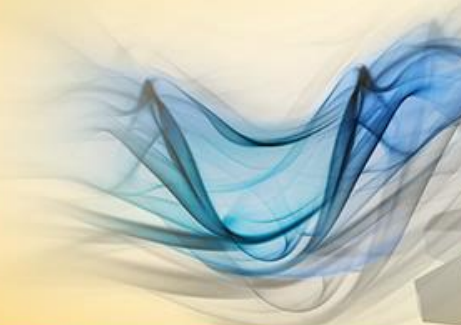


Lecture 3:

Cell Zone and Boundary Conditions

16.0 Release

A visualization of fluid flow, showing blue, wavy, translucent streamlines that represent the movement of a fluid. The flow appears to be coming from the left and moving towards the right.

Fluid Dynamics

A 3D model of a mechanical gear. The gear is purple with a bright white and yellow light source in its center, creating a lens flare effect. It is surrounded by other, fainter gear shapes in the background.

Structural Mechanics

A series of concentric green circles, resembling a target or a cross-section of a magnetic field. The circles are centered and have a glowing, ethereal quality.

Electromagnetics

A 3D arrangement of several blue, translucent cubes. Some cubes are stacked on top of others, and there are light rays or energy beams emanating from the structure, suggesting a complex system or multiphysics simulation.

Systems and Multiphysics

Introduction to ANSYS Fluent

Lecture Theme:

The problem definition for all CFD simulations includes boundary conditions, cell zone conditions and material properties. The accuracy of the simulation results depends on defining these properly.

Learning Aims:

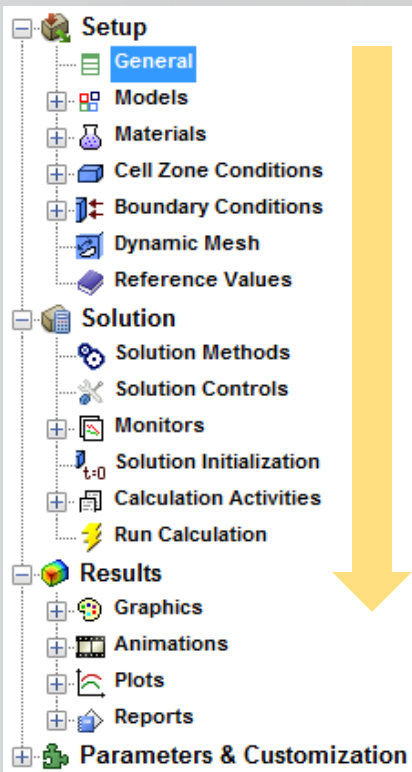
You will learn:

- How to define material properties
- The different boundary condition types in Fluent and how to use them
- How to define mesh interfaces
- How to define cell zone conditions in Fluent including solid zones and porous media
- How to specify well-posed boundary conditions

Learning Objectives:

You will know how to perform these essential steps in setting up a CFD analysis

ANSYS Fluent Workflow



Navigation Pane Guides Basic Workflow

- Read and check mesh (scale if needed in standalone mode)
- Select Physical Models
 - Energy
 - Turbulence
 - Multiphase
 -
- **Create/Assign Materials**
- **Assign Cell & Boundary Conditions**
- Choose Solver Settings
- Create Solution Monitors
- Initialize Solution
- Run Calculation
- Post-Process Results

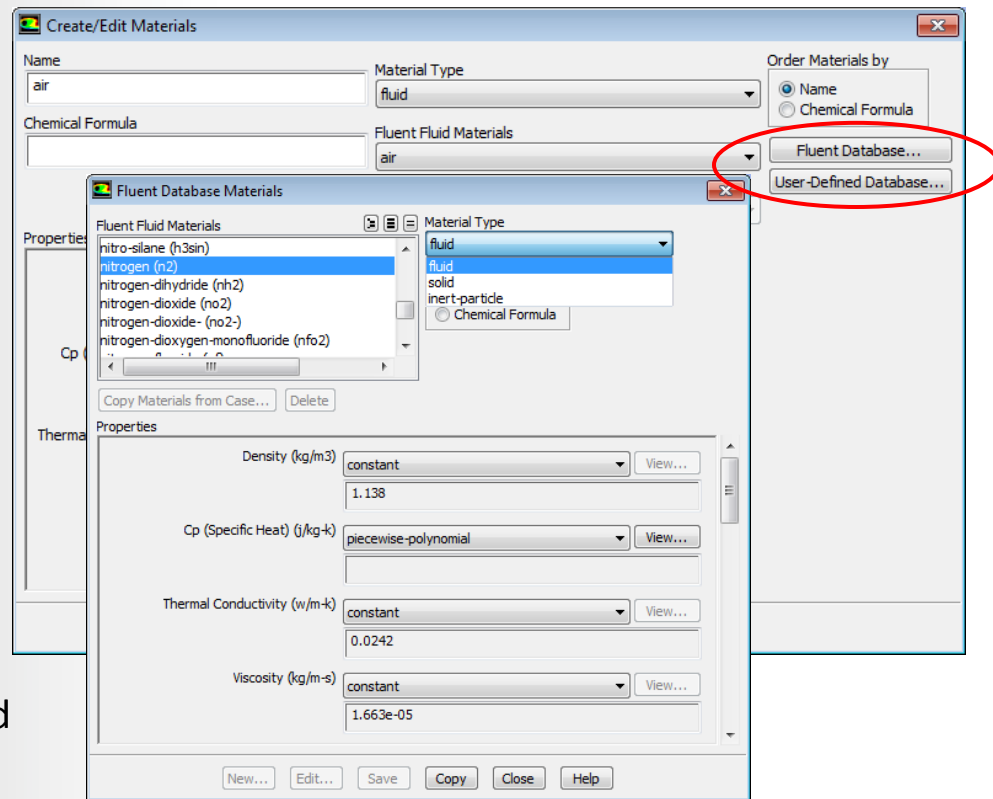
Materials Database

- **Fluent materials database**

- Provides access to a number of pre-defined fluid, solid and mixture materials
- Properties listed depend on the models used
- Materials can be copied to the case file and edited if required

- **User-Defined material database**

- Custom databases can be created, accessed and modified from the standard materials panel in Fluent



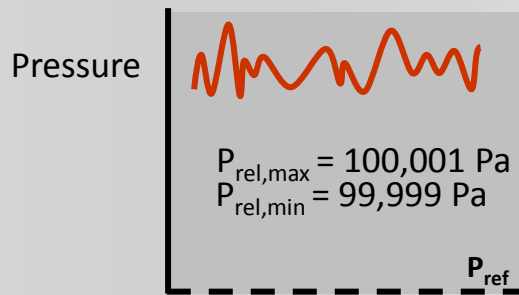
- **For incompressible flow with $\rho = \text{constant}$**
 - Select constant for density
- **Ideal gas properties**
 - Incompressible flow, $\rho = f(T)$
 - Polynomial or piecewise-polynomial
 - Incompressible ideal gas law ($\rho = p_{\text{operating}}/RT$)
 - Set $p_{\text{operating}}$ close to the mean pressure in the problem → [see next slide](#)
 - Compressible flow, $\rho = f(p,T)$
 - Use ideal-gas for density ($\rho = p_{\text{absolute}}/RT$)
 - For low-Mach-number flows, set $p_{\text{operating}}$ close to mean pressure of the problem to avoid round-off errors
 - Use Floating Operating Pressure for unsteady flows with large, gradual changes in absolute pressure (segregated solver only)

Operating Pressure

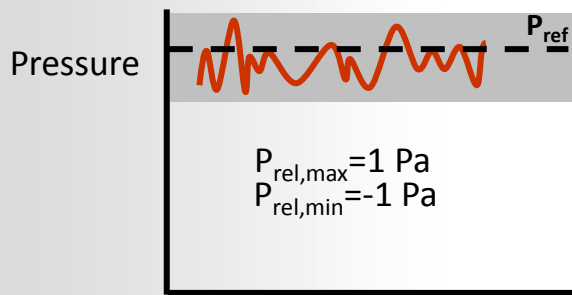
- Represents the absolute pressure datum from which all relative pressures are measured

$$P_{absolute} = P_{operating} + P_{relative}$$

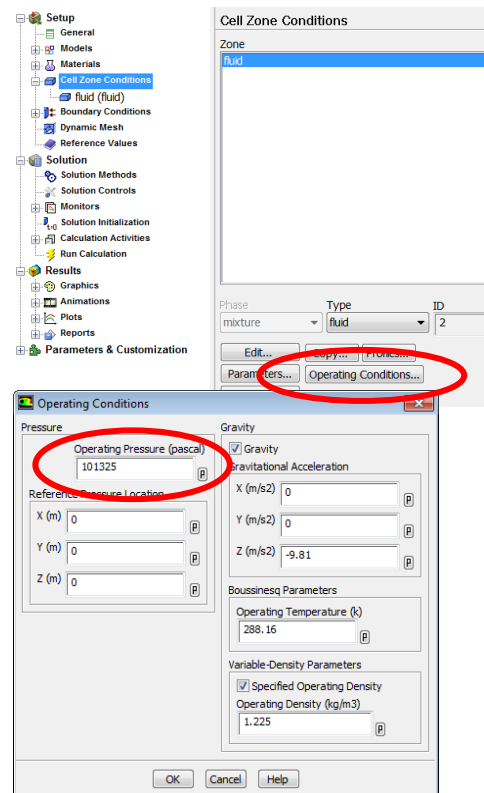
- Pressures specified at boundary conditions and initial conditions are relative to the Operating Pressure
- Used to avoid problems with round-off errors which occur when the dynamic pressure differences in a fluid are small compared to the absolute pressure level



Ex. 1: $P_{operating} = 0 \text{ Pa}$

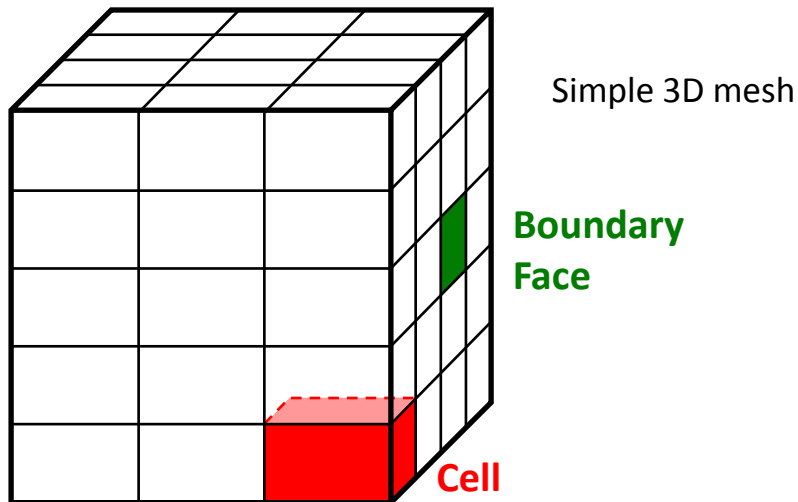


Ex. 2: $P_{operating} = 100,000 \text{ Pa}$



Cell Zones and Boundary Zones

- The mesh consists of a large number of finite volumes, or cells
- The cells are grouped into one or more **cell zones**
 - For instance in a conjugate heat transfer calculation there may be one cell zone for the fluid region and a second cell zone for the solid material
- Each cell is bounded by a number of faces
- These faces are grouped into a number of face zones
- Some of these faces are located on the boundaries of the model
- The zones to which such faces belong are called **boundary zones**

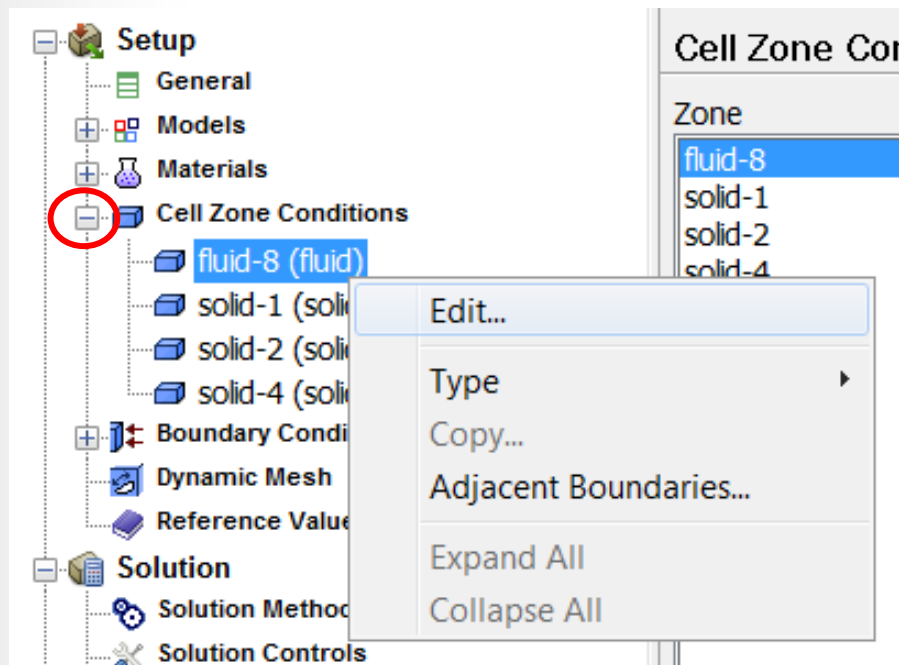


Cell zone conditions are applied to all cell zones

Boundary conditions are applied to all boundary zones

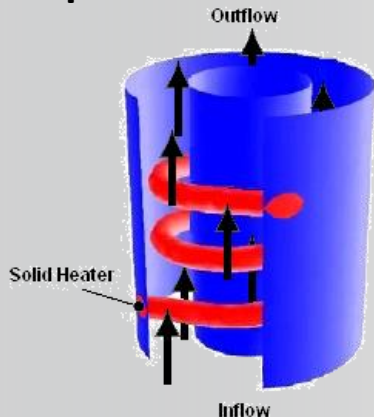
Opening the Cell Zone Conditions Panel

- Expand the Cell Zone Conditions branch of the Tree
- A list of all cell zones will appear
- Right click on the zone of interest and select Edit (or double click on the name of the zone)
- The cell zone conditions panel can be used to define a porous zone, prescribe energy sources in solid and fluid zones, specify inputs for rotating machinery, fix the values of one or more solution variables and many other operations

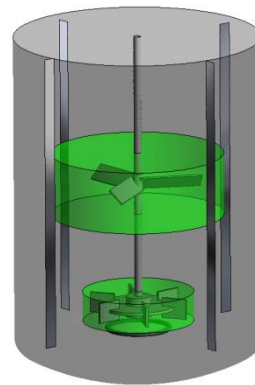


Cell Zones

- A fluid cell zone, or more simply, a fluid zone, is a group of cells for which all active equations are solved



e.g. A simulation of a copper heating coil in water will require a **fluid zone** and a **solid zone**. Using water properties, the equations of flow and heat transfer will be solved in the **fluid zone**. Using copper properties, only the heat transfer equation will be solved in the **solid zone**.



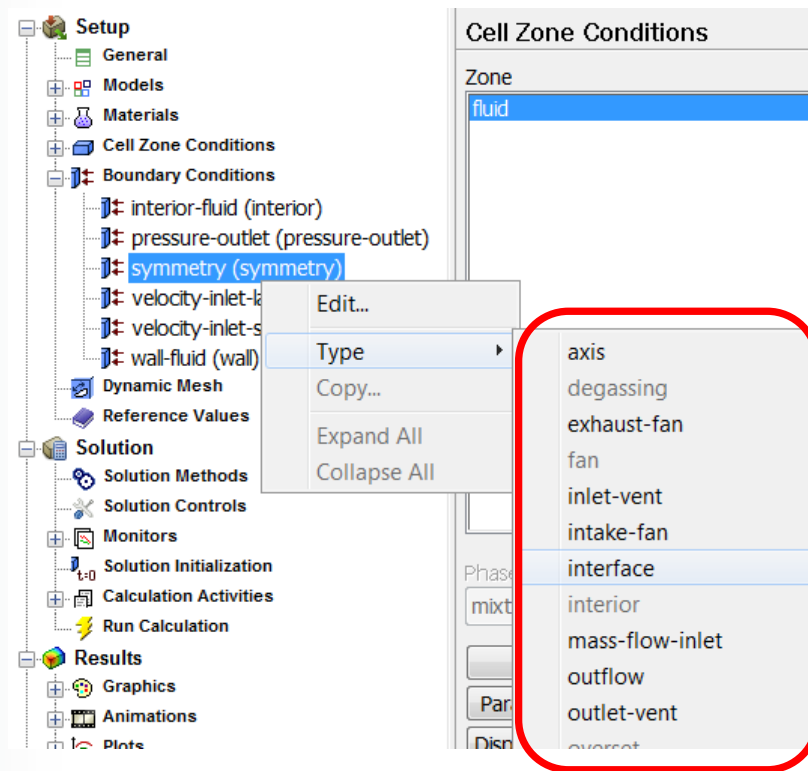
e.g. To account for rotational motion, the impellers are placed in a rotating domain. The **impeller fluid zones** will use equations in the rotating frame of reference. Everywhere else will use equations in the stationary frame of reference.

Defining Boundary Conditions

- **To define a problem that results in a unique solution, you must specify information on the dependent (flow) variables at the domain boundaries**
 - Specify fluxes of mass, momentum, energy, etc. into the domain
- **Poorly defined boundary conditions can have a significant impact on your solution**
- **Defining boundary conditions involves:**
 - Identifying types (e.g. inlets, walls, symmetry,...)
 - Identifying location
 - Supplying required data depending on boundary type, location and physical models
- **Choice depends on:**
 - Geometry
 - Availability of data at the boundary location
 - Numerical considerations

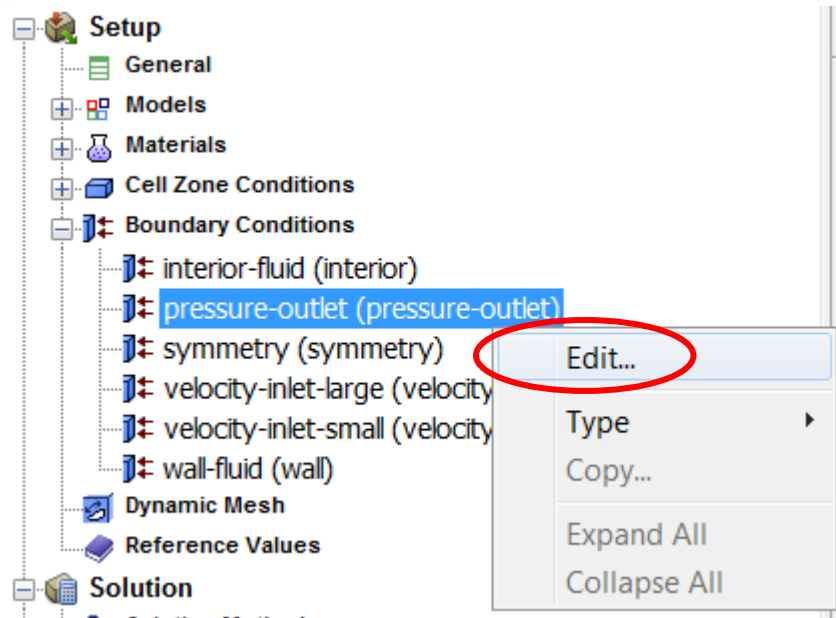
Changing Boundary Condition Types

- Zones and zone types are initially defined in the preprocessing phase
- To change the boundary condition type for a zone:
 - Right click on the zone name in the Tree
 - Select "Type" to open a list of available boundary types and select a new type from the list



Setting Boundary Condition Data

- **Explicitly assign data in BC panels**
 - To set boundary conditions for particular zone:
 - Expand the Boundary Conditions branch in the tree
 - RMB then Edit, or double click on the zone name
 - Boundary condition data can be copied from one zone to another
- **Boundary conditions can also be defined by User-Defined Functions (UDFs) and profiles**
 - Profiles can be generated by:
 - Writing a profile from another CFD simulation
 - Creating an appropriately formatted text file with boundary condition data
 - More information in Appendix



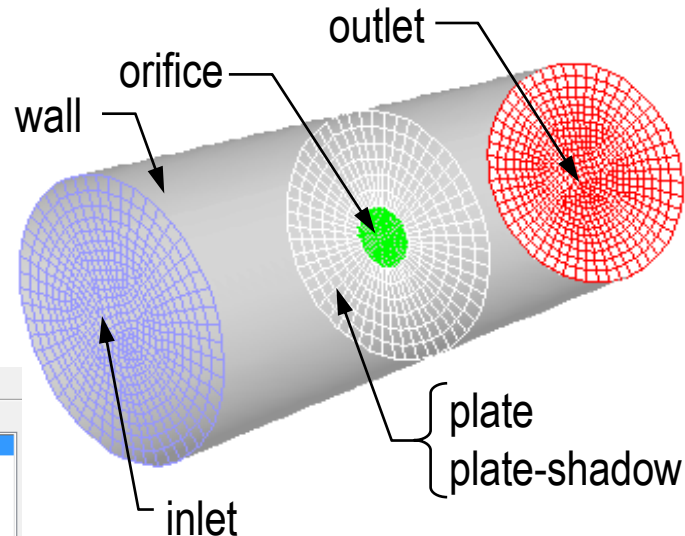
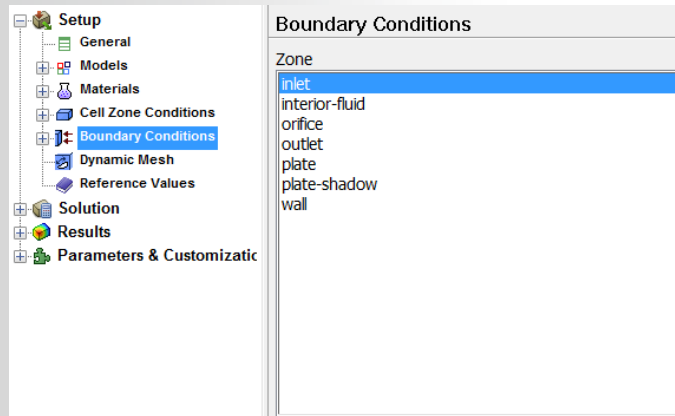
Available Boundary Condition Types

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



General Guidelines for Boundaries in CFD

- **If possible, select inflow and outflow boundary locations and shapes such that flow either goes in or out normal to the boundaries**
 - Typically better convergence
- **Should not observe large gradients in direction normal to boundary**
 - Indicates incorrect set-up
 - Move the boundary further upstream or downstream so it is located away from gradients
- **Minimize grid skewness near the boundary**
 - Errors resulting from high skewness will propagate through the rest of the computational domain

Specifying Well Posed Boundary Conditions

- Consider the following case which contains separate air and fuel supply pipes
- Three possible approaches in locating inlet boundaries:

1 Upstream of manifold

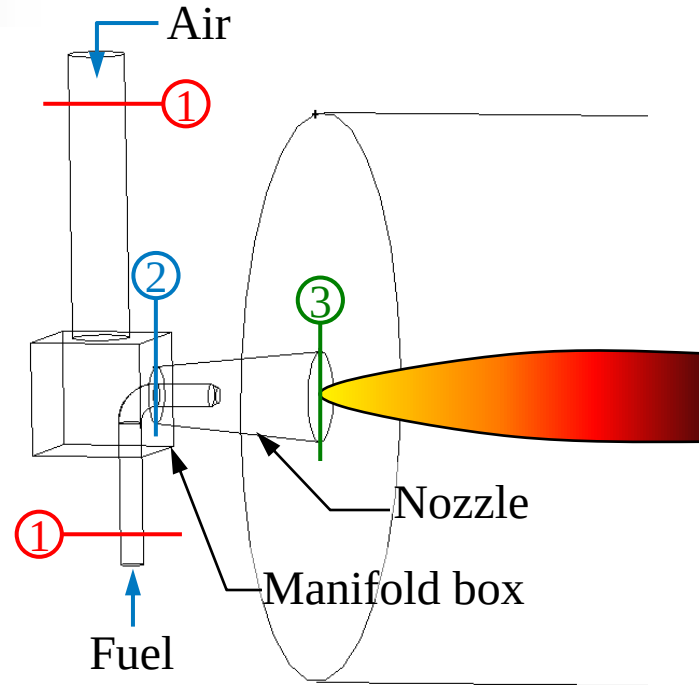
- Can use uniform profiles since natural profiles will develop in the supply pipes
- Requires more elements

2 Nozzle inlet plane

- Requires accurate velocity profile data for the air and fuel

3 Nozzle outlet plane

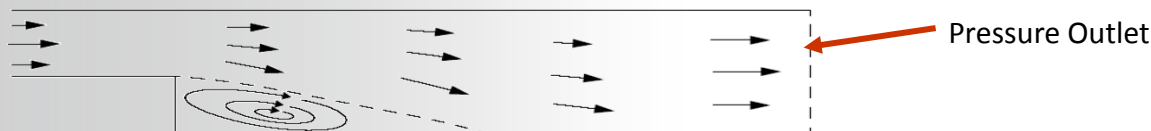
- Requires accurate velocity profile data and accurate profile data for the mixture fractions of air and fuel



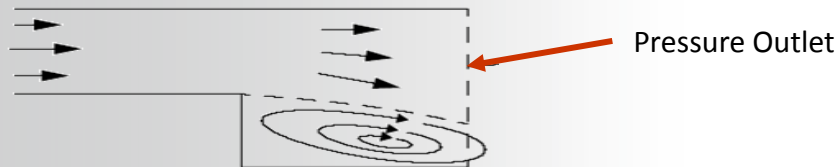
Specifying Well Posed Boundary Conditions

- **Boundaries near recirculation zones**

- **Ideal Location:** Apply an outlet downstream of the recirculation zone to allow the flow to develop. This will make it easier to specify accurate boundary conditions.



- **Bad Location:** Difficult to apply the correct backflow conditions for turbulence, temperature, species, etc. if the pressure outlet is located here.



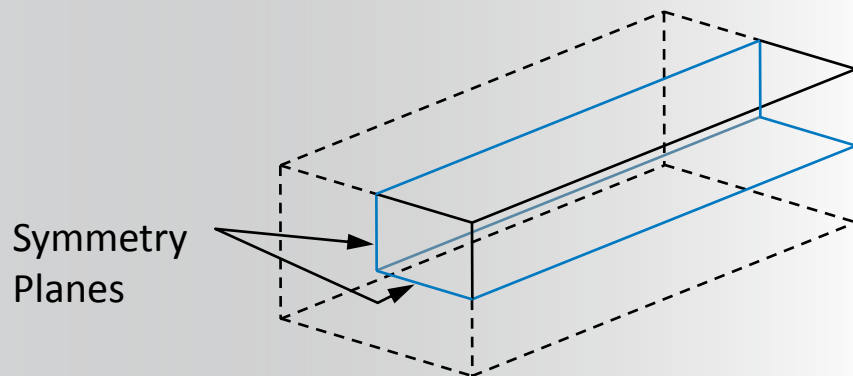
Specifying Well Posed Boundary Conditions

- **When there is 1 Inlet and 1 Outlet:**

- **Most Robust:** Velocity at inlet with static pressure at outlet (Velocity Inlet :: Pressure Outlet)
 - The inlet total pressure is an implicit result of the prediction
- **Robust:** Mass flow rate at inlet with static pressure at outlet (Mass Flow Inlet :: Pressure Outlet)
 - The total pressure at the inlet will be adjusted to set the given mass flow
- **Sensitive to Initial Guess:** Total pressure at inlet with static pressure at outlet (Pressure Inlet :: Pressure Outlet)
 - The system mass flow is part of the solution
- **Very Unreliable:**
 - Total pressure or mass flow rate at inlet with Outflow boundary at outlet (Pressure Inlet :: Outflow or Mass Flow Inlet :: Outflow)
 - This combination should not be used, because the static pressure level is not fixed
 - Mass Flow Inlet :: Outflow combination is ok if the density is constant
 - Velocity at inlet and velocity at outlet – system is numerically unstable

Symmetry Planes

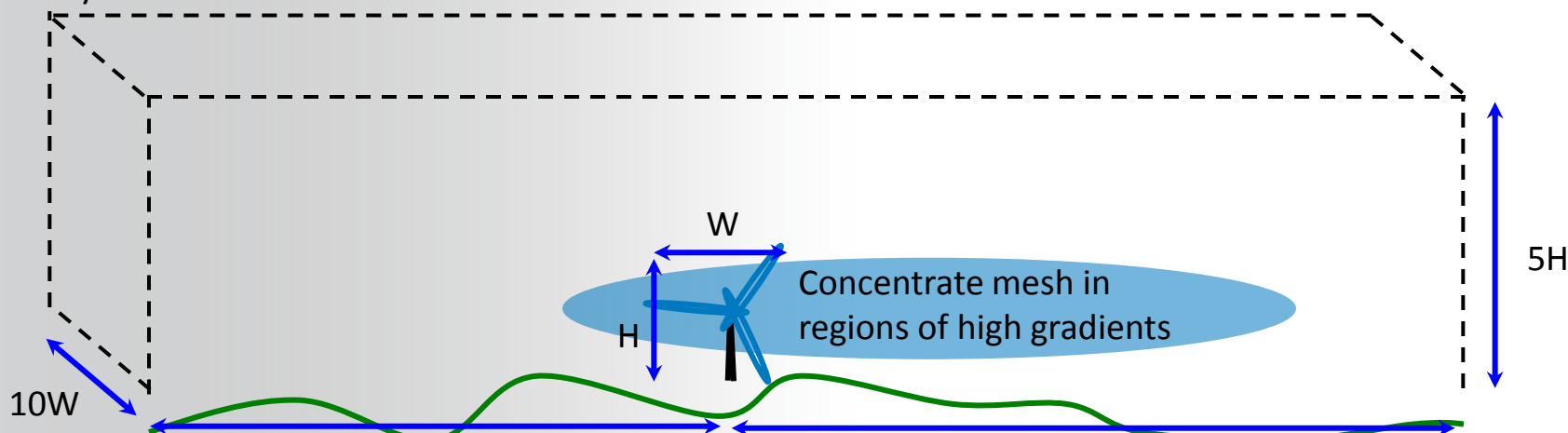
- **Symmetry boundary conditions can be applied at symmetry planes**
 - No inputs are required
 - Both the geometry and the flow field must be symmetric:
 - Zero normal velocity at symmetry plane
 - Zero normal gradients of all variables at symmetry plane
 - Must take care to correctly define symmetry boundary locations



Specifying Well Posed Boundary Conditions

- External Flow

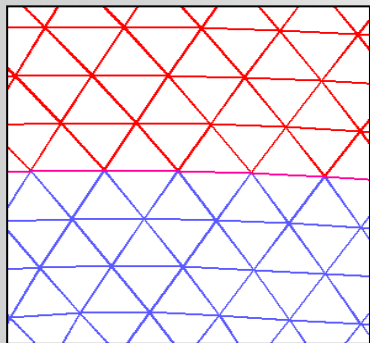
- In general, if the object (building, wind turbine, automobile,...) has height H and width W , you would want your domain to be at least more than: $5H$ high, $10W$ wide, with at least $2H$ upstream of the building and $10H$ downstream of the building
- You would want to verify that there are no significant pressure gradients normal to any of the boundaries of the computational domain. If there are, then it would be wise to enlarge the size of your domain



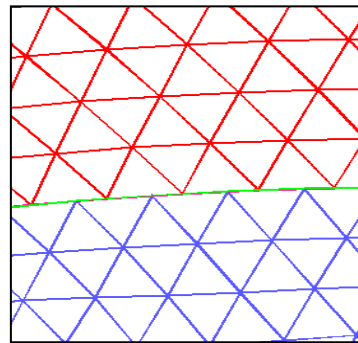
Mesh Interfaces

Across an interface between two cell zones, the nodes may or may not exactly align

- If the nodes match perfectly, this is a '**Conformal**' mesh
 - If using DesignModeler, combining bodies into a single part will give a conformal mesh
- If the nodes do not match up, this is a '**Non Conformal**' mesh
 - Fluent can interpolate across the interface, but this must be defined in the GUI.
 - If not, Fluent will treat the interface as a wall, and no fluid can flow through.



