Lecture 6 - Boundary Conditions

Applied Computational Fluid Dynamics

Instructor: André Bakker

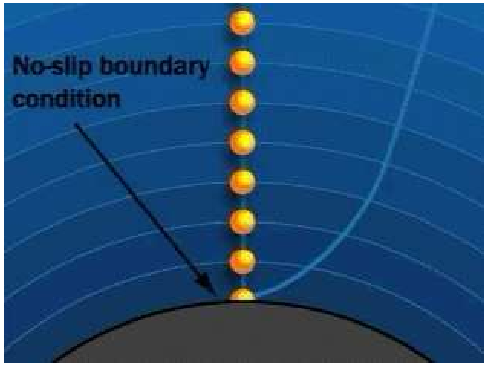
http://www.bakker.org

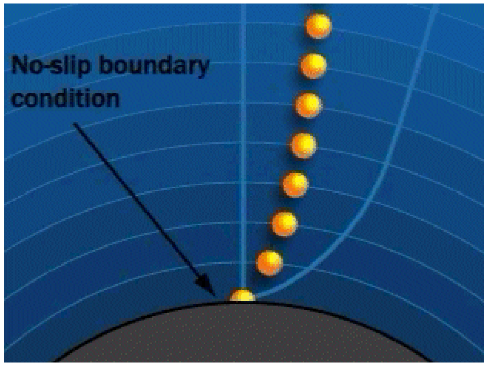
Outline

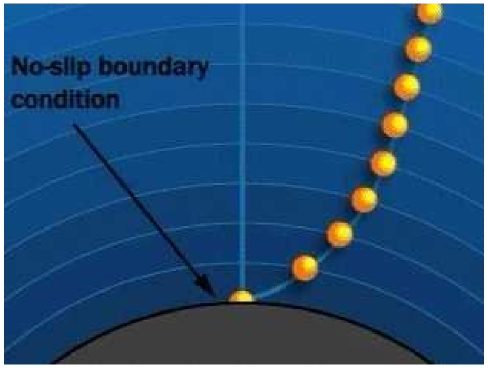
- Overview.
- Inlet and outlet boundaries.
  - Velocity.
  - Pressure boundaries and others.
- Wall, symmetry, periodic and axis boundaries.
- Internal cell zones.
  - Fluid, porous media, moving cell zones.
  - Solid.
- Internal face boundaries.
- Material properties.
- Proper specification.

Boundary conditions

- When solving the Navier-Stokes equation and continuity equation, appropriate initial conditions and boundary conditions need to be applied.
- In the example here, a no-slip boundary condition is applied at the solid wall.
- Boundary conditions will be treated in more detail in this lecture.

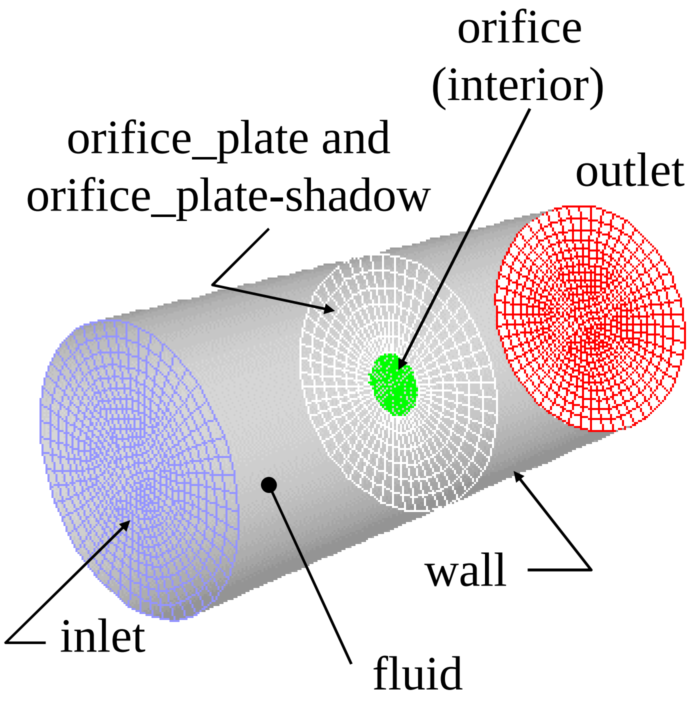






Overview

- Boundary conditions are a required component of the mathematical model.
- Boundaries direct motion of flow.
- Specify fluxes into the computational domain, e.g. mass, momentum, and energy.
- Fluid and solid regions are represented by cell zones.
- Material and source terms are assigned to cell zones.
- Boundaries and internal surfaces are represented by face zones.
- Boundary data are assigned to face zones.



Example: face and cell zones associated with pipe flow through orifice plate

Neumann and Dirichlet boundary conditions

- When using a Dirichlet boundary condition, one prescribes the value of a variable at the boundary, e.g. $u(x) = constant$.
- When using a Neumann boundary condition, one prescribes the gradient normal to the boundary of a variable at the boundary, e.g. $\partial_n u(x) = constant$.
- When using a mixed boundary condition a function of the form $au(x)+b\partial_n u(x) = constant$ is applied.
- Note that at a given boundary, different types of boundary conditions can be used for different variables.

Flow inlets and outlets

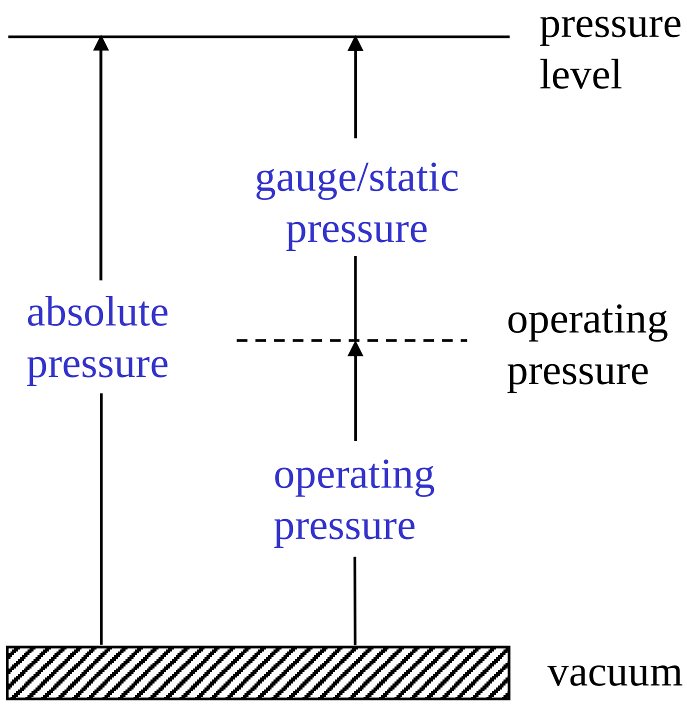
- A wide range of boundary conditions types permit the flow to enter and exit the solution domain:
  - General: pressure inlet, pressure outlet.
  - Incompressible flow: velocity inlet, outflow.
  - Compressible flows: mass flow inlet, pressure far-field.
  - Special: inlet vent, outlet vent, intake fan, exhaust fan.
- Boundary data required depends on physical models selected.
- General guidelines:
  - Select boundary location and shape such that flow either goes in or out. Not mandatory, but will typically result in better convergence.
  - Should not observe large gradients in direction normal to boundary near inlets and outlets. This indicates an incorrect problem specification.
  - Minimize grid skewness near boundary.

Pressure boundary conditions

- Pressure boundary conditions require static gauge pressure inputs:

$$p_{absolute} = p_{static} + p_{operating}$$

- The operating pressure input is set separately.
- Useful when:
  - Neither the flow rate nor the velocity are known (e.g. buoyancy-driven flows).
  - A "free" boundary in an external or unconfined flow needs to be defined.


pressure level

gauge/static pressure

absolute pressure

operating pressure

operating pressure

vacuum

Pressure inlet boundary (1)

- One defines the total gauge pressure, temperature, and other scalar quantities at flow inlets:

$$p_{total} = p_{static} + \frac{1}{2}\rho v^2 \qquad \text{incompressible flows}$$

$$p_{total} = p_{static}(1 + \frac{k-1}{2}M^2)^{k/(k-1)} \qquad \text{compressible flows}$$

- Here k is the ratio of specific heats (c_p/c_v) and M is the Mach number. If the inlet flow is supersonic you should also specify the static pressure.
- Suitable for compressible and incompressible flows. Mass flux through boundary varies depending on interior solution and specified flow direction.
- The flow direction must be defined and one can get non-physical results if no reasonable direction is specified.
- Outflow can occur at pressure inlet boundaries. In that case the flow direction is taken from the interior solution.

Pressure inlet boundary (2)

- For non-isothermal incompressible flows, one specifies the inlet temperature.
- For compressible flows, one specifies the total temperature T_0, which is defined as the temperature that the flow would have if it were brought to a standstill under isentropic conditions:

$$T_0 = T_s\left[1+\frac{k-1}{2}M^2\right]$$

- Here k is the ratio of specific heats (c_p/c_v), M is the Mach number, and T_s is the static temperature.

Pressure outlet boundary

- Here one defines the static/gauge pressure at the outlet boundary. This is interpreted as the static pressure of the environment into which the flow exhausts.
- Usually the static pressure is assumed to be constant over the outlet. A radial equilibrium pressure distribution option is available for cases where that is not appropriate, e.g. for strongly swirling flows.
- Backflow can occur at pressure outlet boundaries:
  - During solution process or as part of solution.
  - Backflow is assumed to be normal to the boundary.
  - Convergence difficulties minimized by realistic values for backflow quantities.
  - Value specified for static pressure used as total pressure wherever backflow occurs.
- Pressure outlet must always be used when model is set up with a pressure inlet.

Velocity inlets

- Defines velocity vector and scalar properties of flow at inlet boundaries.
- Useful when velocity profile is known at inlet. Uniform profile is default but other profiles can be implemented too.
- Intended for incompressible flows. The total (stagnation) properties of flow are not fixed. Stagnation properties vary to accommodate prescribed velocity distribution. Using in compressible flows can lead to non-physical results.
- Avoid placing a velocity inlet too close to a solid obstruction. This can force the solution to be non-physical.

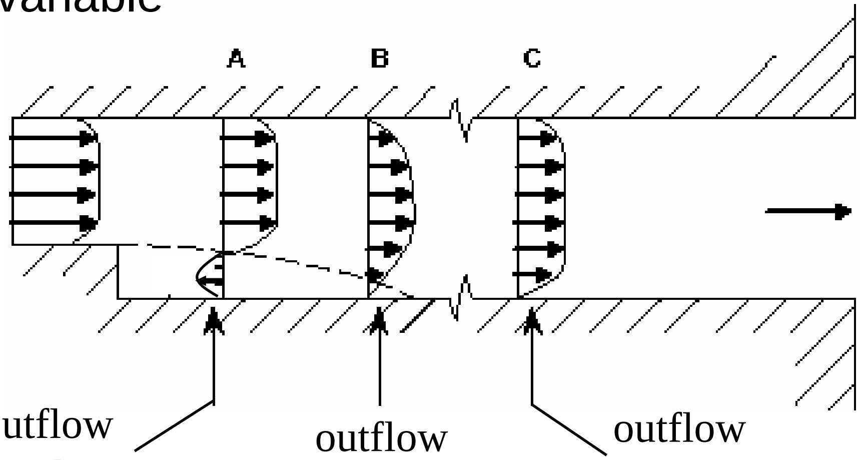
Outflow boundary

- Outflow boundary conditions are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem.
- Appropriate where the exit flow is close to a fully developed condition, as the outflow boundary condition assumes a zero normal gradient for all flow variables except pressure. The solver extrapolates the required information from interior.
- Furthermore, an overall mass balance correction is applied.

Restrictions on outflow boundaries

- Outflow boundaries cannot be used:
  - With compressible flows.
  - With the pressure inlet boundary condition (use velocity inlet instead) because the combination does not uniquely set a pressure gradient over the whole domain.
  - In unsteady flows with variable density.

- Do not use outflow boundaries where:
  - Flow enters domain or when backflow occurs (in that case use pressure b.c.).
  - Gradients in flow direction are significant.
  - Conditions downstream of exit plane impact flow in domain.



outflow condition ill-posed

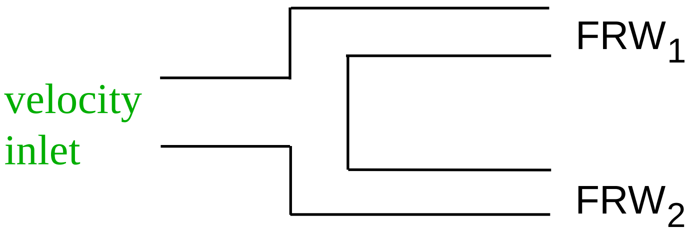
outflow condition not obeyed

outflow condition obeyed

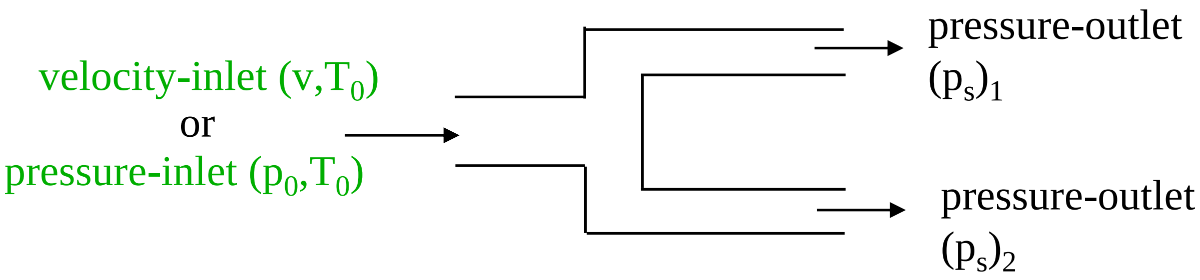
outflow condition closely obeyed

Modeling multiple exits

- Using outflow boundary condition:
  - Mass flow divided equally among all outflow boundaries by default.
  - Flow rate weighting (FRW) set to one by default.
  - For uneven flow distribution one can specify the flow rate weighting for each outflow boundary: $m_i=FRW_i/\Sigma FRW_i$. The static pressure then varies among the exits to accommodate this flow distribution.



- Can also use pressure outlet boundaries to define exits.



Other inlet and outlet boundary conditions

- Mass flow inlet.
  - Used in compressible flows to prescribe mass flow rate at inlet.
  - Not required for incompressible flows.
- Pressure far field.
  - Available when density is calculated from the ideal gas law.
  - Used to model free-stream compressible flow at infinity, with free-stream Mach number and static conditions specified.
- Exhaust fan/outlet vent.
  - Model external exhaust fan/outlet vent with specified pressure jump/loss coefficient and ambient (discharge) pressure and temperature.
- Inlet vent/intake fan.
  - Model inlet vent/external intake fan with specified loss coefficient/ pressure jump, flow direction, and ambient (inlet) pressure and temperature.

Determining turbulence parameters

- When turbulent flow enters domain at inlet, outlet, or at a far-field boundary, boundary values are required for:
  - Turbulent kinetic energy *k.*
  - Turbulence dissipation rate ε.
- Four methods available for specifying turbulence parameters:
  - Set *k* and ε explicitly.
  - Set turbulence intensity and turbulence length scale.
  - Set turbulence intensity and turbulent viscosity ratio.
  - Set turbulence intensity and hydraulic diameter.

Turbulence intensity

- The turbulence intensity I is defined as:

$$I = \frac{\sqrt{\frac{2}{3}k}}{u}$$

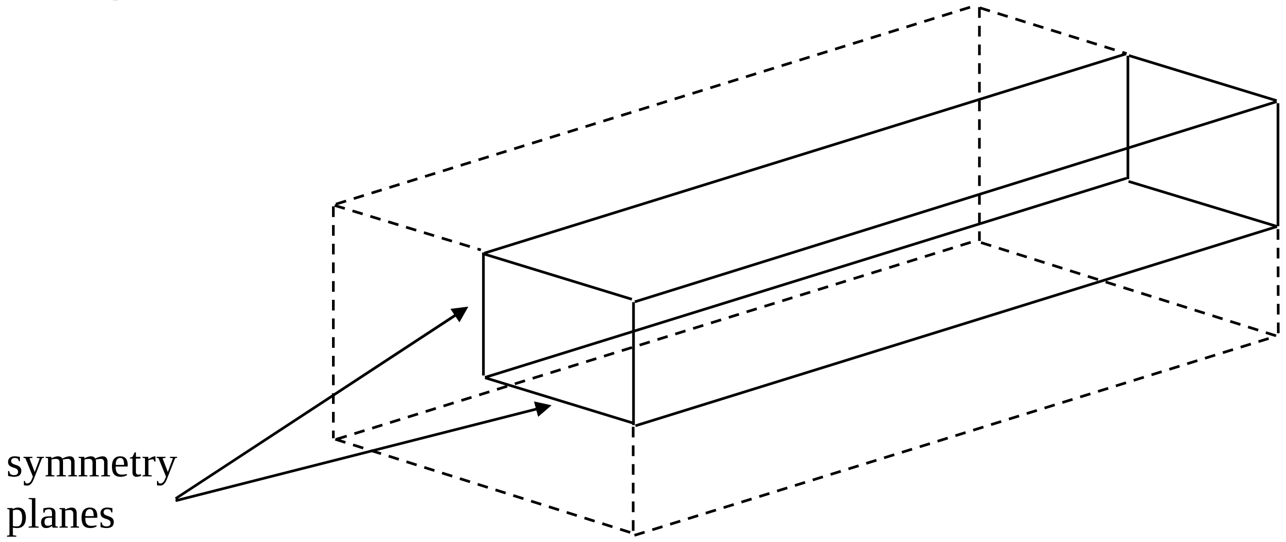
- Here *k* is the turbulent kinetic energy and u is the local velocity magnitude.
- Intensity and length scale depend on conditions upstream:
  - Exhaust of a turbine.
    Intensity = 20%. Length scale = 1 - 10 % of blade span.
  - Downstream of perforated plate or screen.
    Intensity = 10%. Length scale = screen/hole size.
  - Fully-developed flow in a duct or pipe.
    Intensity = 5%. Length scale = hydraulic diameter.

Wall boundaries

- Used to bound fluid and solid regions.
- In viscous flows, no-slip condition enforced at walls.
  - Tangential fluid velocity equal to wall velocity.
  - Normal velocity component is set to be zero.
- Alternatively one can specify the shear stress.
- Thermal boundary condition.
  - Several types available.
  - Wall material and thickness can be defined for 1-D or in-plane thin plate heat transfer calculations.
- Wall roughness can be defined for turbulent flows.
  - Wall shear stress and heat transfer based on local flow field.
- Translational or rotational velocity can be assigned to wall.

Symmetry boundaries

- Used to reduce computational effort in problem.
- Flow field and geometry must be symmetric:
  - Zero normal velocity at symmetry plane.
  - Zero normal gradients of all variables at symmetry plane.
- No inputs required.
  - Must take care to correctly define symmetry boundary locations.
- Also used to model slip walls in viscous flow.

