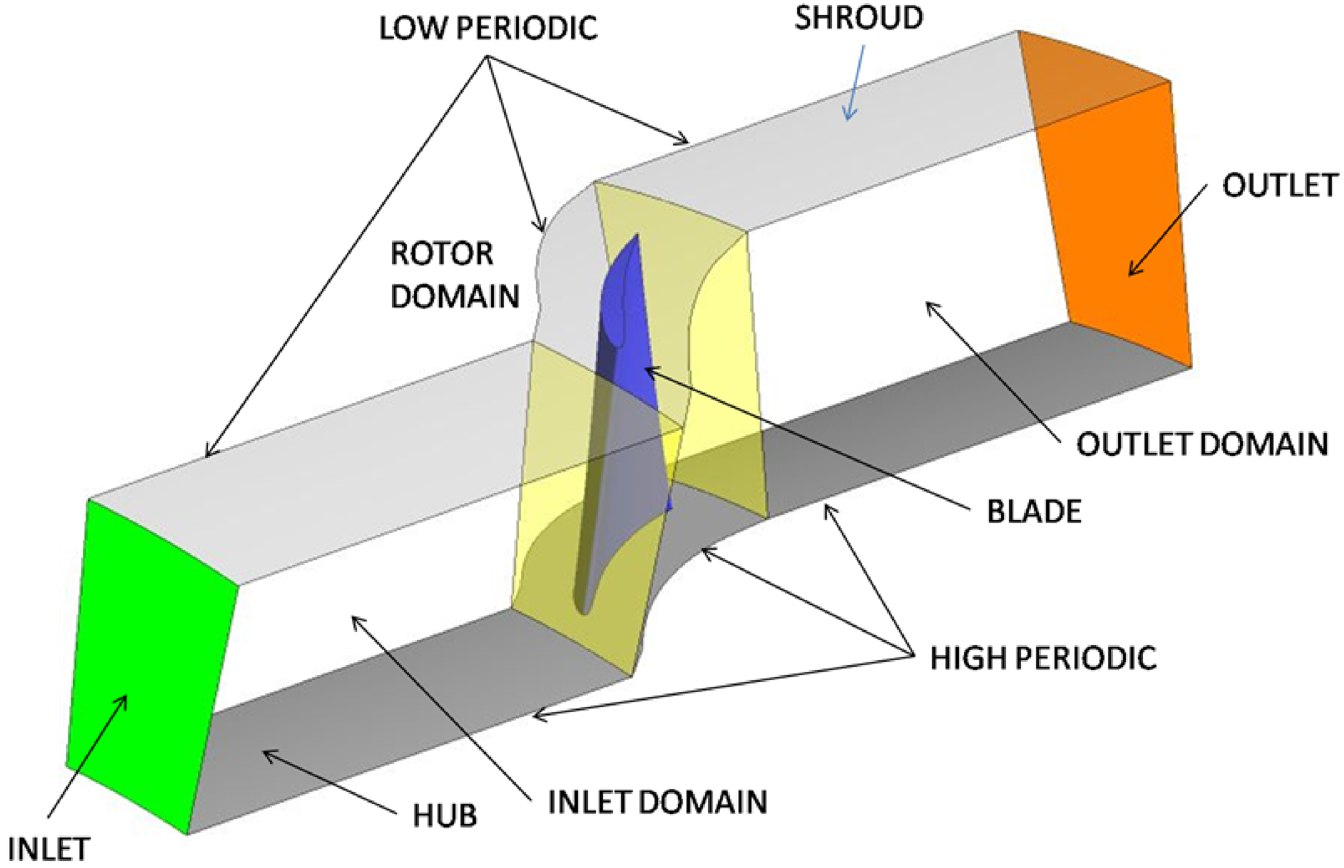
C-grid topology

Boundary conditions

- To capture the flow physics correctly, the boundary conditions must be set appropriately.
- Quality of the solutions is a strong function of the boundary conditions.
- Turbomachinery flows
  - Inlet boundary
  - Exit boundary
  - Periodic boundary
  - Walls or surfaces

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay

Boundary conditions



Typical flow domain with the boundaries

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay

Boundary conditions

- Inlet boundary conditions
  - Depends upon the application
  - Flow conditions (incompressible or compressible)
  - Total pressure, total temperature, velocity components/profile (most commonly used)
  - There are other forms of specifying the inlet boundary conditions: velocity inlet, mass flow inlet etc.: not commonly used due to several limitations.