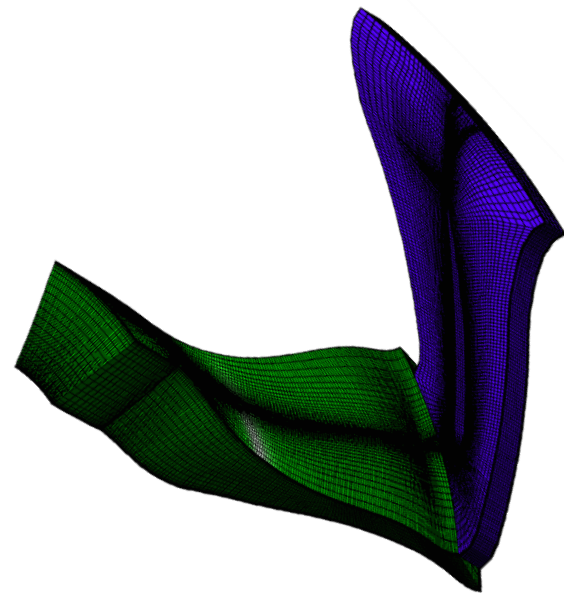
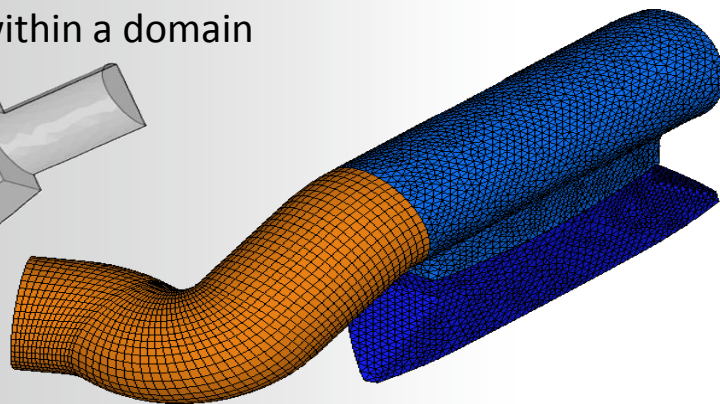
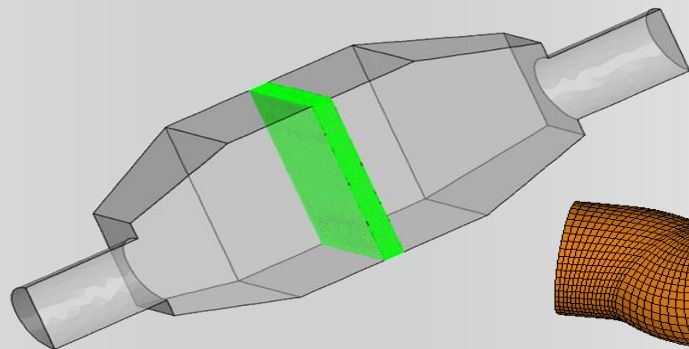
Conformal



Non-Conformal

Non-conformal Interfaces can be used for:

- Connection of mismatched meshes (hex to tet for example)
 - a single mesh file may contain non-matching mesh regions and require non-conformal interfaces
- Changes in reference frames between cell zones
 - even if the mesh matches
- Connect different types of cell zones together (e.g. Fluid and Solid)
- Create periodic regions within a domain



Defining Mesh Interfaces

- To create a non-conformal interface:
 - **Step 1: Define/Boundary Conditions**
 - Change the type of each pair of zones that comprises the non-conformal boundary to interface
 - **Step 2: Define/Mesh Interfaces**
 - Enter a name for the interface in the Mesh Interface text-entry box
 - Specify the zones comprising the interface by selecting one or more zones below Interface Zone 1 and one or more below Interface Zone 2
 - If one interface zone is much smaller than the other, specify the smaller zone as Interface Zone 1 to improve the accuracy of the intersection calculation
 - Enable the desired interface options if appropriate

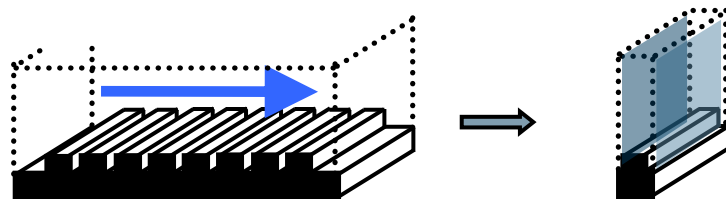
Periodic Boundary Condition

- **To create a Periodic boundary condition**

- Enable the Periodic Boundary Condition option in the Mesh interfaces panel
- Select either Translational or Rotational as the periodic boundary condition Type
 - Retain the enabled default setting of Auto Compute Offset if you want ANSYS Fluent to automatically compute the offset
- Mesh can be conformal or non-conformal

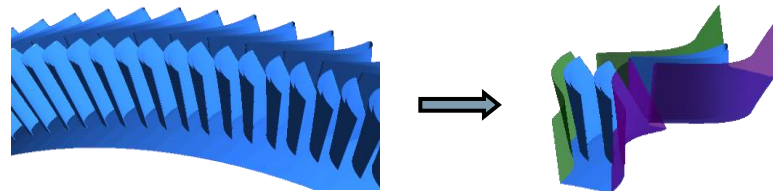
- **Translational Periodicity**

- Simulates geometries that have translational periodicity
- Allows for either the mass flow rate or the pressure change across the interface to be specified
- The quantity not specified will be part of the solution



- **Rotational Periodicity**

- Simulates rotationally periodic geometries
- Before proceeding, you have to correctly enter the rotational axis for the corresponding cell zone in the BC panel



- **Cell zones (or continuum zones) are used to assign which fluid/solid material(s) exist in a region**
 - Also have options for porous media, laminar region, fixed value, etc.
- **Fluent has an extensive, customizable database of material properties**
 - Numerous models and options are available for definition of properties
- **Mesh interfaces can be defined to allow flow and energy to pass between zones connected with non-conformal mesh**
- **Boundary zones are used to assign boundary conditions at external and internal boundaries**
- **Locations and types of boundary condition are extremely important for good convergence and accurate results**

Appendix

Options for Defining Common Properties

Density

- Constant
- Incompressible Ideal Gas
- Ideal Gas
- Real Gas (5 Built-in Models)
- Temperature Dependent¹
- Boussinesq
- User-defined

Viscosity

- Constant
- Temperature Dependent¹
- Sutherland
- Power Law
- Kinetic Theory
- Non-Newtonian (4 Built-in Models)
- User-defined

Thermal Conductivity

- Constant
- Temperature Dependent¹
- Kinetic Theory
- User-defined

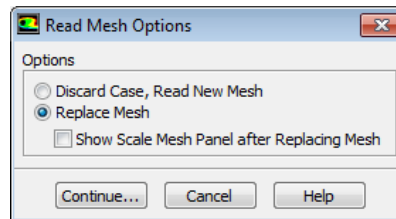
Specific Heat

- Constant
- Temperature Dependent¹
- User-defined

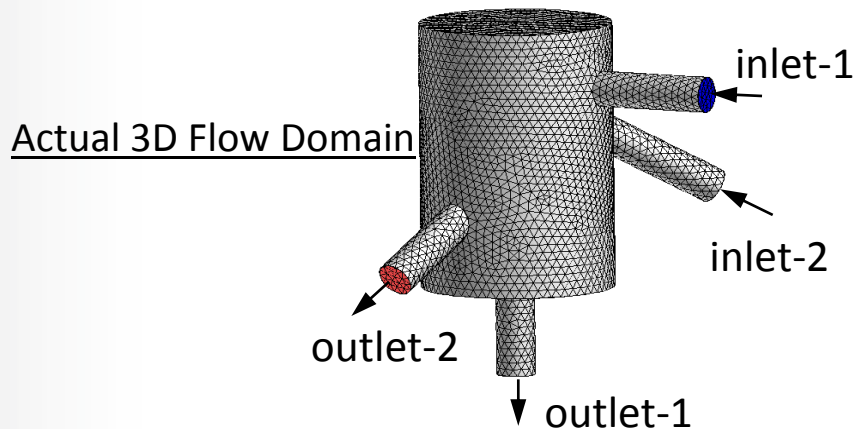
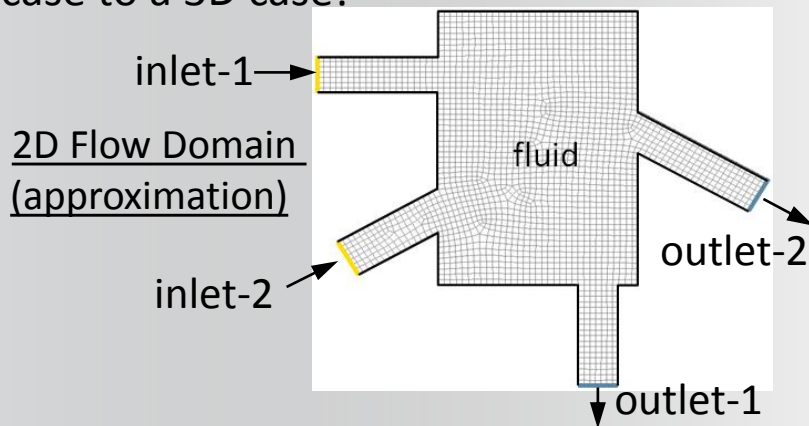
¹ Temperature Dependent options include definition of properties as piecewise linear , polynomial or piecewise polynomial functions temperature

Case Setup Replication

- To replicate a case setup:
 - Open the actual case file and go to File/Read/Mesh
 - Choose the option : Replace Mesh

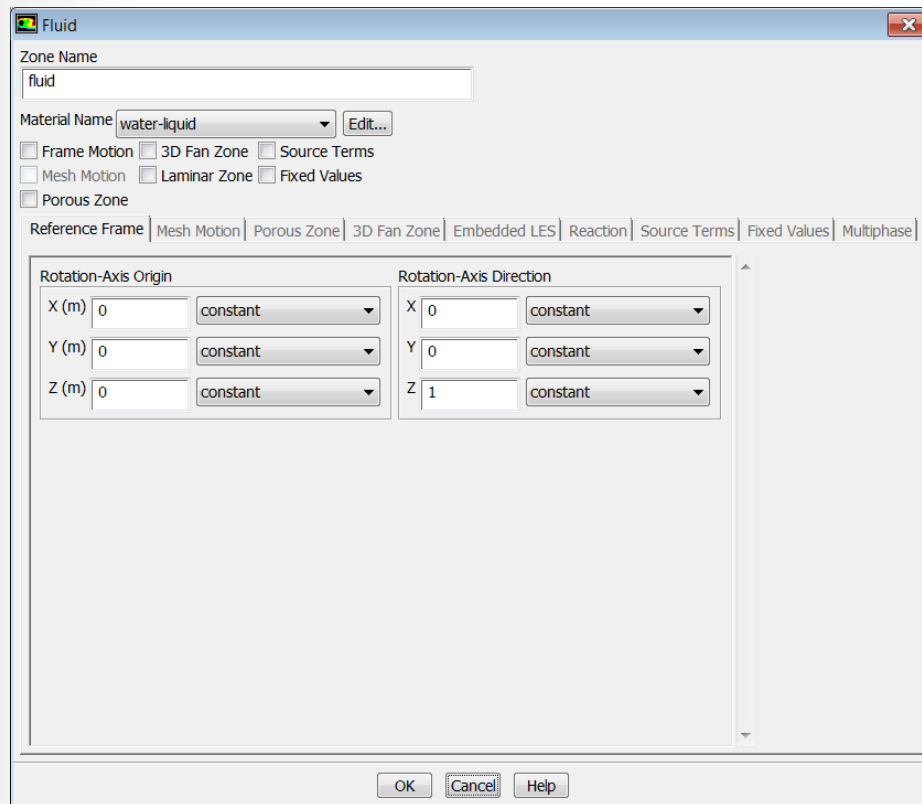


The text user interface (TUI) commands `/file/write-settings` and `/file/read-settings` can also be used for the same purpose. You can transfer settings from a 2D case to a 3D case!



Cell Zone Definition – Fluid

- **Fluid material selection is required**
 - For multiple species or multiphase flows, the material is not shown
 - Instead, the fluid zone consists of the mixture of the phases
- **Optional inputs**
 - Frame/Mesh Motion
 - 3D Fan Zone
 - Porous region
 - Source terms
 - Laminar region
 - Fixed Values

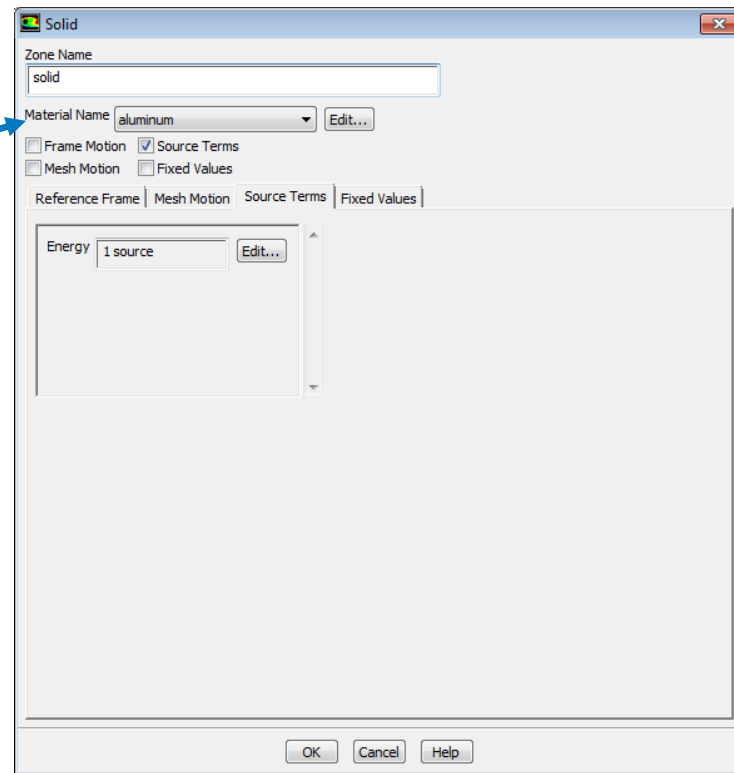


The screenshot shows the 'Fluid' dialog box in ANSYS. The 'Zone Name' field is set to 'fluid'. The 'Material Name' dropdown is set to 'water-liquid'. The 'Reference Frame' tab is selected, showing options for 'Frame Motion', '3D Fan Zone', 'Source Terms', 'Mesh Motion', 'Laminar Zone', and 'Fixed Values'. The 'Rotation-Axis Origin' and 'Rotation-Axis Direction' sections are visible, with X, Y, and Z coordinates and constant values.

Rotation-Axis Origin		Rotation-Axis Direction	
X (m)	0	X	0
Y (m)	0	Y	0
Z (m)	0	Z	1

Cell Zones Definition – Solid

- A solid zone is a group of cells for which only the heat conduction equation is solved. Flow equations are not solved
 - The only required input is the Material Name (defined in the Materials panel)
 - Optional inputs allow you to set volumetric heat generation rate (heat source)
 - Motion can be defined for a solid zone
 - However, this is uncommon
 - Rotation axis must be specified if the solid zone is rotating or if rotationally periodic boundaries are adjacent to the solid zone

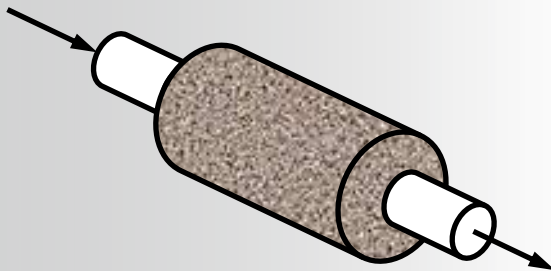


Cell Zone Definition - Porous Media

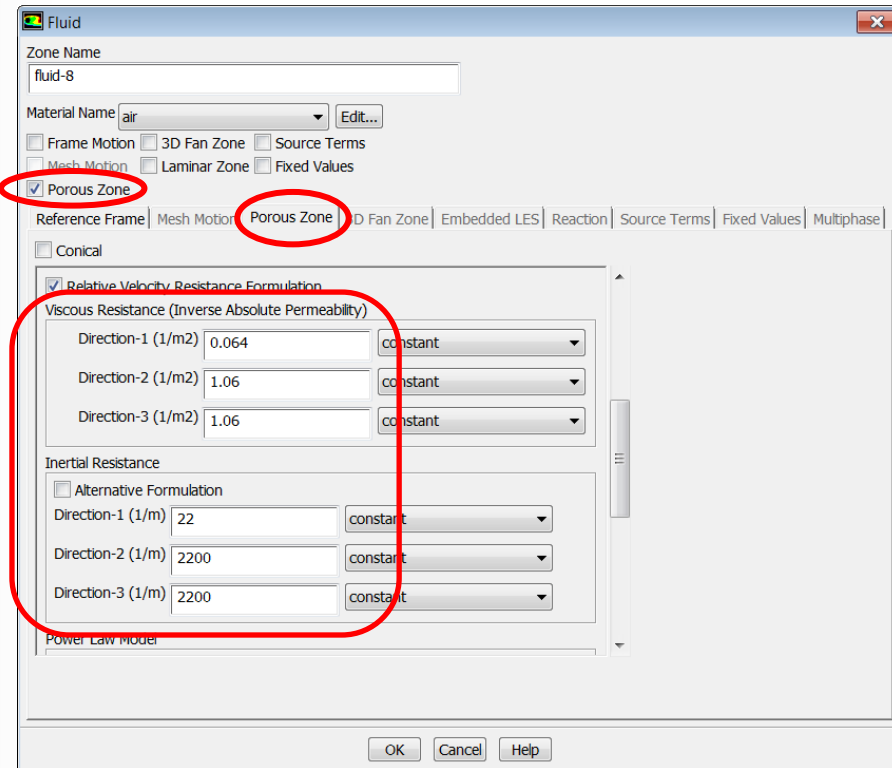
- A porous zone is a special type of fluid zone
 - Enable Porous Zone option in the Fluid panel
 - Pressure loss in flow determined via user inputs of resistance coefficients to lumped parameter model

- Used to model flow through porous media and other uniformly distributed flow resistances

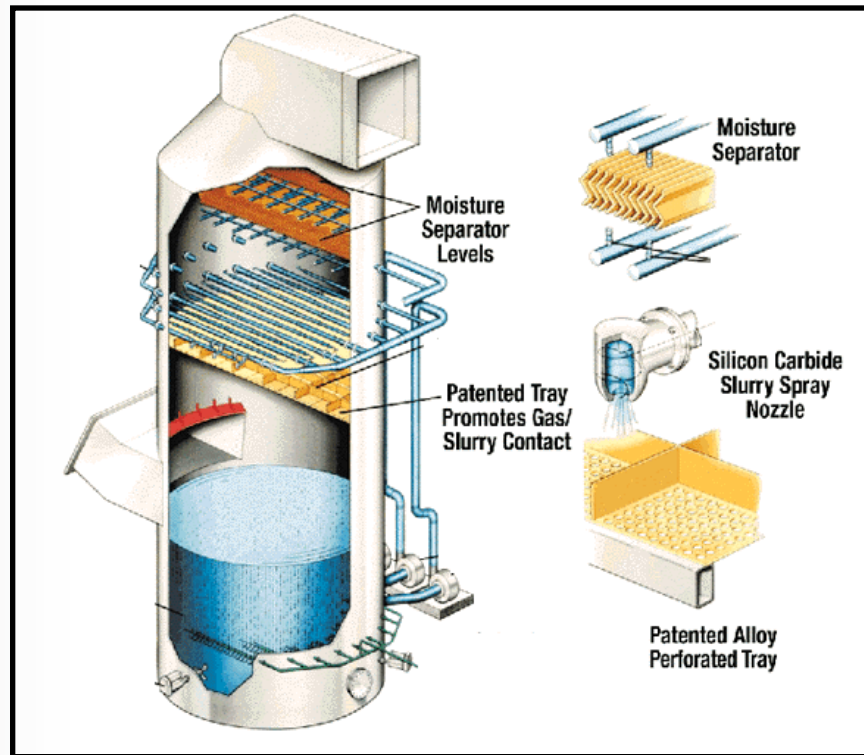
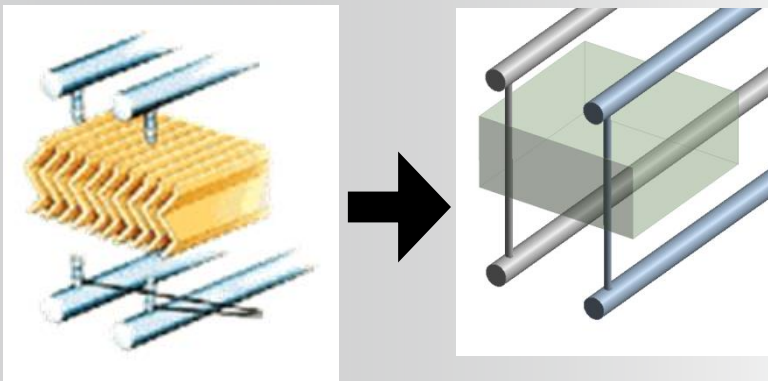
- Packed beds
- Filter papers
- Perforated plates
- Flow distributors
- Tube banks



- Inputs are directional viscous and inertial resistance coefficients



- Porous media can be used to model some flows where the geometry is too complex, or has too many scales, to resolve with a grid
- Instead of including the geometric details, their effects are accounted for numerically through the use of loss coefficients

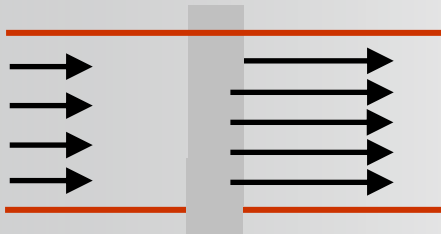


Images Courtesy of Babcock and Wilcox, USA

Porous Media Inputs

• Fluid Porosity

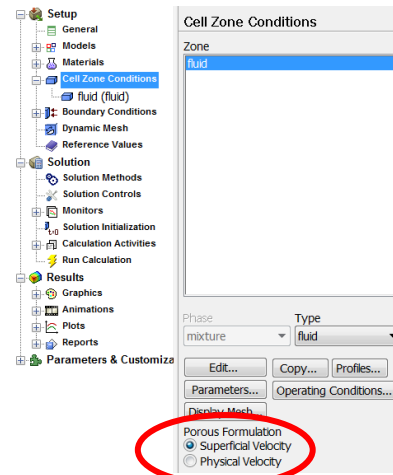
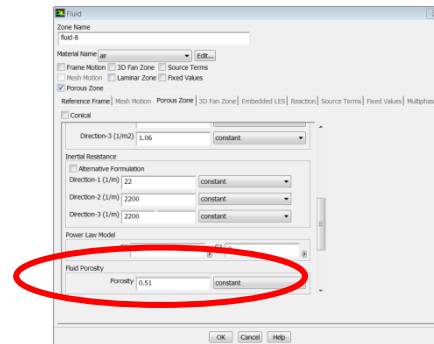
- The local ratio of the volume of fluid to the total physical volume



• Superficial Velocity

- By default ANSYS Fluent calculates the superficial velocity based on volumetric flow rate
- ANSYS Fluent allows the calculation of the physical velocity using the Porous Formulation

$$V_{\text{physical}} = \frac{V_{\text{superficial}}}{\gamma}$$



Porous Media Loss Coefficients

- **Directional Losses**

- Different losses can be defined for the streamwise and transverse directions
 - Honeycombs and Porous plates
- For an isotropic porous medium such as a packed bed, the same value can be applied in all three directions

- **Losses are applied using Darcy's Law**

- Permeability and Loss Coefficients

$$\frac{-dp}{dx_i} = \frac{\mu}{K_{perm}} U_i + K_{loss} \frac{\rho}{2} |U| U_i$$

- Viscous Resistance input is $(1/K_{perm})$ in the equation
- Inertial Resistance input is K_{loss} in the equation

Fluid

Zone Name: fluid

Material Name: water-liquid

☐ Frame Motion ☐ 3D Fan Zone ☐ Source Terms

☐ Mesh Motion ☐ Laminar Zone ☐ Fixed Values

☒ Porous Zone

Reference Frame | Mesh Motion | Porous Zone | 3D Fan Zone | Embedded LES | Reaction | Source Terms | Fixed Values | Multiphase

☐ Conical

☒ Relative Velocity Resistance Formulation

Viscous Resistance (Inverse Absolute Permeability)

Direction-1 (1/m2): 5149770 constant

Direction-2 (1/m2): 5149770 constant

Direction-3 (1/m2): 5149770 constant

Inertial Resistance

☐ Alternative Formulation

Direction-1 (1/m): 63 constant

Direction-2 (1/m): 63 constant

Direction-3 (1/m): 63 constant

Power Law Model

OK Cancel Help

Profile Data and Solution Data Interpolation

- Fluent allows interpolation of selected variable data on both face zones and cell zones by using profile files and data interpolation files, respectively.
 - For example, a velocity profile from experimental data or previous Fluent run at an inlet, or a solution interpolated from a coarse mesh to fine mesh.

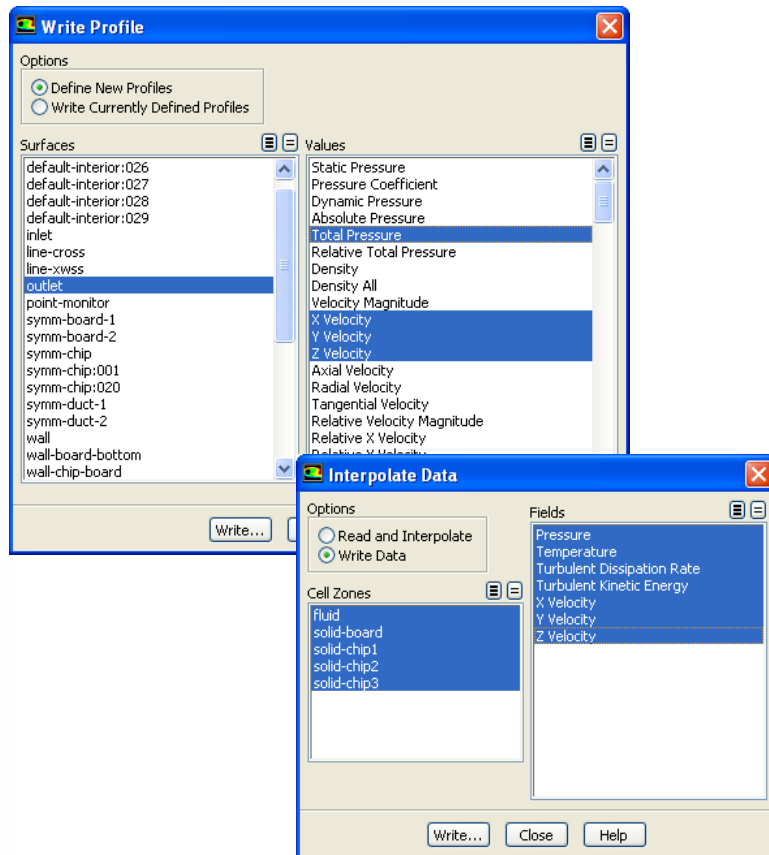
- Profile files are data files which contain point data for selected variables on particular face zones, and can be both written and read in a Fluent session.

File > Write > Profile

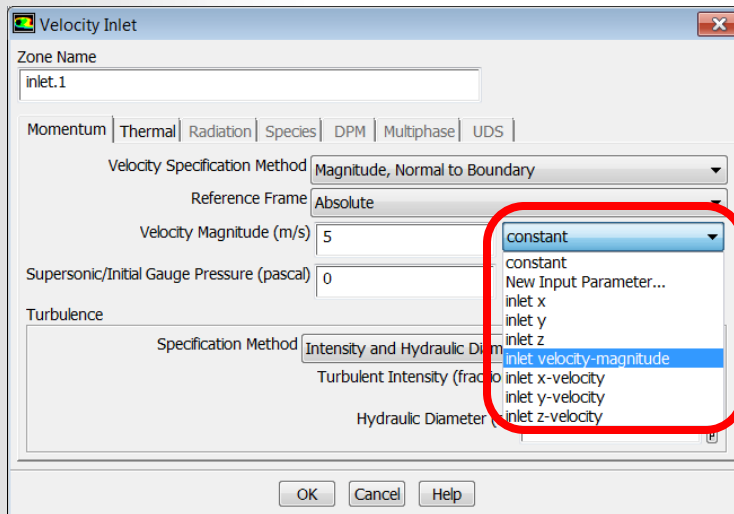
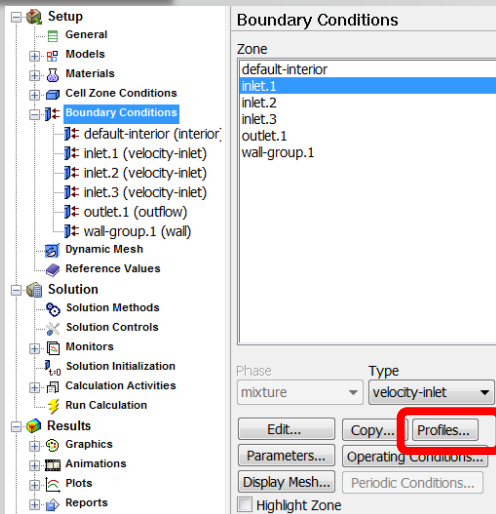
File > Read > Profile

- Similarly, interpolation files contain discrete data for selected field variables on particular cell zones to be written and read into Fluent.

File > Interpolate



Profile Boundary Conditions



- Select Profiles in the Boundary Conditions panel (left figure)
- After reading the profile, open the panel for the boundary where it is to be applied
- Select the arrow and scroll down in the drop-down list until the desired profile is reached (right figure)
 - The first three items in the list will usually be the the coordinates of the profile variables – do not select these
- Profiles can be created from experimental data by creating an appropriately formatted file
 - The file format details are in the User's Guide

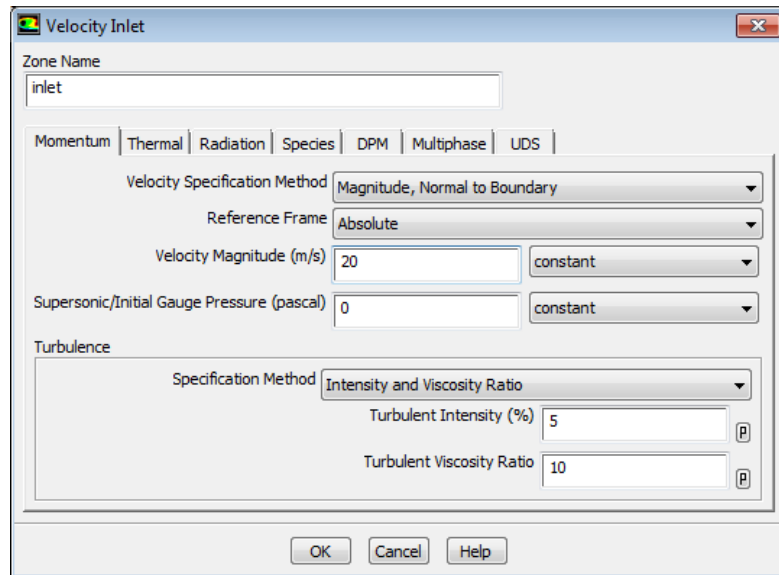
Velocity Inlet

- **Velocity Specification Method**

- Magnitude, Normal to Boundary
- Components
- Magnitude and Direction
- Turbulence quantities (if applicable)
 - Covered in Day 2
- Thermal conditions (if applicable)
 - Covered in Day 2

- Applies a uniform velocity profile at the boundary unless UDF or profile is used
- Velocity Magnitude input can be negative, implying that you can prescribe the exit velocity

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



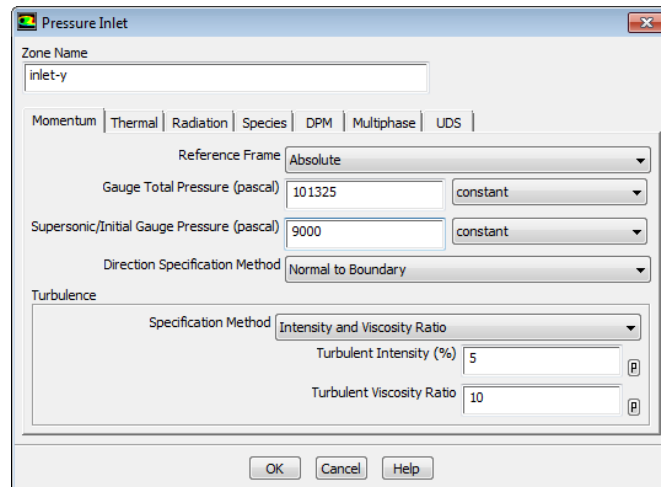
Pressure Inlet

- **Required inputs**

- Gauge *Total* Pressure
- Supersonic / Initial Gauge Pressure
- Inlet flow direction
- Turbulence quantities (if applicable)
- *Total* temperature (if heat transfer and/or compressible)

- Pressure inlet boundary is treated as a loss-free transition from stagnation to inlet conditions
- Fluent calculates the static pressure and velocity at the inlet and the mass flux through the boundary varies depending on the interior solution and specified flow direction

- **Pressure inlets are suitable for both compressible and incompressible flows**



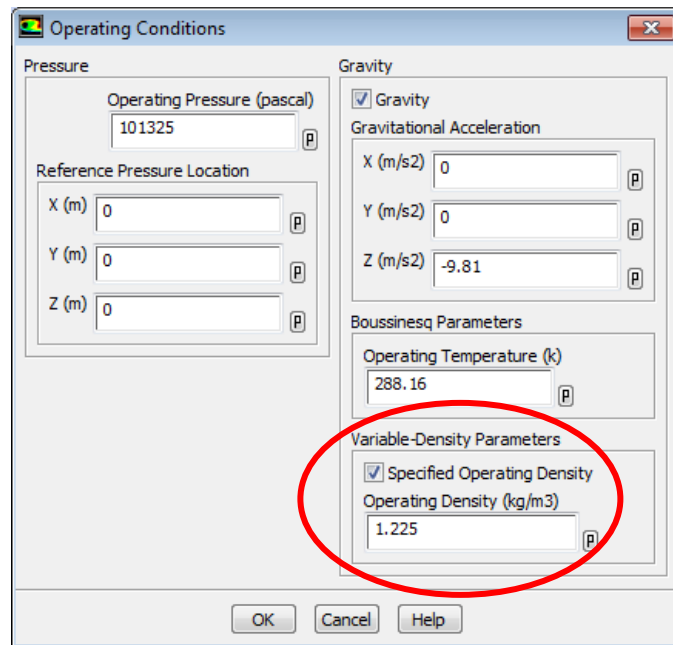
$$\text{Incompressible: } p_{\text{total}} = p_{\text{static}} + \frac{\rho V^2}{2}$$

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

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

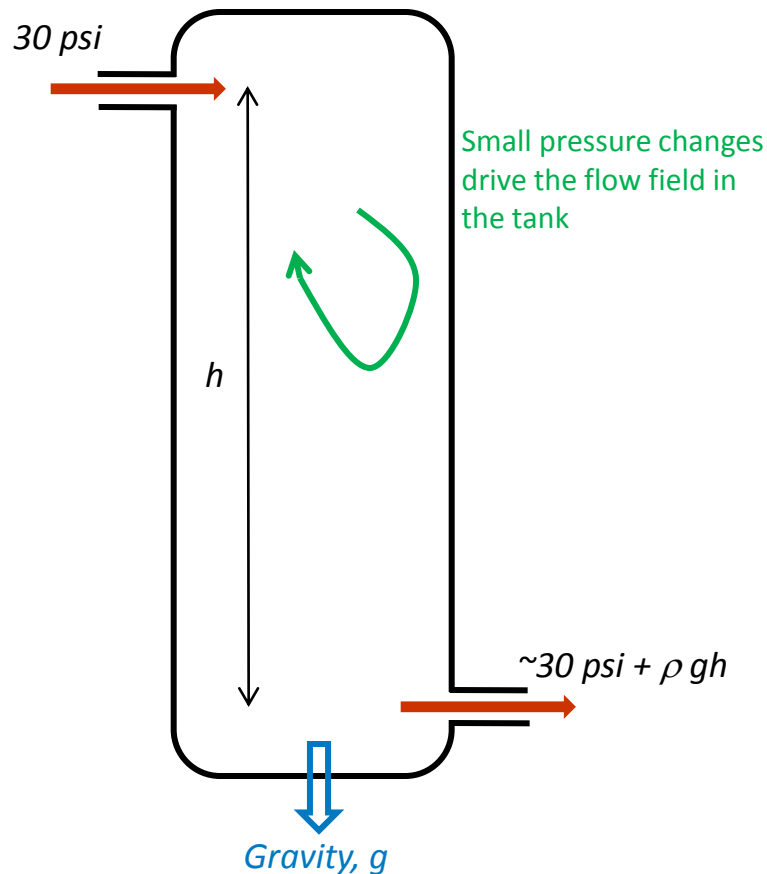
- **Operating Density**

- The *Operating Density* is used to avoid round-off errors by solving at an offset level
- Similar to the use of the *Operating Pressure* to offset the operating pressure of the domain, the *Operating Density* is used to offset the hydrostatic pressure in the domain
 - The pressure solution is relative to $\rho_{op} g h$, where h is relative to the Reference Location
 - If $\rho_{op} =$ the fluid density (ρ), then the pressure field in the solution is relative to the hydrostatic pressure, so when visualizing **Pressure** you only see the pressure that is driving the flow
- For a non-buoyant flow the numerical solution ignores the hydrostatic pressure as it has no effect on fluid motion
 - However, if necessary, hydrostatic pressure can be accounted for in Fluent for gravity driven flow of a constant density fluid



Pressure and Buoyancy Example

- Consider the case of flow through a tank
 - The inlet is at 30 [psi] absolute
 - Buoyancy is included, therefore a hydrostatic pressure gradient exists
 - The outlet pressure will be approximately 30 [psi] plus the hydrostatic pressure given by $\rho g h$
 - The flow field is driven by small dynamic pressure changes
 - NOT by the large hydrostatic pressure
 - To accurately resolve the small dynamic pressure changes, we use the *Operating Pressure* and the *Operating Density* to offset the hydrostatic pressure
- Details presented in Lecture 8: Heat Transfer*



Mass Flow Inlet

- **Required inputs**

- 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

- Total pressure adjusts to accommodate mass flow inputs
- More difficult to converge than Pressure Inlet

- **Mass flow inlets are intended for compressible flows; however, they can be used for incompressible flows**

Mass-Flow Inlet

Zone Name
inlet-z

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Reference Frame: Absolute

Mass Flow Specification Method: Mass Flow Rate

Mass Flow Rate (kg/s): 1.54 constant

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Direction Specification Method: Direction Vector

Coordinate System: Cartesian (X, Y, Z)

X-Component of Flow Direction: 1 constant

Y-Component of Flow Direction: 0 constant

Z-Component of Flow Direction: 0 constant

Turbulence

Specification Method: Intensity and Viscosity Ratio

Turbulent Intensity (%): 5 P

Turbulent Viscosity Ratio: 10 P

OK Cancel Help

Pressure Outlet

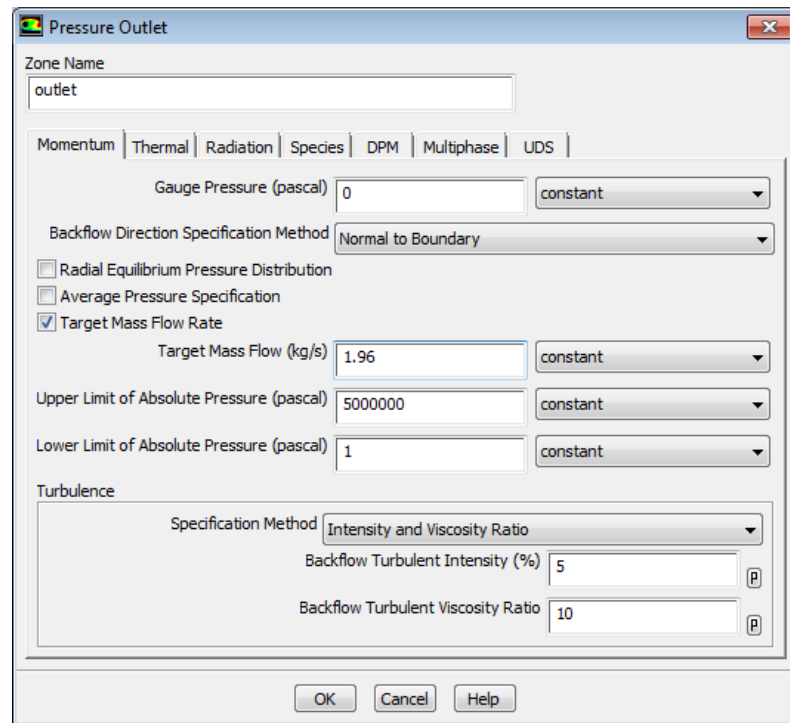
Required inputs

- Gauge Pressure (static) – static pressure of the environment into which the flow exits
 - Specified pressure is ignored if flow is locally supersonic at the outlet
- **Backflow quantities – Used as inlet conditions if/when backflow occurs (outlet acts like an inlet)**

- Can be used as a “free” boundary in an external or unconfined flow
- Target Mass Flow Rate Option can be applied
 - Not available for multiphase problems

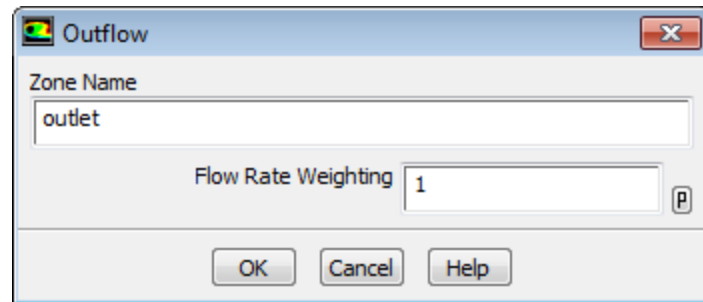
Suitable for compressible and incompressible flows

- Non-reflecting outlet boundary conditions (NRBC) are available for ideal gas (compressible) flow



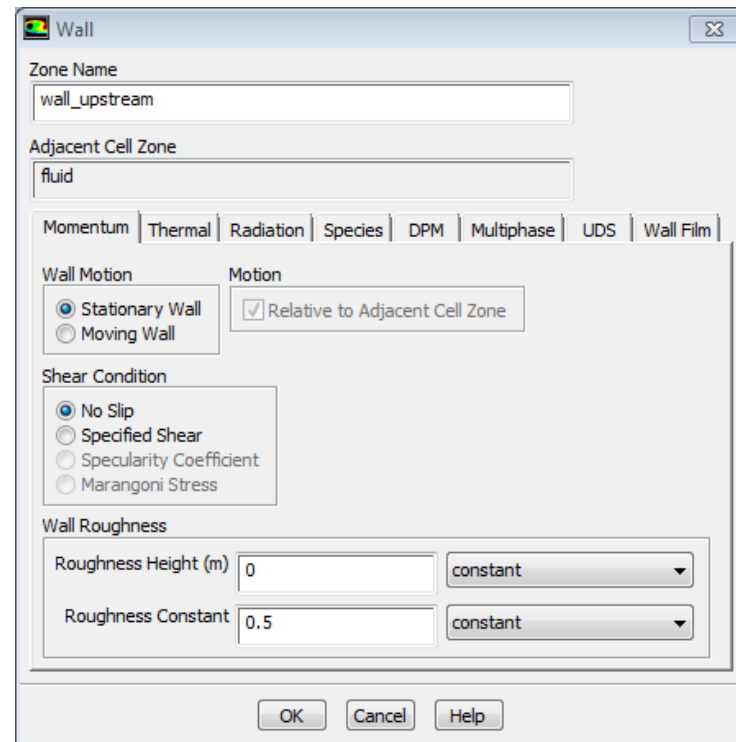
The screenshot shows the 'Pressure Outlet' dialog box in ANSYS. The 'Zone Name' is 'outlet'. The 'Momentum' tab is selected. The 'Gauge Pressure (pascal)' is set to 0 with a 'constant' dropdown. The 'Backflow Direction Specification Method' is 'Normal to Boundary'. The 'Radial Equilibrium Pressure Distribution' and 'Average Pressure Specification' are unchecked. The 'Target Mass Flow Rate' is checked, with a 'Target Mass Flow (kg/s)' of 1.96 and a 'constant' dropdown. The 'Upper Limit of Absolute Pressure (pascal)' is 5000000 with a 'constant' dropdown. The 'Lower Limit of Absolute Pressure (pascal)' is 1 with a 'constant' dropdown. The 'Turbulence' section shows the 'Specification Method' as 'Intensity and Viscosity Ratio'. The 'Backflow Turbulent Intensity (%)' is 5 with a 'p' icon. The 'Backflow Turbulent Viscosity Ratio' is 10 with a 'p' icon. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

- **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
- **Limitations :**
 - **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**
- **The outflow boundary is intended for use with incompressible flows**
- **Poor rate of convergence when backflow occurs during iterations**
 - Cannot be used if backflow is expected in the final solution



Wall Boundaries

- In viscous flows, no-slip conditions are applied at walls
 - Shear stress can be applied
 - Wall roughness can be defined for turbulent flows
 - Modification of the Logarithmic Standard Wall Function
- Wall thermal boundary conditions described in Lecture 08 "Heat Transfer"



The image shows the 'Wall' dialog box in ANSYS. The 'Zone Name' field contains 'wall_upstream' and the 'Adjacent Cell Zone' field contains 'fluid'. The 'Momentum' tab is selected, showing 'Stationary Wall' as the 'Wall Motion' and 'No Slip' as the 'Shear Condition'. The 'Wall Roughness' section has 'Roughness Height (m)' set to 0 and 'Roughness Constant' set to 0.5, both with 'constant' as the profile. The 'Motion' checkbox 'Relative to Adjacent Cell Zone' is checked. The bottom of the dialog has 'OK', 'Cancel', and 'Help' buttons.

