Customer Training Material

Lecture 4

Cell Zones and Boundary Conditions

Introduction to ANSYS FLUENT

© 2010 ANSYS, Inc. All rights reserved.

Defining Cell Zones and Boundary Conditions

To properly define any CFD problem, you must define:

- Cell zones
  - These relate to the middle of the grid cells
  - Typically this always involves saying which material (fluid) is in that cell
  - Other values may also apply (porous resistance, heat sources etc)

- Boundary conditions
  - Where fluid enters or leaves the domain, the conditions must be set (velocity / pressure / temperature)
  - Other boundaries also need declaring, like walls (smooth/rough, heat transfer?)
  - There may also be symmetry, periodic or axis boundaries.

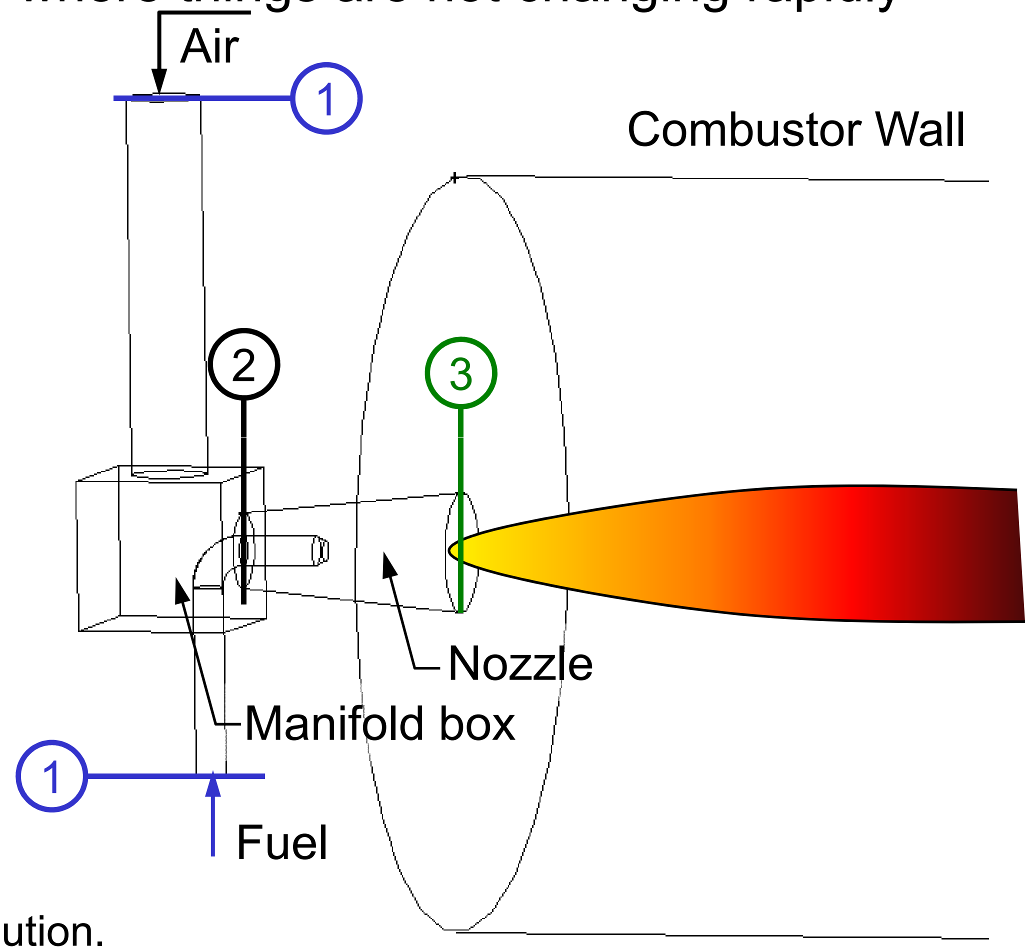
- The data required at a boundary depends upon the boundary condition type and the physical models employed.

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Locating Boundaries – An Example

- When planning a model, it is important to decide where to place the boundaries. The best answer is usually a combination of where you know exactly what the conditions are, or where things are not changing rapidly

1. Upstream of manifold
   - Can use uniform profile.
   - Properly accounts for mixing.
   - Non-premixed reaction models
   - Requires more cells.
2. Nozzle inlet plane
   - Non-premixed reaction models
   - Requires accurate inlet profile
   - Flow is still non-premixed.
3. Nozzle outlet plane
   - Premixed reaction model.
   - Requires accurate profile.
   - Not generally recommended since inlet BCs may drive the interior solution.



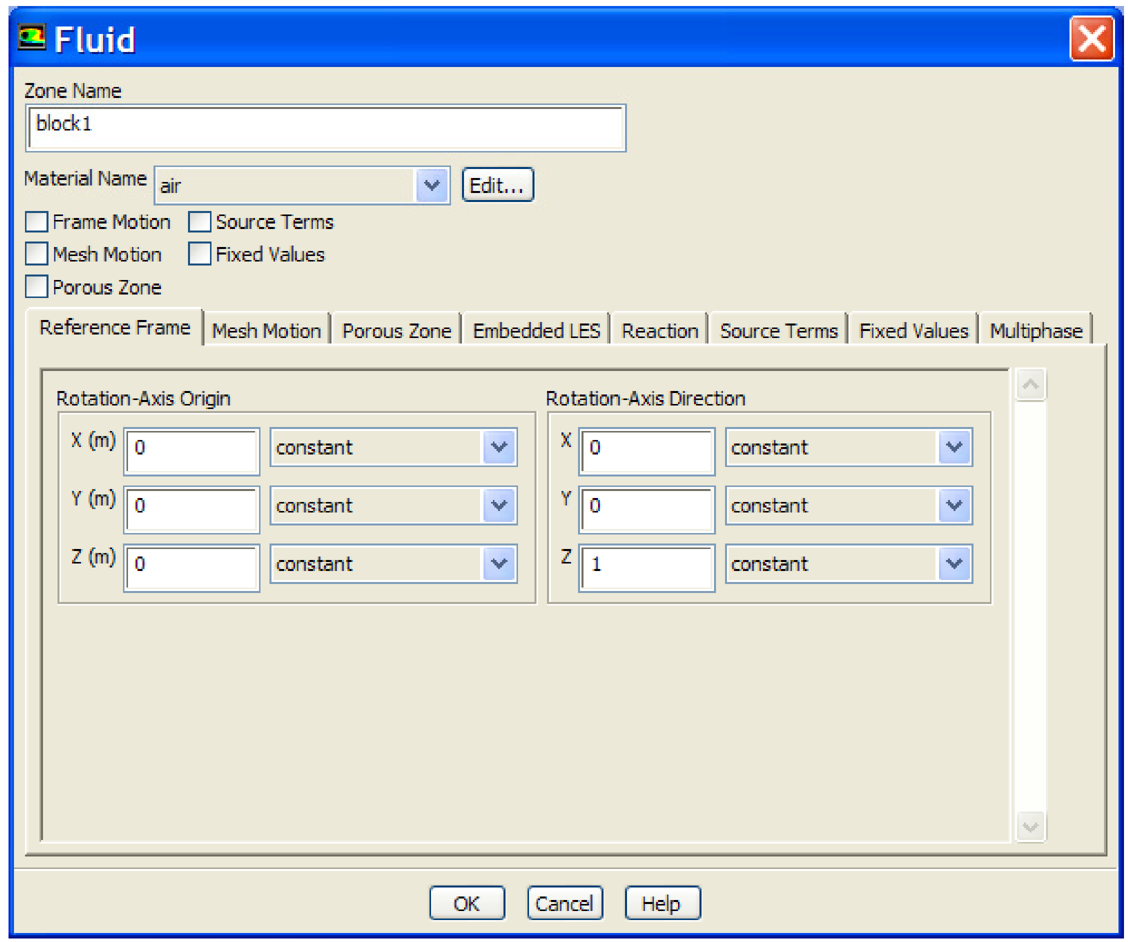
© 2010 ANSYS, Inc. All rights reserved.

Cell Zones – Fluid

- A fluid cell zone is a group of cells for which all active equations are solved.

- The material in the cell zone must be declared.
  - Although if using multi-species or multiphase, the composition is set elsewhere.

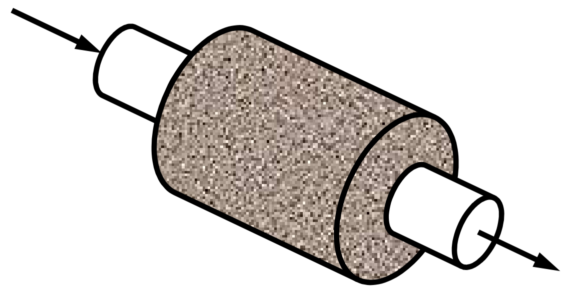
- Optional inputs
  - Moving zones
  - Porous region
  - Source terms
  - Fixed Values



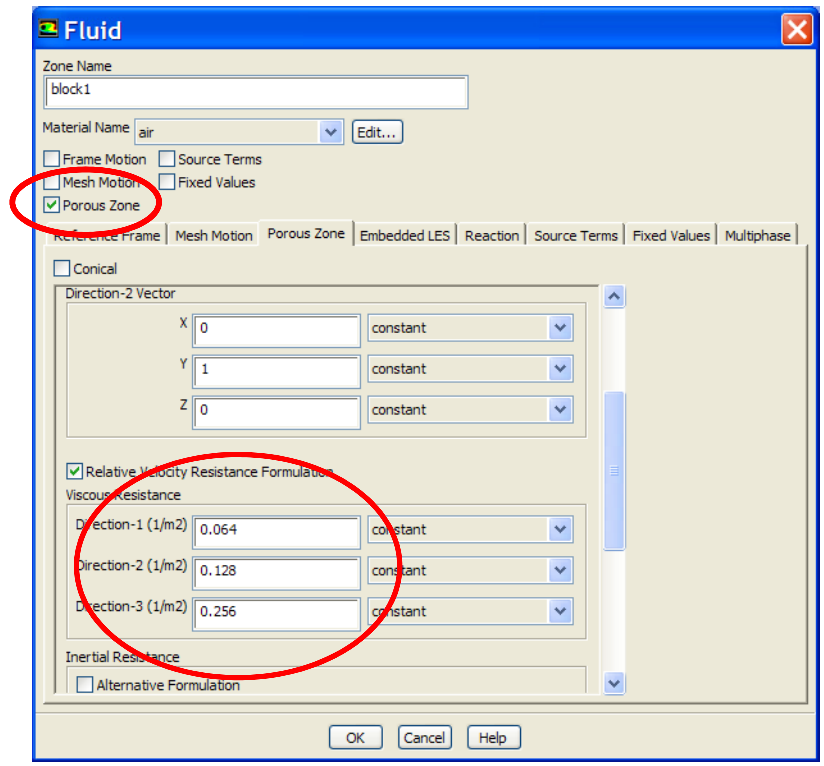
© 2010 ANSYS, Inc. All rights reserved.

Cell Zones - Porous Media

- Some fluid regions are obviously porous and impossible to resolve exactly in a mesh:
  - Filter papers
  - Packed beds



- In other problems, there are regions where it is impractical to resolve all the detail because of cell count yet the effect of these objects must be accounted for:
  - External airflow flow past small pipes or other steelwork
  - Effects of trees

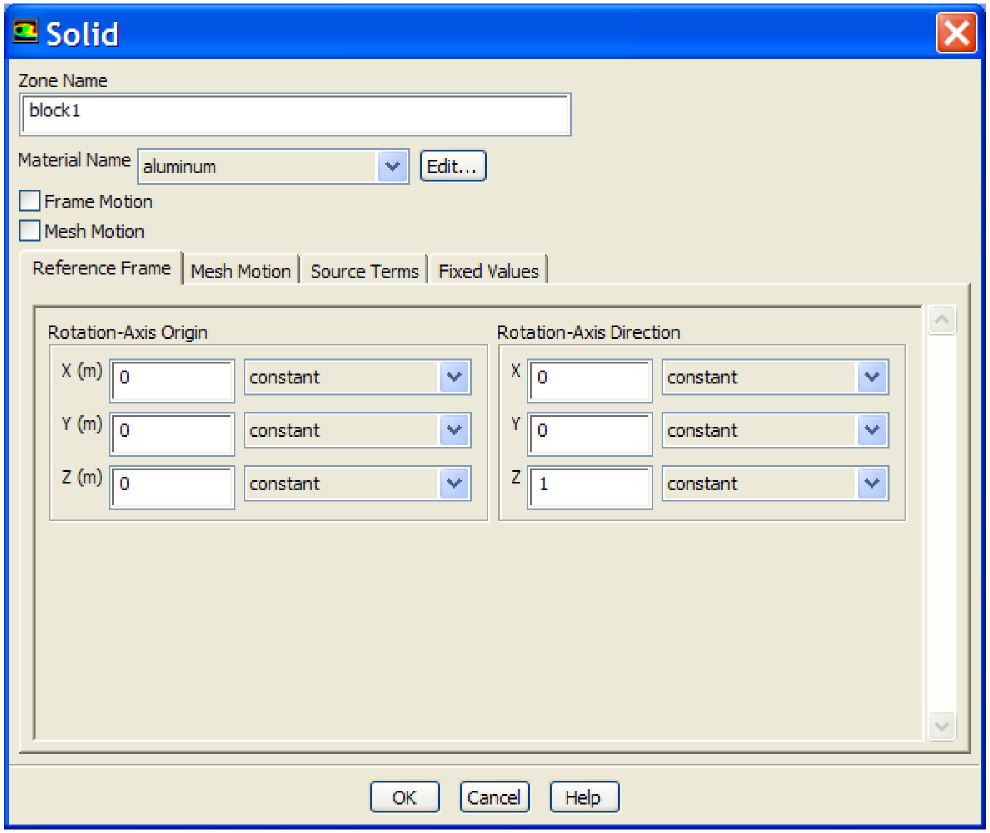


- Inputs are directional viscous and inertial resistance coefficients.

© 2010 ANSYS, Inc. All rights reserved.

Cell Zones – Solid

- A solid zone is a group of cells for which only the energy equation is solved.
- Only required input is the material name (defined in the Materials panel).
- Optional inputs allow you to set volumetric heat generation rate (heat source).
- Need to specify rotation axis if rotationally periodic boundaries adjacent to solid zone.
- Can define motion for a solid zone

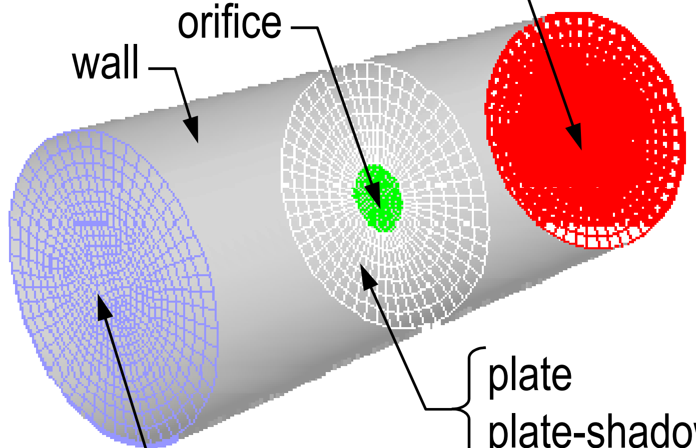


ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Available Types

- **External Boundaries**
  - General
    - Pressure Inlet
    - Pressure Outlet
  - Incompressible
    - Velocity Inlet
    - Outflow (not recommended)
  - Compressible
    - Mass Flow Inlet
    - Pressure Far Field
  - Other
    - Wall
    - Symmetry
    - Axis
    - Periodic
  - Special
    - Inlet / Outlet Vent
    - Intake / Exhaust Fan

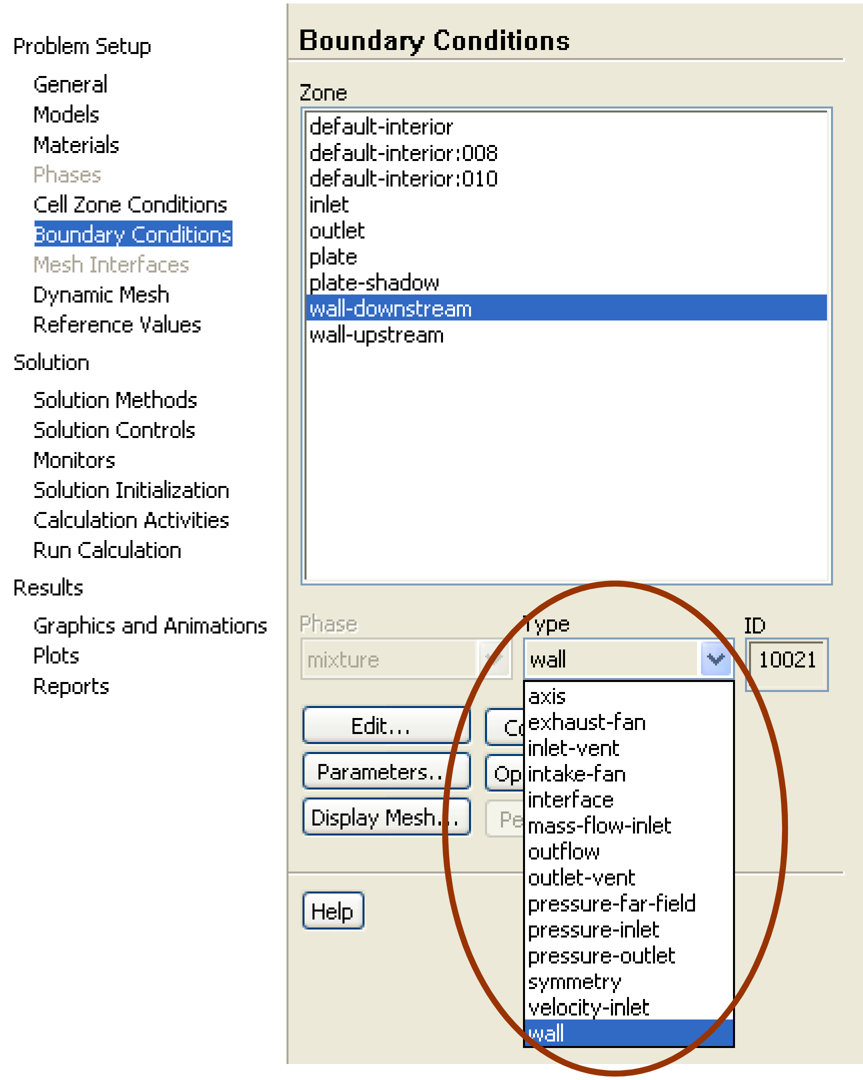
- **Internal Boundaries**
  - Fan
  - Interior
  - Porous Jump
  - Radiator
  - Wall
- Cell (Continuum) zones
  - Fluid
  - Solid
  - Porous media



ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Changing Types

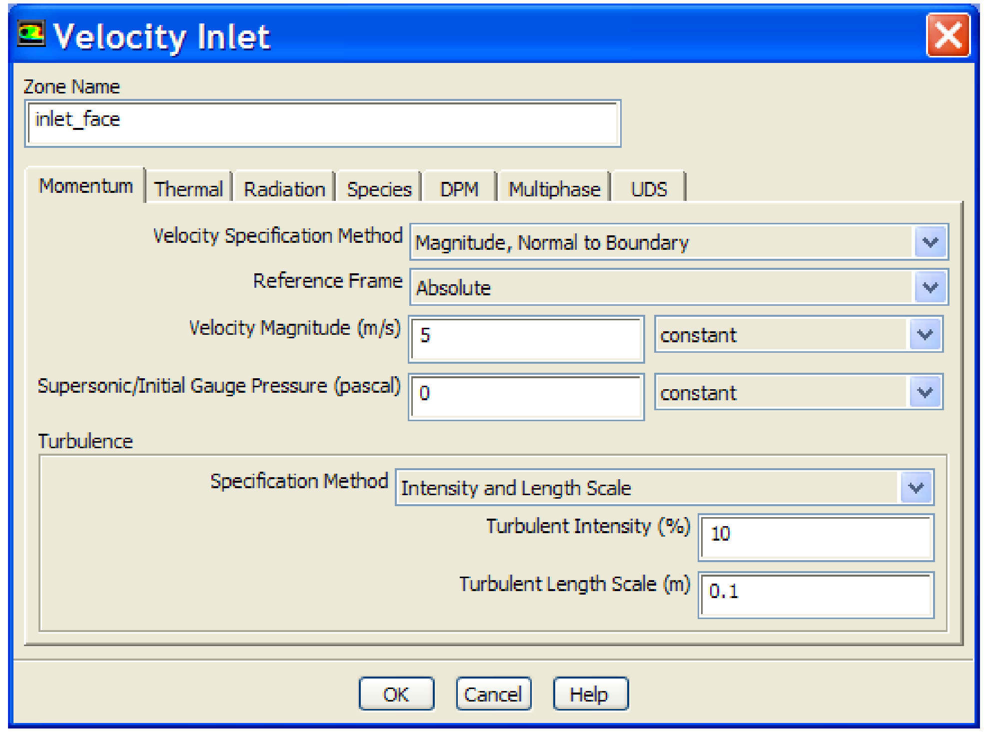
- Zones and zone types are initially defined in the preprocessing phase.
- To change the boundary condition type for a zone:
  - Choose the zone name in the Zone list.
  - Select the type you wish to change it to in the Type pull-down list.



© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Velocity Inlet

- Velocity Specification Method
  - Magnitude, Normal to Boundary
  - Components
  - Magnitude and Direction



- Applies a uniform velocity profile at the boundary, unless UDF or profile is used.

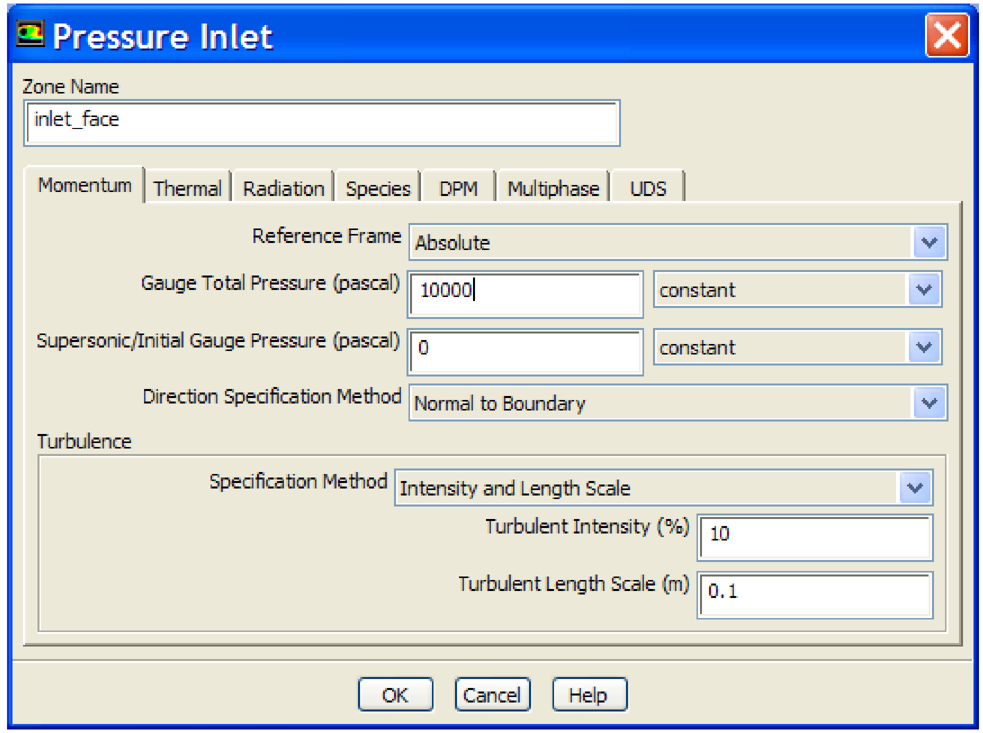
- Velocity inlets are intended for use in incompressible flows and are not recommended for compressible flows.

- Velocity Magnitude input can be negative, implying that you can prescribe the exit velocity.

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Pressure Inlet

- Pressure inlets are suitable for both compressible and incompressible flows.
  - Pressure inlet boundary is treated as a loss-free transition from stagnation to inlet conditions.
  - FLUENT calculates static pressure and velocity at inlet
  - Mass flux through boundary varies depending on the interior solution and specified flow direction.
- Required inputs
  - Gauge Total Pressure
  - Supersonic / Initial Gauge Pressure
  - Inlet flow direction
  - Turbulence quantities (if applicable)
  - Total temperature (if heat transfer and/or compressible).



Incompressible: $$p_{\text{total}} = p_{\text{static}} + \frac{\rho V^2}{2}$$

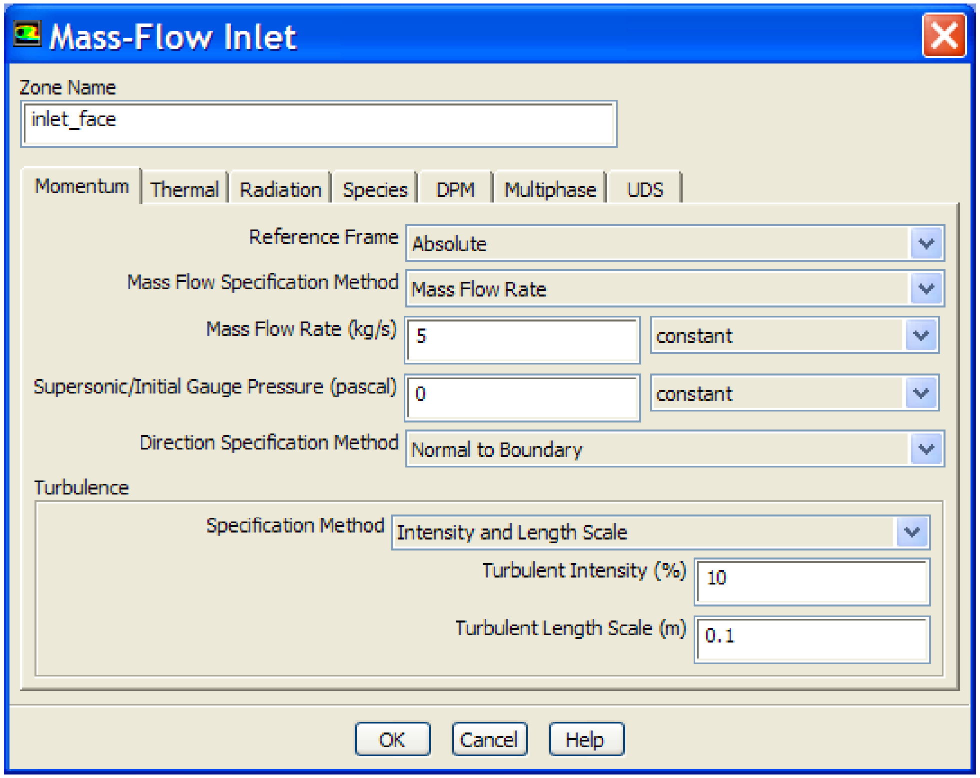
Compressible: $$p_{\text{total,abs}} = p_{\text{static,abs}} \left(1 + \frac{k-1}{2} M^2\right)^{\frac{k}{k-1}}$$

$$T_{\text{total,abs}} = T_{\text{static,abs}} \left(1 + \frac{k-1}{2} M^2\right)$$

© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Mass Flow Inlet

- Mass flow inlets are intended for compressible flows; however, they can be used for incompressible flows.
  - Total pressure adjusts to accommodate mass flow inputs.
  - More difficult to converge than pressure inlet.



- Required information
  - Mass Flow Rate or Mass Flux
  - Supersonic/Initial Gauge Pressure
    - Static pressure where flow is locally supersonic; ignored if subsonic
    - Will be used if flow field is initialized from this boundary.
  - Total Temperature (on Thermal tab)
    - Used as static temperature for incompressible flow.
  - Direction Specification Method

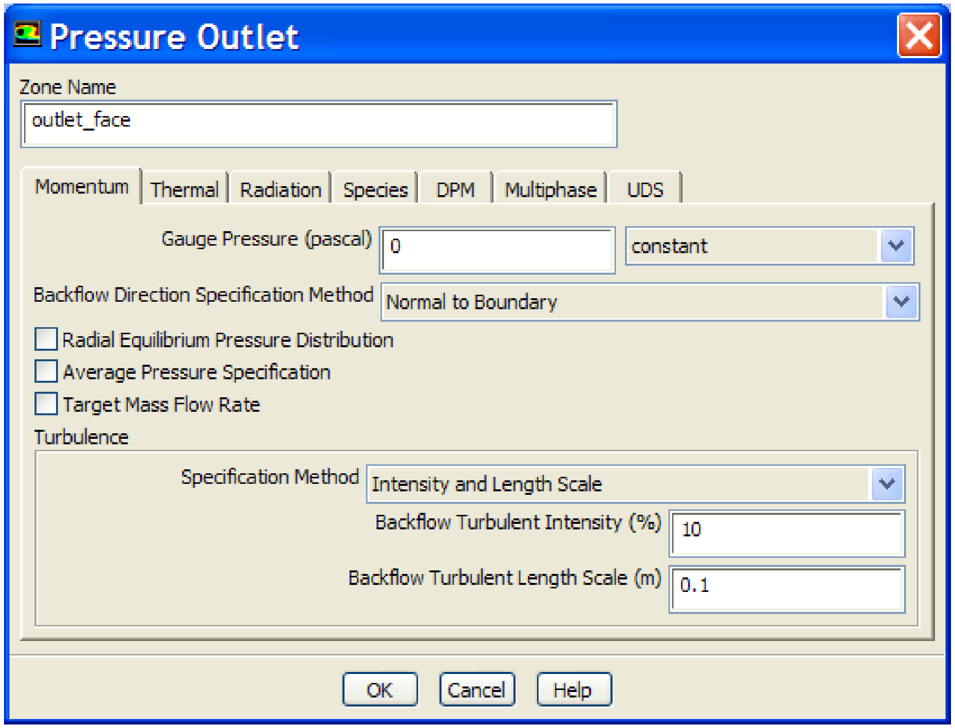
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Pressure Outlet

- Suitable for compressible and incompressible flows.
  - Specified pressure is ignored if flow is locally supersonic at the outlet.
  - Can be used as a “free” boundary in an external or unconfined flow.

- Required information
  - Gauge Pressure (static) – static pressure of the environment into which the flow exits.
  - Backflow quantities – Used as inlet conditions if/when backflow occurs (outlet acts like an inlet).

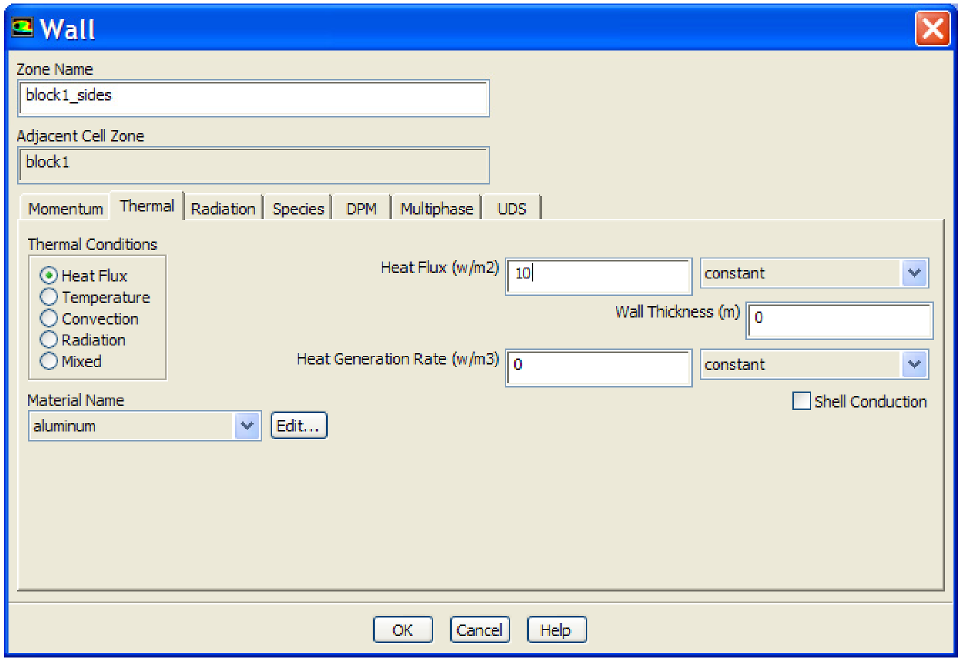
- For ideal gas (compressible) flow, non-reflecting outlet boundary conditions (NRBC) are available.



ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Wall Boundaries

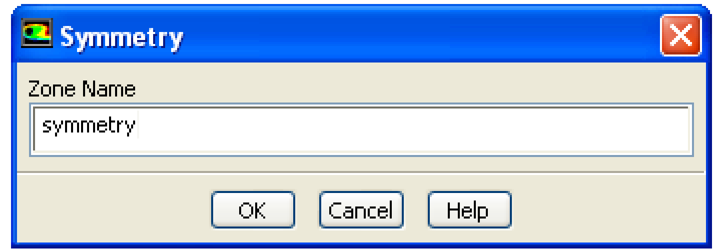
- In viscous flows, no-slip conditions are applied at walls.
  - Shear stress can be applied.
- Thermal boundary conditions
  - Several types of thermal BCs are available.
  - Wall material and thickness can be defined for 1D or shell conduction heat transfer calculations (details will be discussed in the Heat Transfer lecture).
- 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 boundaries.

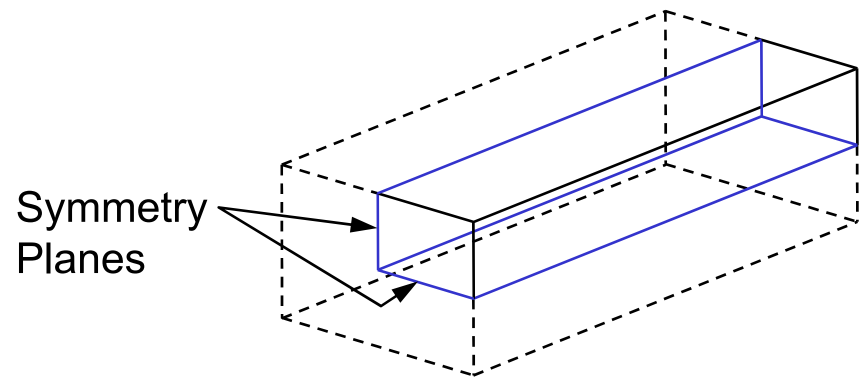


ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

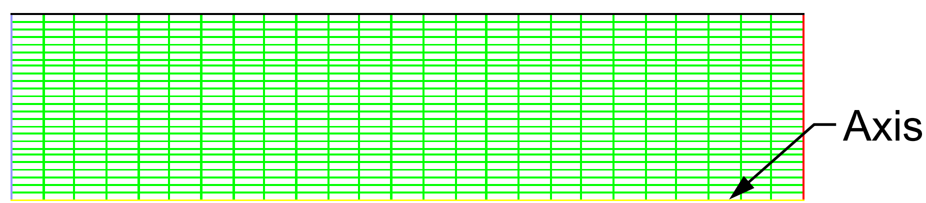
Boundary Conditions - Symmetry and Axis

- **Symmetry Boundary**
  - No inputs are required.
  - Flow field and geometry must be symmetric:
    - Zero normal velocity at symmetry plane
    - Zero normal gradients of all variables at symmetry plane
    - Must take care to correctly define symmetry boundary locations.





- **Axis Boundary**
  - Used at the center line for axisymmetric problems.
  - No user inputs required.
  - Must coincide with the positive x direction!



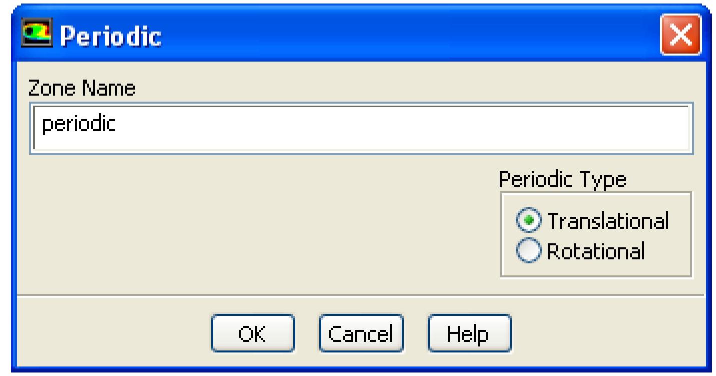
© 2010 ANSYS, Inc. All rights reserved.

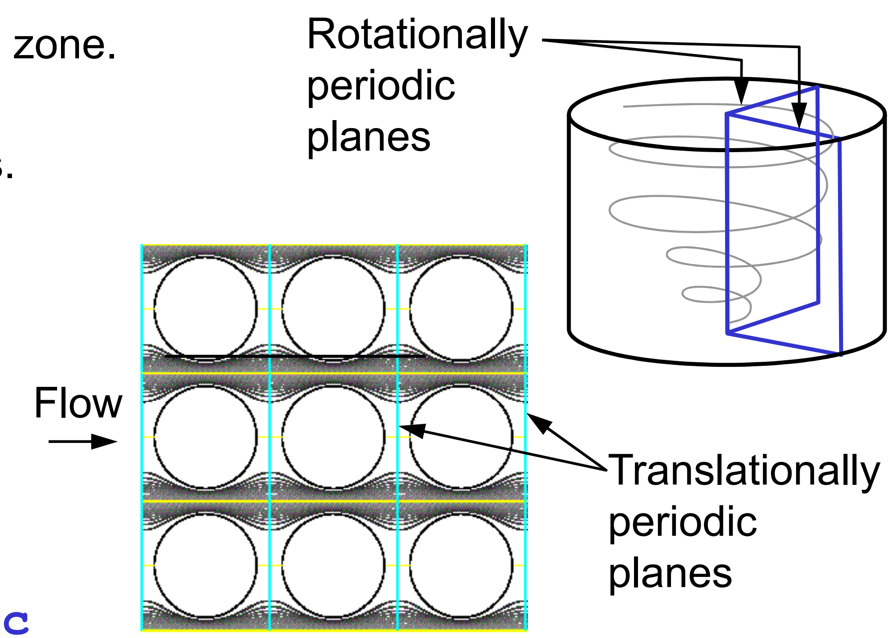
Boundary Conditions - Periodic Boundaries

- Used to reduce the overall mesh size.

- Flow field and geometry must contain either rotational or translational periodicity.
  - Rotational periodicity
    - ΔP = 0 across periodic planes.
    - Axis of rotation must be defined in fluid zone.
  - Translational periodicity
    - ΔP can be finite across periodic planes.
    - Models fully developed conditions.
    - Specify either mean ΔP per period or net mass flow rate.

- Periodic conditions can be defined (if not defined in the mesh) using the FLUENT TUI:
  `/mesh/modify-zones/make-periodic`





2D Tube Heat Exchanger

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Internal Faces

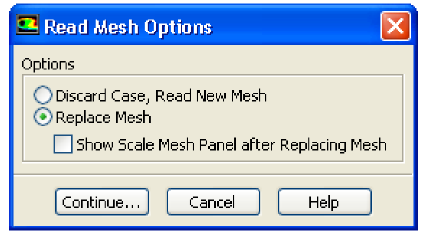
- Defined on the cell faces only:
  - Thickness of these internal faces is zero
  - These internal faces provide means of introducing step changes in flow properties.

- Used to implement various physical models including:
  - Fans
  - Radiators
  - Porous-jump models
    - Preferable over porous media for its better convergence behavior.
  - Interior walls

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Case Setup Replication

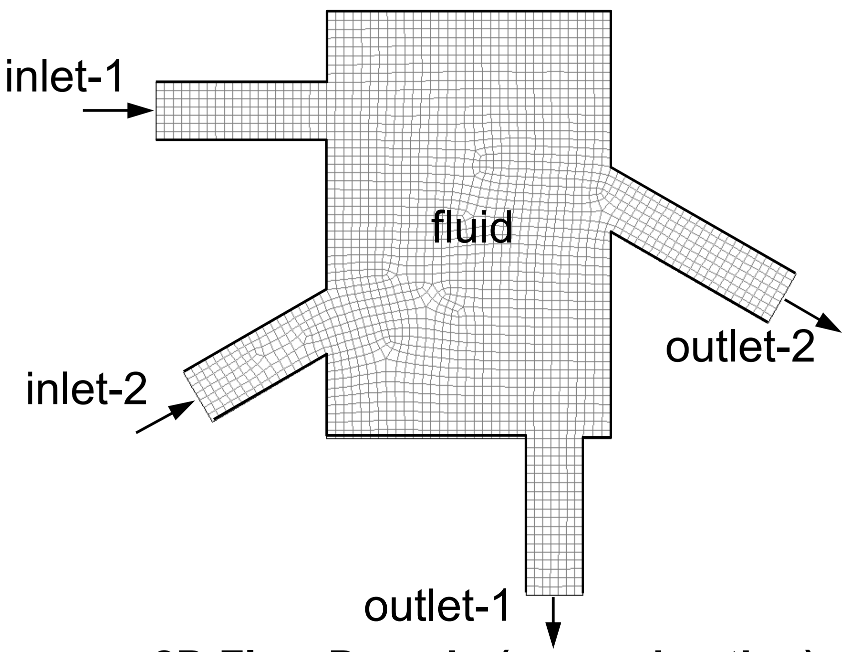
- To replicate a case setup:
  – Use the Read Mesh Options



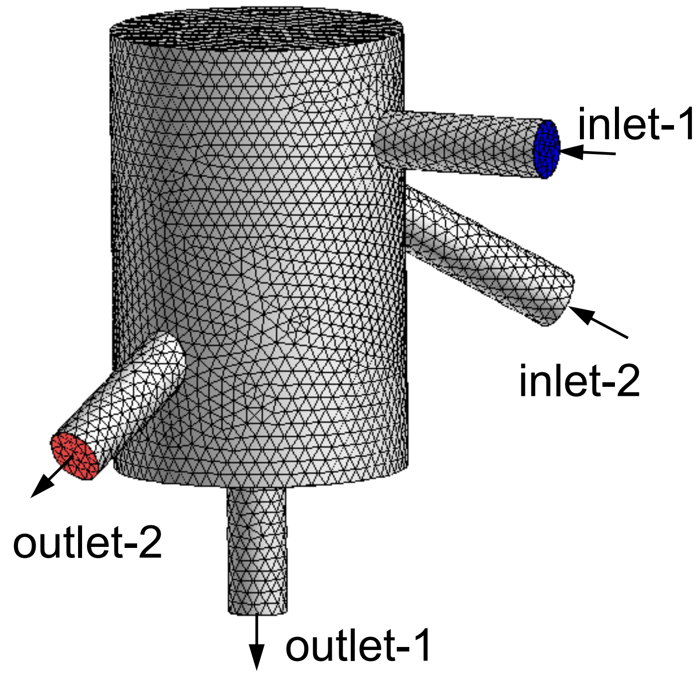
  – Or use the read/write boundary conditions feature via TUI command:

`/file/write-settings`	Creates a BC file
`/file/read-settings`	Reads an existing BC file

  – You can transfer settings from a 2D case to a 3D case!