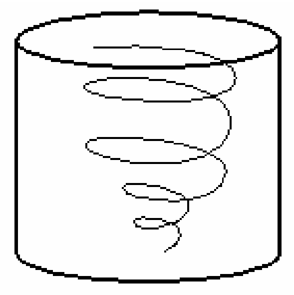


Periodic boundaries

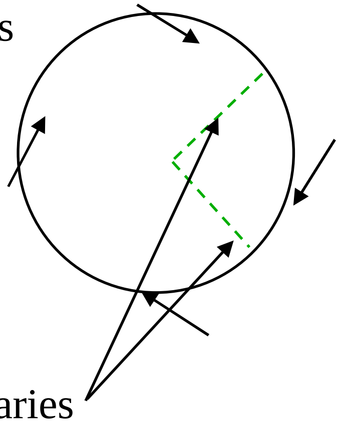
- Used when physical geometry of interest and expected flow pattern and the thermal solution are of a periodically repeating nature.
  - Reduces computational effort in problem.
- Two types available:
  - $\Delta p = 0$ across periodic planes.
    - Rotationally or translationally periodic.
    - Rotationally periodic boundaries require axis of rotation be defined in fluid zone.
  - Δp is finite across periodic planes.
    - Translationally periodic only.
    - Models fully developed conditions.
    - Specify either mean Δp per period or net mass flow rate.

Periodic boundaries: examples

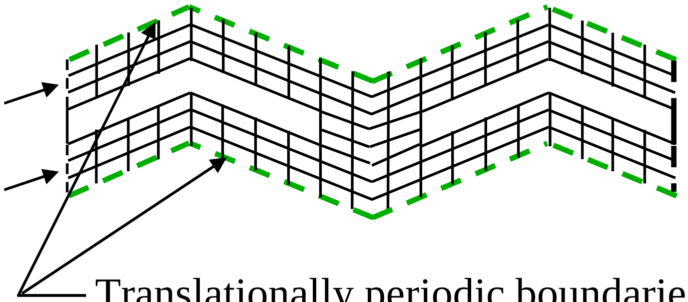
- **Δp = 0:**



4 tangential inlets



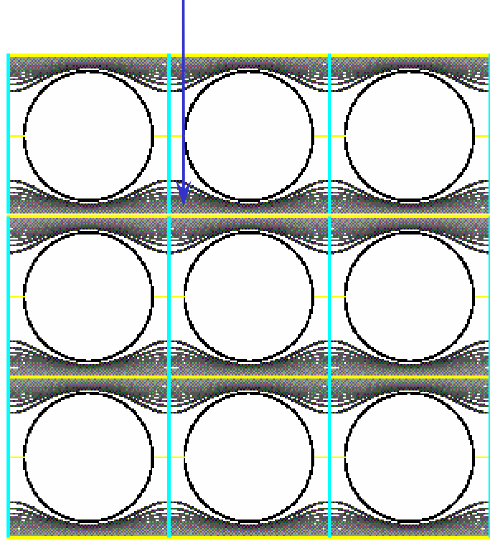
Rotationally periodic boundaries



Translationally periodic boundaries

- **Δp > 0:**

computational domain

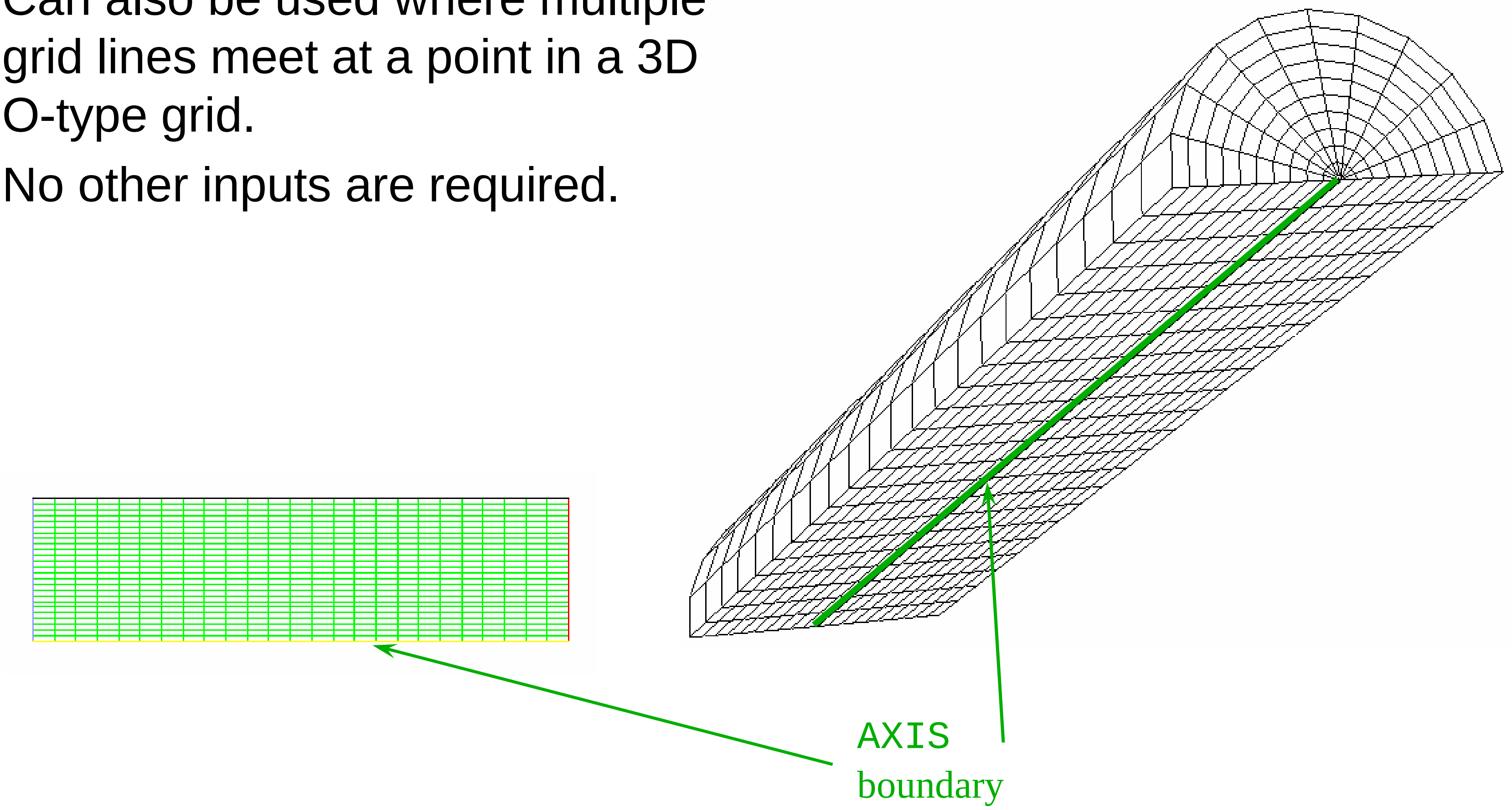


flow direction

Streamlines in a 2D tube heat exchanger

Axis boundaries

- Used at the centerline (y=0) of a 2-D axisymmetric grid.
- Can also be used where multiple grid lines meet at a point in a 3D O-type grid.
- No other inputs are required.

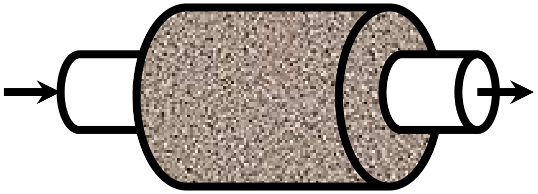


Cell zones: fluid

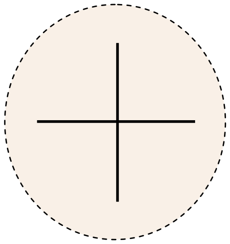
- A fluid zone is the group of cells for which all active equations are solved.
- Fluid material input required.
  - Single species, phase.
- Optional inputs allow setting of source terms:
  - Mass, momentum, energy, etc.
- Define fluid zone as laminar flow region if modeling transitional flow.
- Can define zone as porous media.
- Define axis of rotation for rotationally periodic flows.
- Can define motion for fluid zone.

Porous media conditions

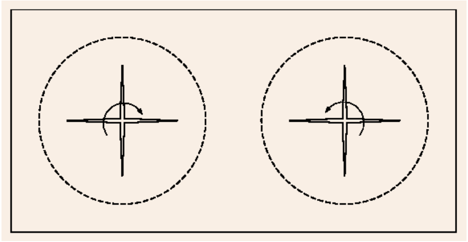
- Porous zone modeled as special type of fluid zone.
  - Enable the porous zone option in the fluid boundary conditions panel.
  - Pressure loss in flow determined via user inputs of resistance coefficients to lumped parameter model.
- Used to model flow through porous media and other “distributed” resistances, e.g:
  - Packed beds.
  - Filter papers.
  - Perforated plates.
  - Flow distributors.
  - Tube banks.



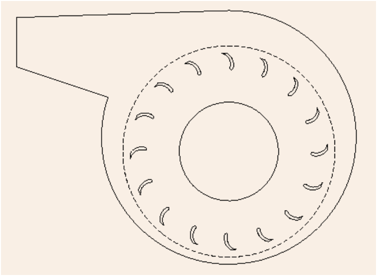
Moving zones



- For single zone problems use the rotating reference frame model. Define the whole zone as moving reference frame. This has limited applicability.
- For multiple zone problems each zone can be specified as having a moving reference frame:
  - Multiple reference frame model. Least accurate, least demanding on CPU.
  - Mixing plane model. Field data are averaged at the outlet of one zone and used as inlet boundary data to adjacent zone.



- Or each zone can be defined as moving mesh using the sliding mesh model. Must then also define interface. Mesh positions are calculated as a function of time. Relative motion must be tangential (no normal translation).



Cell zones: solid

- A solid zone is a group of cells for which only heat conduction is solved and no flow equations are solved.
- The material being treated as solid may actually be fluid, but it is assumed that no convection takes place.
- The only required input is material type so that appropriate material properties are being used.
- Optional inputs allow you to set a volumetric heat generation rate (heat source).
- Need to specify rotation axis if rotationally periodic boundaries adjacent to solid zone.
- Can define motion for solid zone.

Internal face boundaries

- Defined on cell faces.
  - Do not have finite thickness.
  - Provide means of introducing step change in flow properties.
- Used to implement physical models representing:
  - Fans.
  - Radiators.
  - Porous jumps.
  - Interior walls. In that case also called “thin walls.”

Material properties

- For each zone, a material needs to be specified.
- For the material, relevant properties need to be specified:
  - Density.
  - Viscosity, may be non-Newtonian.
  - Heat capacity.
  - Molecular weight.
  - Thermal conductivity.
  - Diffusion coefficients.
- Which properties need to be specified depends on the model. Not all properties are always required.
- For mixtures, properties may have to be specified as a function of the mixture composition.

Fluid density

- For constant density, incompressible flow: ρ = constant.
- For compressible flow: $\rho = p_{absolute}/RT$.
- Density can also be defined as a function of temperature (polynomial, piece-wise polynomial, or the Boussinesq model where ρ is considered constant except for the buoyancy term in the momentum equations) or be defined with user specified functions.
- For incompressible flows where density is a function of temperature one can also use the so-called incompressible-ideal-gas law: $\rho = p_{operating}/RT$.
- Generally speaking, one should set $p_{operating}$ close to the mean pressure in the domain to avoid round-off errors.
- However, for high Mach number flows using the coupled solver, set $p_{operating}$ to zero.

When is a problem properly specified?

- Proper specification of boundary conditions is very important.
- Incorrect boundary conditions will lead to incorrect results.
- Boundary conditions may be overspecified or underspecified.
- Overspecification occurs when more boundary conditions are specified than appropriate and not all conditions can hold at the same time.
- Underspecification occurs when the problem is incompletely specified, e.g. there are boundaries for which no condition is specified.
- Commercially available CFD codes will usually perform a number of checks on the boundary condition set-up to prevent obvious errors from occurring.

Summary

- Zones are used to assign boundary conditions.
- Wide range of boundary conditions permit flow to enter and exit solution domain.
- Wall boundary conditions used to bound fluid and solid regions.
- Repeating boundaries used to reduce computational effort.
- Internal cell zones used to specify fluid, solid, and porous regions.
- Internal face boundaries provide way to introduce step change in flow properties.