Wall

Zone Name
wall_upstream

Adjacent Cell Zone
fluid

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS | Wall Film

Wall Motion
☒ Stationary Wall
☐ Moving Wall

Motion
☒ Relative to Adjacent Cell Zone

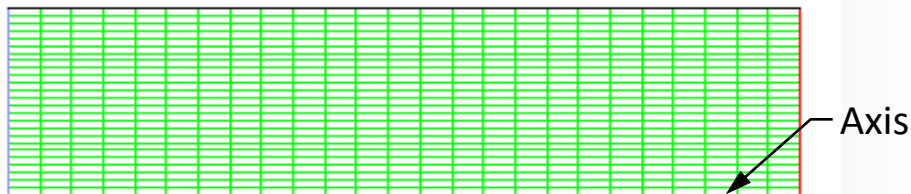
Shear Condition
☒ No Slip
☐ Specified Shear
☐ Specularity Coefficient
☐ Marangoni Stress

Wall Roughness
Roughness Height (m) 0 constant
Roughness Constant 0.5 constant

OK Cancel Help

Axis Boundaries

- **An axis boundary is used at the center line for 2d axisymmetric problems**
 - Does not apply in 3d cases
- **No user inputs required**
- **Fluent requires any axis boundary to be located at $y=0$**
 - Solution will diverge immediately if this condition is not satisfied
 - If necessary, use Grid > Translate to reposition the mesh so the axis is at $y=0$



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

- **Exhaust Fan / Outlet Vent**

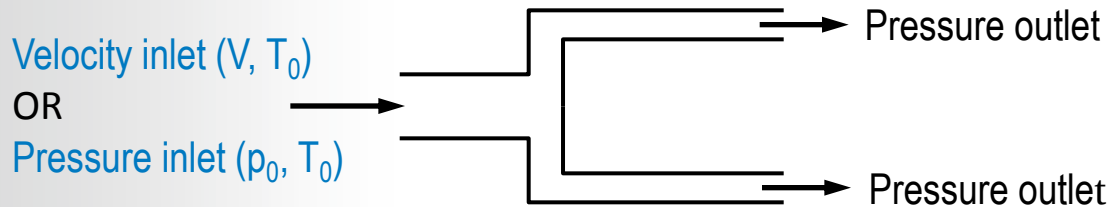
- Models an external exhaust fan or outlet vent with specified pressure rise/loss coefficient and ambient discharge pressure and temperature

- **Inlet Vent / Intake Fan**

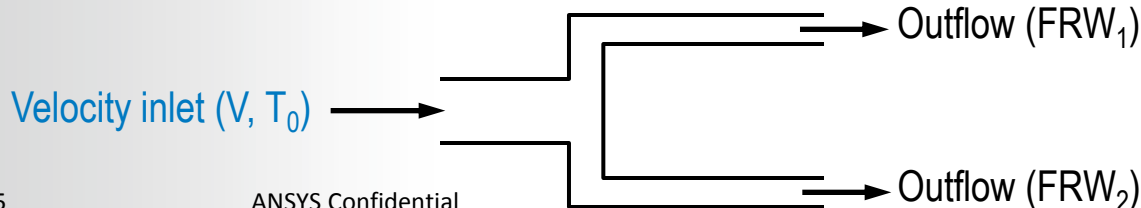
- Models an inlet vent / external intake fan with specified loss coefficient / pressure rise, flow direction and ambient inlet pressure and temperature

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

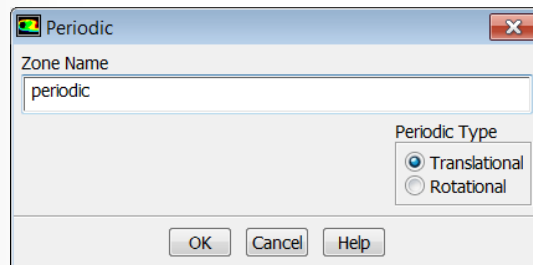


- Outflow:
 - Mass flow rate fraction determined from Flow Rate Weighting (FRW, varies from 0 to 1) by
$$\frac{\dot{m}_i}{\dot{m}_{\text{total}}} = \frac{\text{FRW}_i}{\sum_i \text{FRW}_i}$$
 - Static pressure varies among exits to accommodate the prescribed flow distribution

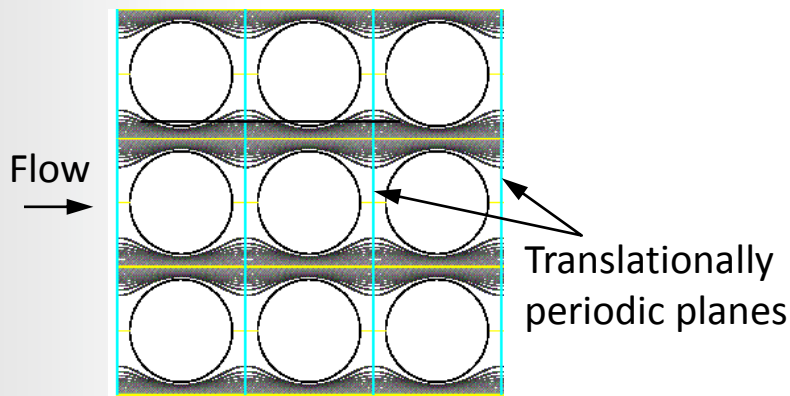
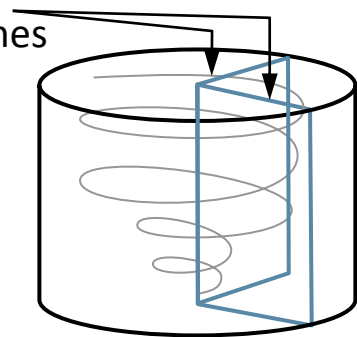


Periodic Boundaries

- Used to reduce the overall mesh size
- Flow field and geometry must contain either rotational or translational periodicity
 - Rotational periodicity
 - $\Delta P = 0$ 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 boundaries can be either conformal or non-conformal
 - See next two slides



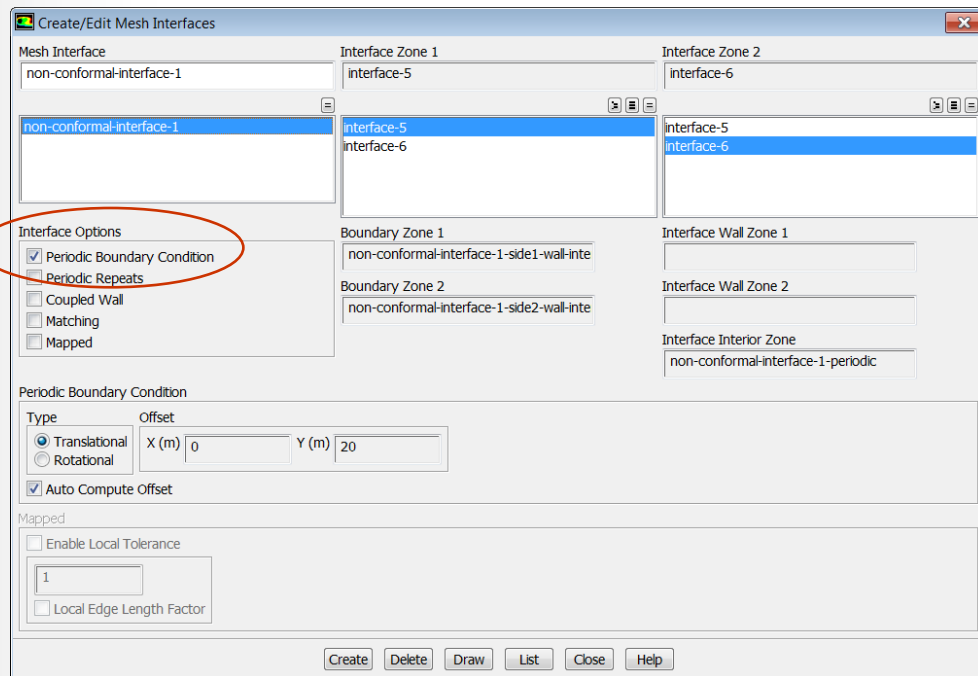
Rotationally periodic planes



2D Tube Heat Exchanger

Non-conformal Periodic Boundary Conditions

- Fluent permits the use of non-conformal rotationally periodic BCs
- Non-conformal periodics do not require a matching mesh on the boundaries
 - Coupling of the periodic zones is accomplished using the same algorithms employed in non-conformal interfaces
- Non-conformal periodic can now be created in the Create/Edit Mesh Interfaces GUI!
 - Select Periodic Boundary Condition option and choose the Type (Translational or Rotational)
 - Offset is computed automatically, but check this value to make sure it is evenly divisible into 360 deg!

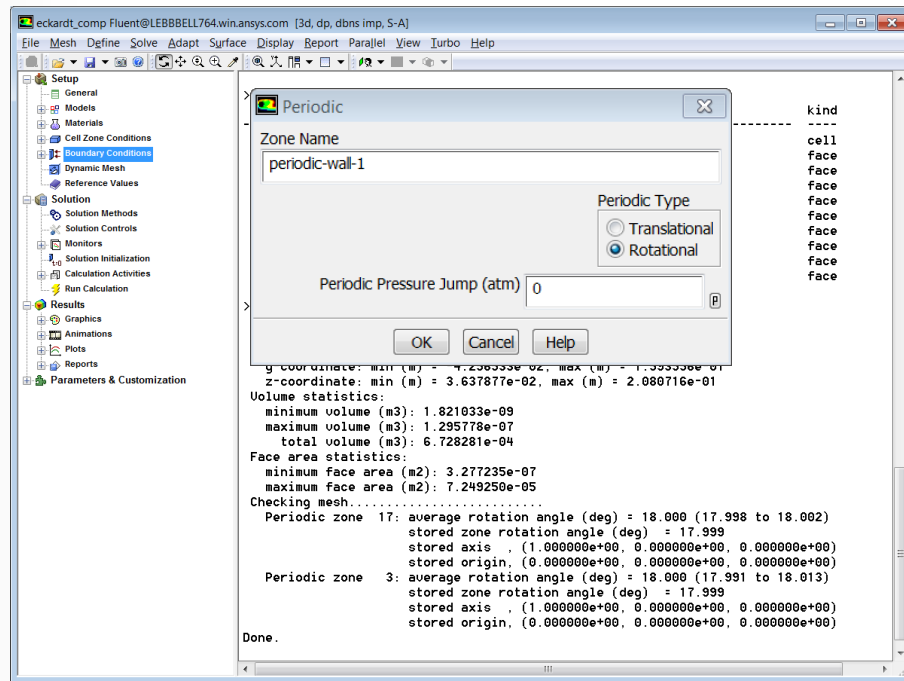


Conformal Periodic Boundary Conditions

- Conformal periodic BCs in Fluent require that the boundary face mesh elements match one-for-one on the periodic boundary
- Rotationally periodic BCs rely on the rotational axis specification to transfer information correctly
- Rotationally periodic boundaries can be used in moving reference frame problems to reduce mesh size provided both the geometry and flow are periodic

Notes:

- If you are using the `make-periodic` command in the TUI, make sure you set the rotational axis in the Fluid BC panel first before creating the periodic BCs
- Once the periodic BCs have been set, perform a mesh check to see if the reported periodic angles are correct



Internal Face Boundaries

- **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

Specifying Well Posed Boundary Conditions

- **Turbulence at an Inlet:**

- If you have no idea, use the default settings:
 - Turbulent Intensity = 5%
 - Turbulent Viscosity Ratio = 10
- Typically, turbulence intensities range from 1% to 5% but will depend on your specific application. The values given above are sufficient for nominal turbulence through a circular inlet, and are good estimates in the absence of experimental data
- For situations where turbulence is generated by wall friction, consider extending the domain upstream to allow the walls to generate turbulence and the flow profiles to become developed
- Addressed in ***Lecture 7: Turbulence***