2D Flow Domain (approximation)



Actual 3D Flow Domain

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Summary

- **Cell zones** are used to assign which fluid/solid material(s) exist in a region.
  - Options for porous media, laminar region, fixed value, etc.

- **Boundary zones** are used to control the solution at external and internal boundaries. Many different boundary types exist for prescribing boundary information.

- Computational effort can be reduced through use of **symmetry and periodic** boundaries.

- There are several other boundary condition types which were not presented (see appendix for more information about these).
  - Pressure Far Field
  - Exhaust Fan / Outlet Vent
  - Inlet Vent / Intake Fan
  - Outflow

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Appendix : Other Boundary Types

ANSYS, Inc. Proprietary
© 2009 ANSYS, Inc. All rights reserved.

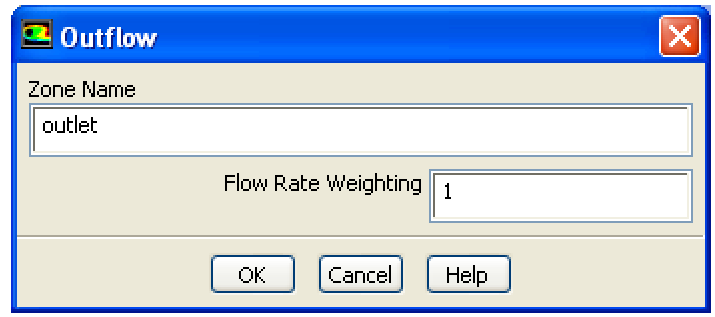
Other Inlet / Outlet Boundary Conditions

- **Pressure Far Field**
  - Used to model free-stream compressible flow at infinity, with prescribed static conditions and the free-stream Mach number.
  - Available only when density is calculated using the ideal gas law.
- **Target Mass Flow Rate option for pressure** outlets (not available for the multiphase models)
  - Provides the ability to fix the mass flow rate on a pressure outlet (either constant or via UDF hook)
  - Options to choose iteration method in TUI
- **Exhaust Fan / Outlet Vent**
  - Models an external exhaust fan or outlet vent with specified pressure rise / loss coefficient and ambient (discharge) pressure and temperature**.**
- **Inlet Vent / Intake Fan**
  - Models an inlet vent / external intake fan with specified loss coefficient / pressure rise, flow direction, and ambient (inlet) pressure and temperature.
- Inlet boundary conditions for **large-eddy / detached-eddy simulations** are covered in the Turbulence Modeling lecture.

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Boundary Conditions - Outflow

- No pressure or velocity information is required.
  - Data at exit plane is extrapolated from interior.
  - Mass balance correction is applied at boundary.

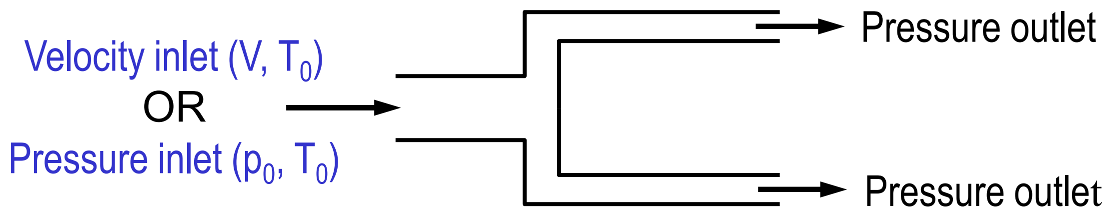


- Flow exiting outflow boundary exhibits zero normal diffusive flux for all flow variables.
  - Appropriate where the exit flow is fully developed.
- The outflow boundary is intended for use with incompressible flows.
  - Cannot be used with a pressure inlet boundary (must use velocity-inlet).
    - Combination does not uniquely set pressure gradient over whole domain.
  - Cannot be used for unsteady flows with variable density.
- Poor rate of convergence when backflow occurs during iterations.
  - Cannot be used if backflow is expected in the final solution.

ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Modeling Multiple Exits

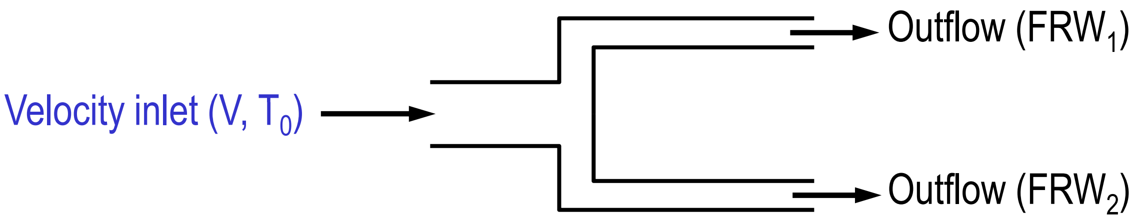
- Flows with multiple exits can be modeled using pressure outlet or outflow boundaries, depending on the information you know.
  - Pressure outlets – requires knowledge of downstream pressures; FLUENT calculates the fraction of total flow through each branch.



  - Outflow:
    - Mass flow rate fraction determined from Flow Rate Weighting (FRW) by

$$\frac{\dot{m}_i}{\dot{m}_{\text{total}}} = \frac{\text{FRW}_i}{\sum_i \text{FRW}_i}$$

    - Static pressure varies among exits to accommodate the prescribed flow distribution.



© 2010 ANSYS, Inc. All rights reserved.