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay

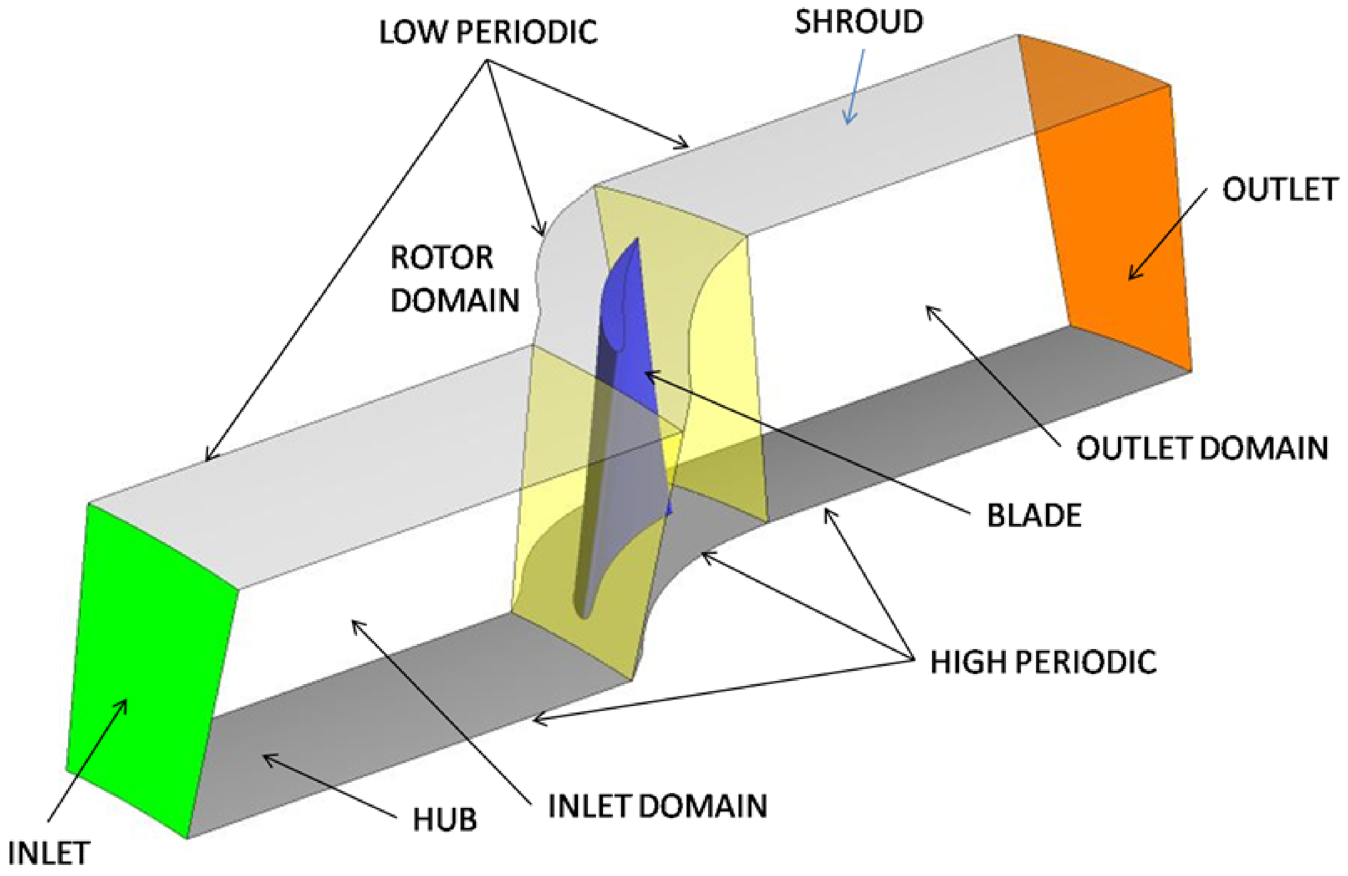
Boundary conditions

- Exit boundary conditions
  - Exit static pressure to achieve the required mass flow
  - It is also possible to specify a static pressure distribution at the exit domain.
  - Alternatively, mass flow can be directly specified at the exit.
  - For incompressible flows, using either of the two does not affect the results.
  - However, for compressible flows, static pressure outlet condition yields better results.

Boundary conditions

- For single passage simulations, periodic boundary conditions are used for simulating the effect of a blade row.
- The domain must be appropriately chosen to ensure that periodic boundary conditions are indeed valid.
- On surfaces (blade, hub and shroud), no-slip and adiabatic conditions are usually used.
- In turbines with hot gases present, the adiabatic condition may be replaced by constant heat flux condition.

Boundary conditions



Typical flow domain with the boundaries

In this lecture...

- Computational Fluid Dynamics for turbomachinery
  - Introduction and overview
  - Grid generation
  - Boundary conditions

Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay