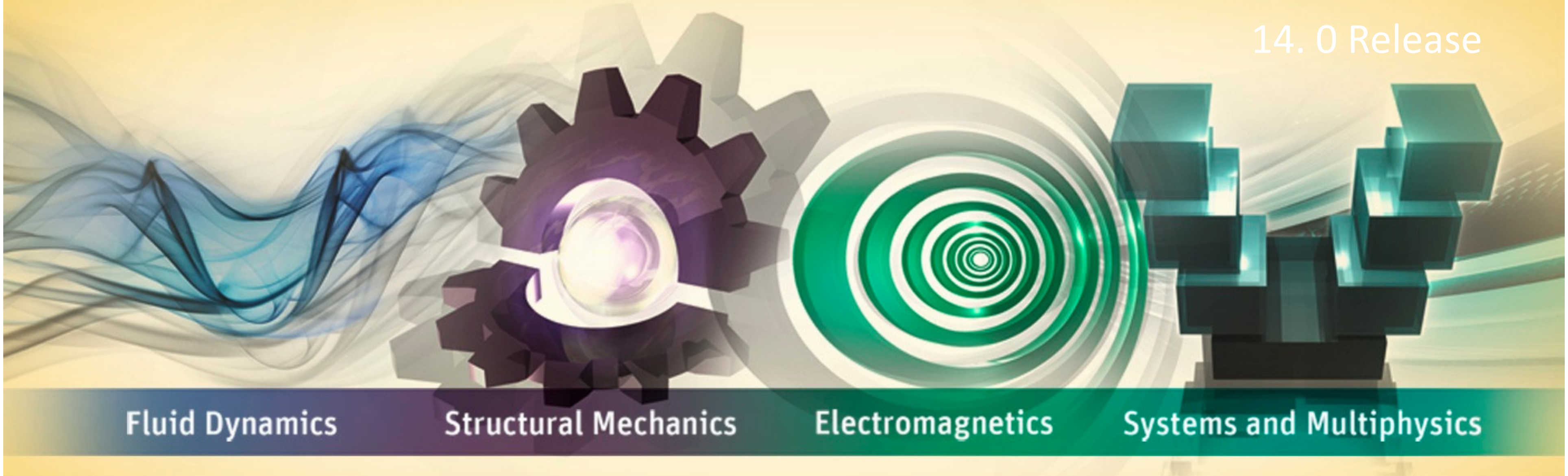
Realize Your Product Promise™

# Lecture 2
# Boundary Conditions

14. 0 Release




# Introduction to ANSYS FLUENT

# Introduction

**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 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

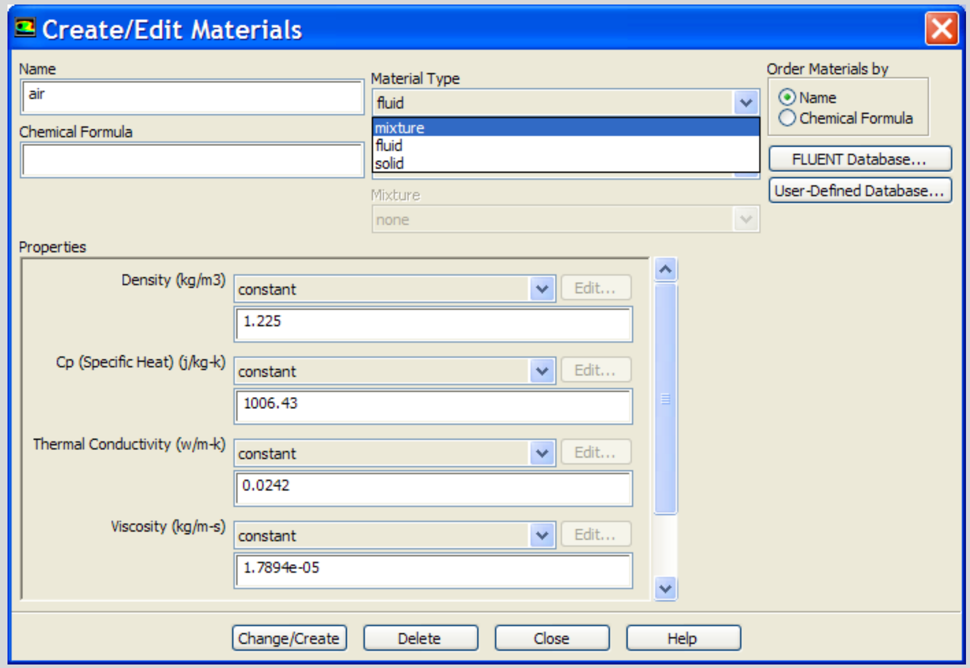
  

# Material Properties

- **FLUENT provides a standard database of materials and the ability to create a custom user-defined database**

- **Your choice of physical models may require multiple materials and dictate which material properties must be defined**
  - Multiphase (multiple materials)
  - Combustion (multiple species)
  - Heat transfer (thermal conductivity)
  - Radiation (emissivity and absorptivity)

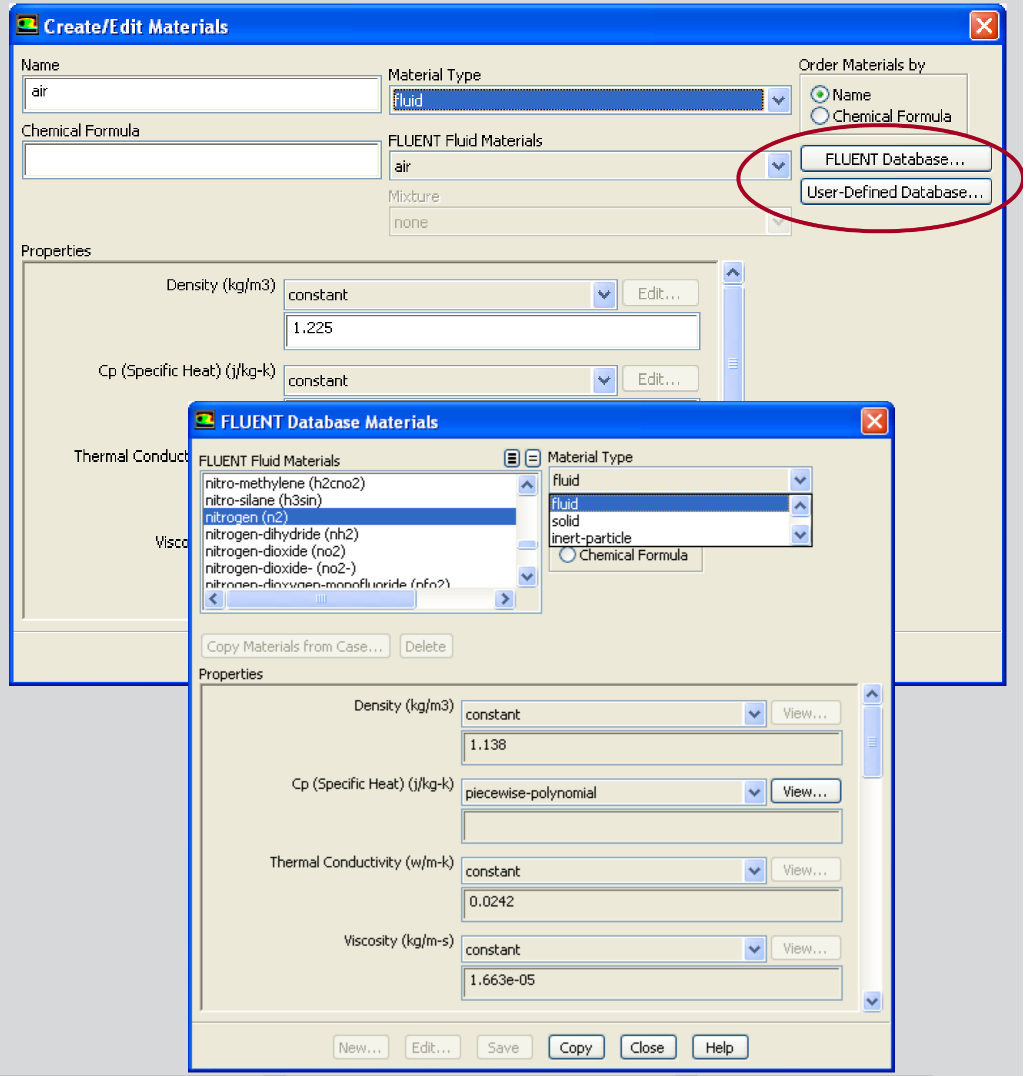
- **Select Materials in the Project Setup tree**
- **Or Define → Materials... in the Menu bar**




- **Material properties can be customized as function of temperature, mass fraction or pressure (density)**
  - Use of other solution variable(s) requires a User-Defined Function (UDF)

# Materials Databases

- **FLUENT materials database**
  - Provides access to a number of pre-defined fluid, solid and mixture materials
  - 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



# Fluid Density

- **For incompressible flow with $\rho$ = constant**
  - Select constant for density
- **Ideal gas properties**
  - Incompressible flow, $\rho = f(T)$
    - Polynomial or piecewise-polynomial
    - Incompressible ideal gas law ($\rho = p_{operating}/RT$)
      - Set $p_{operating}$ close to the mean pressure in the problem ➔ see Slide 8
  - Compressible flow, $\rho = f(p,T)$
    - Use ideal-gas for density ($\rho = p_{absolute}/RT$)
      - For low-Mach-number flows, set $p_{operating}$ 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)

# Fluid Density – Additional Options

- **Real gas properties**
  - Redlich-Kwong (gas and supercritical fluid regimes)
  - Aungier-Redlich-Kwong (gas and supercritical fluid regimes)
  - Soave-Redlich-Kwong (gas and supercritical fluid regimes)
  - Peng-Robinson model (gas and supercritical fluid regimes)
  - NIST properties
    - TUI : define/user-defined/real-gas-models
    - Liquid, gas and supercritical fluid regimes
      - But not applicable when temperature and pressure are such that the fluid state falls inside the vapor dome
- **Compressible liquids**
  - The density variation is specified by a user-defined density function
    - See User's Guide for details

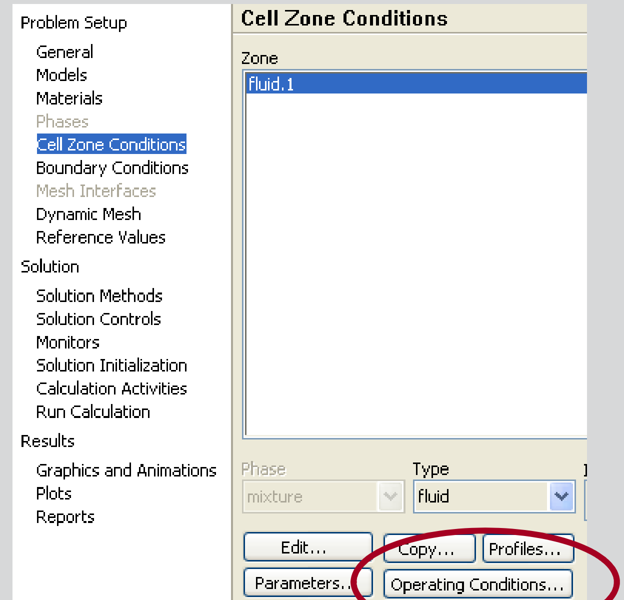
# Options for Defining Common Properties

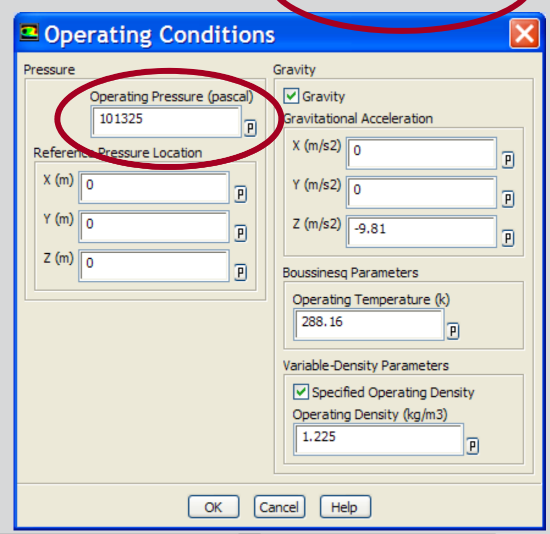
| Density | Viscosity |
|---|---|
| – Constant<br>– Incompressible Ideal Gas<br>– Ideal Gas<br>– Real Gas (5 Built-in Models)<br>– Temperature Dependent[1]<br>– Boussinesq<br>– User-defined | – Constant<br>– Temperature Dependent[1]<br>– Sutherland<br>– Power Law<br>– Kinetic Theory<br>– Non-Newtonian (4 Built-in Models)<br>– User-defined |
| **Thermal Conductivity** | **Specific Heat** |
| – Constant<br>– Temperature Dependent[1]<br>– Kinetic Theory<br>– User-defined | – Constant<br>– Temperature Dependent[1]<br>– User-defined |

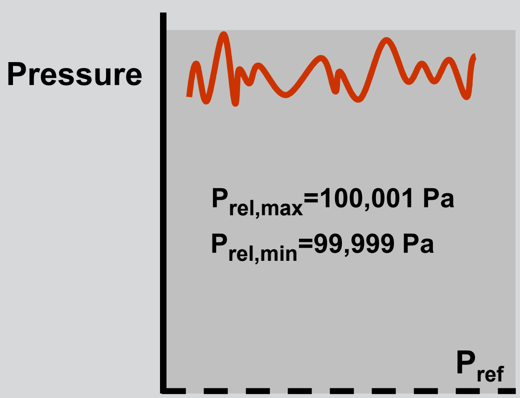
[1] **Temperature Dependent options include definition of properties as piecewise linear , polynomial or piecewise polynomial functions temperature**

# Operating Pressure

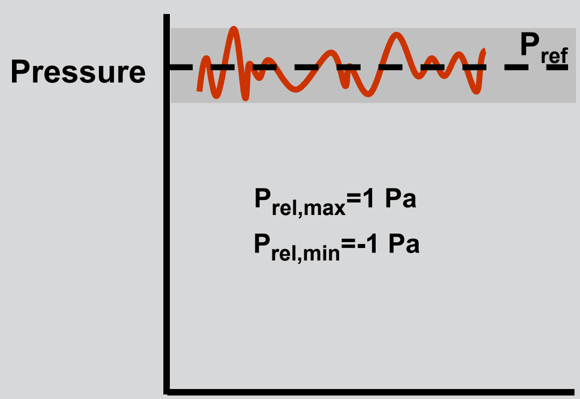
- Represents the absolute pressure datum from which all relative pressures are measured
  - $P_{absolute} = P_{operating} + P_{relative}$
  - – Pressures specified at boundary conditions and initial conditions are relative to the Operating Pressure
- Used to avoid problems with round-off errors which occur when the dynamic pressure differences in a fluid are small compared to the absolute pressure level



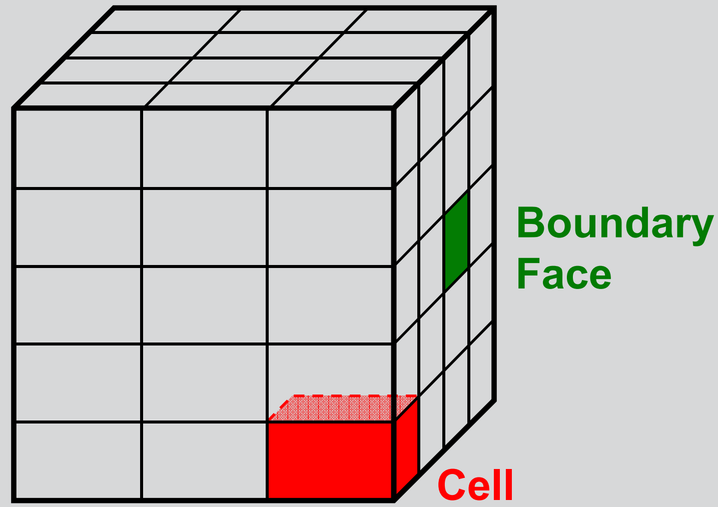

Ex. 1: $P_{operating}$= 0 Pa




Ex. 2: $P_{operating}$= 100,000 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**



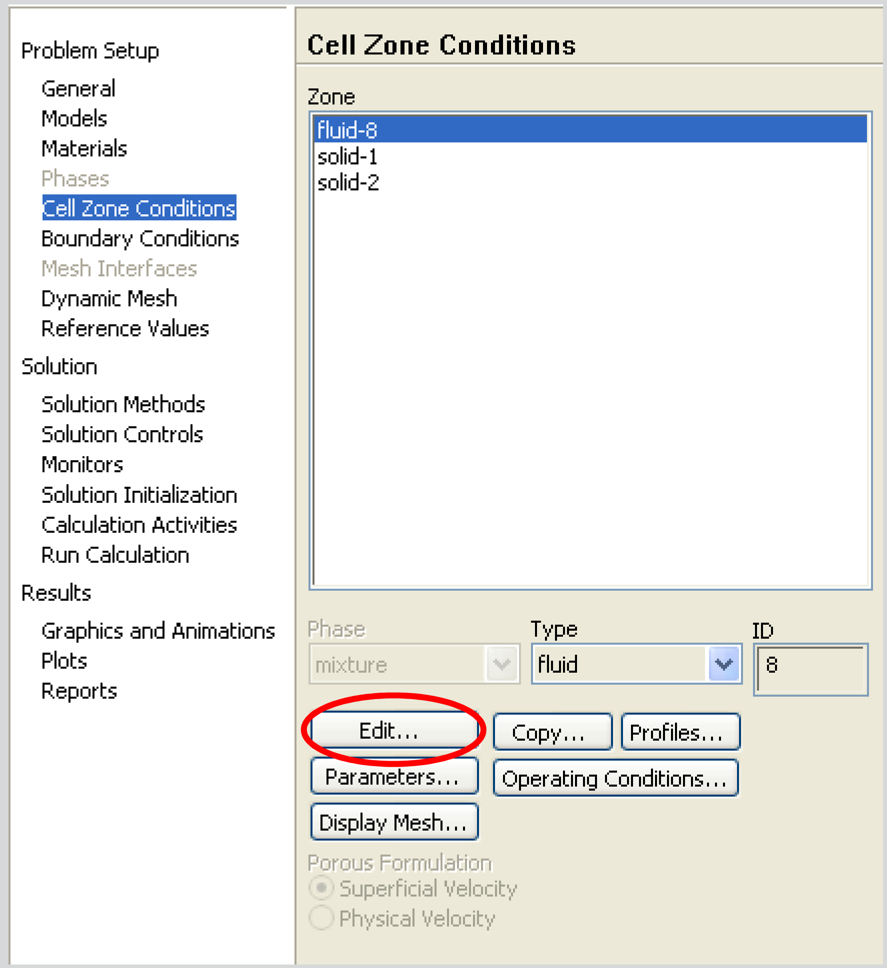
Simple 3D mesh

**Cell zone conditions** are applied to all cell zones

**Boundary conditions** are applied to all boundary zones

# Opening the Cell Zone Conditions Panel

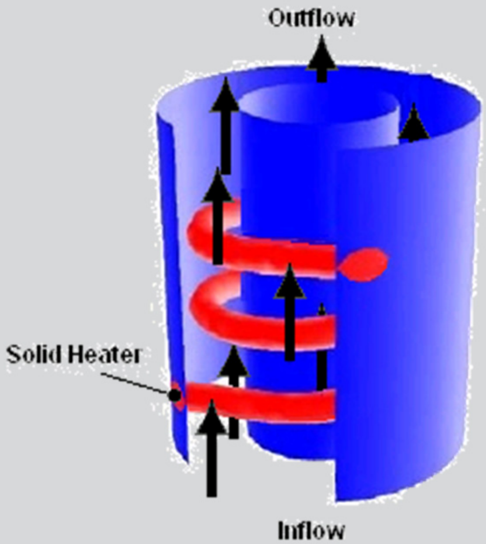
- In the Problem Setup tree, select Cell Zone Conditions
- A list of all cell zones will appear
- Select the zone of interest and click Edit

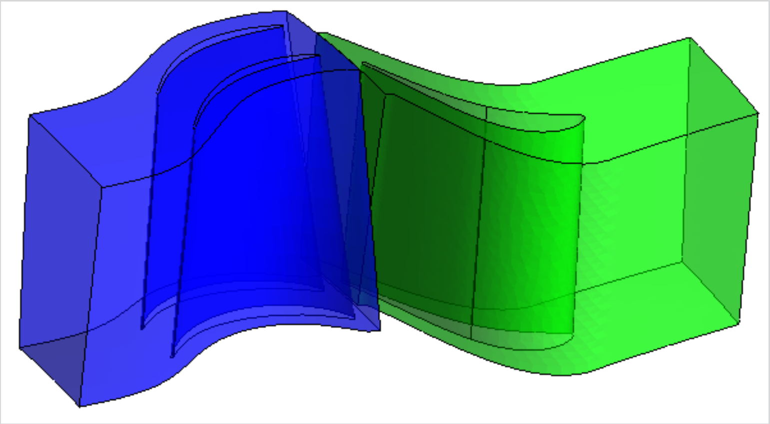


# Cell Zones

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



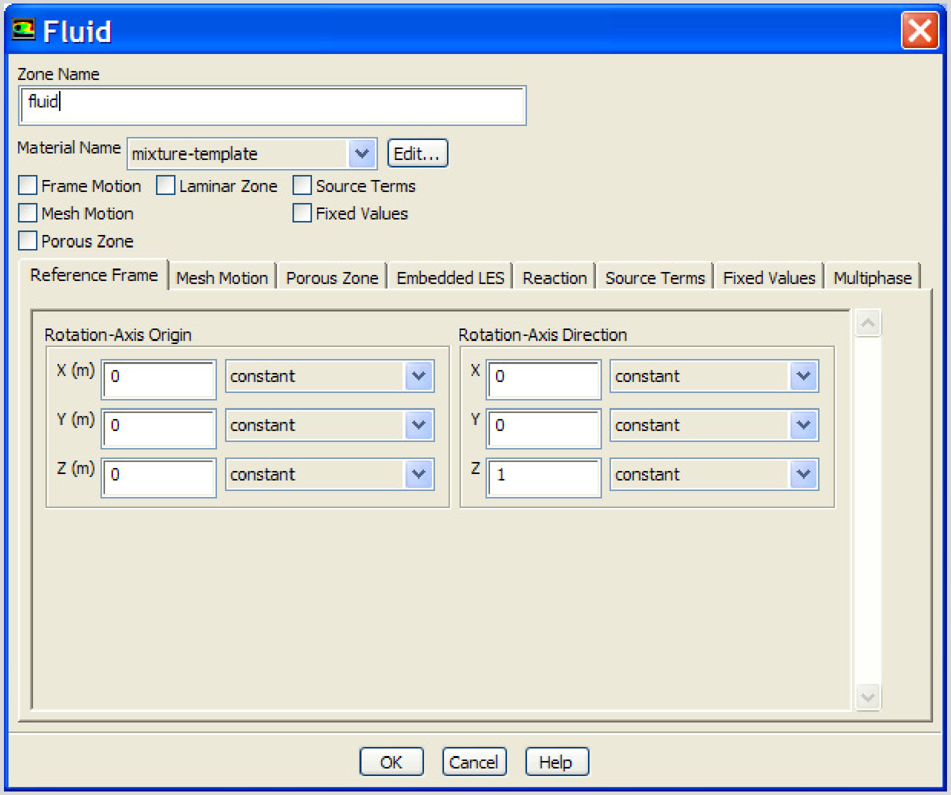



e.g. A simulation of a copper heating coil in water will require a fluid zone and a solid zone Using water properties, the equations of flow and heat transfer will be solved in the fluid zone Using copper properties, only the heat transfer equation will be solved in the solid zone

e.g. To account for rotational motion, the rotor is placed in a rotating domain. The rotor fluid zone will use equations in the rotating frame of reference. The stator fluid zone will use equations in the stationary frame of reference

# Cell Zone Definition – Fluid

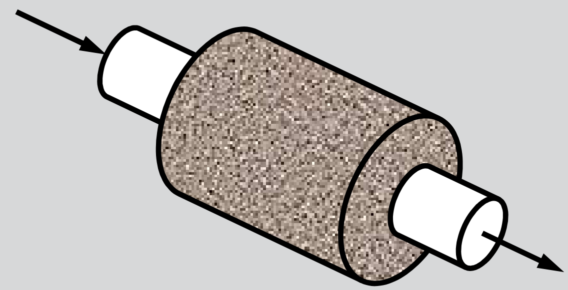
- **Fluid material selection is required**
  - For multiple species or multiphase flows, the material is not shown Instead, the fluid zone consists of the mixture of the phases




- **Optional inputs**
  - Frame/Mesh Motion
  - Porous region
  - Source terms
  - Laminar region
  - Fixed Values

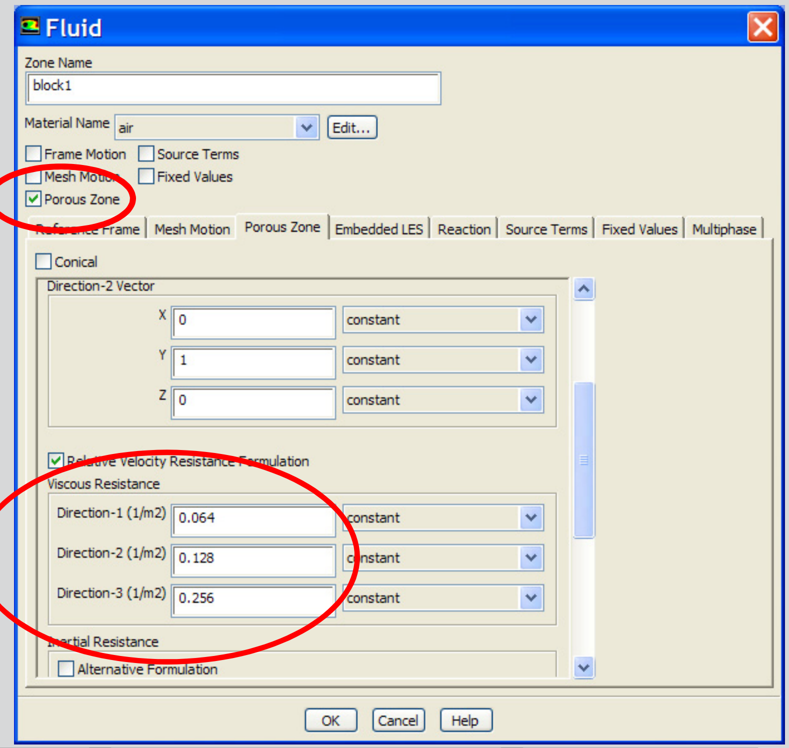
# Cell Zone Definition - Porous Media

- **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



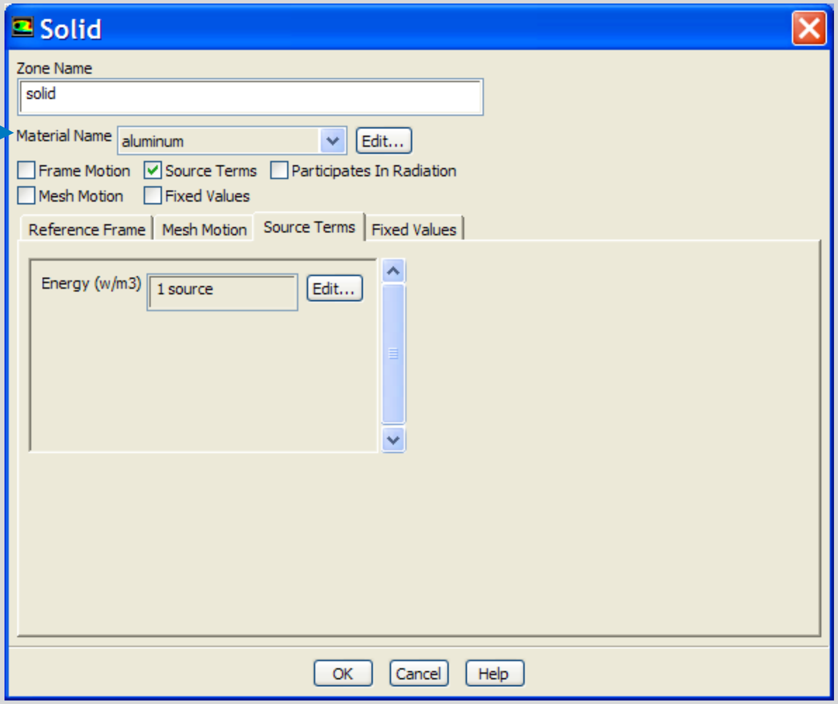
- **Used to model flow through porous media and other uniformly distributed flow resistances**
  - Packed beds
  - Filter papers
  - Perforated plates
  - Flow distributors
  - Tube banks

- **Inputs are directional viscous and inertial resistance coefficients**



# 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
  - Rotation axis must be specified if the solid zone is rotating or if rotationally periodic boundaries are adjacent to the solid zone



# 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

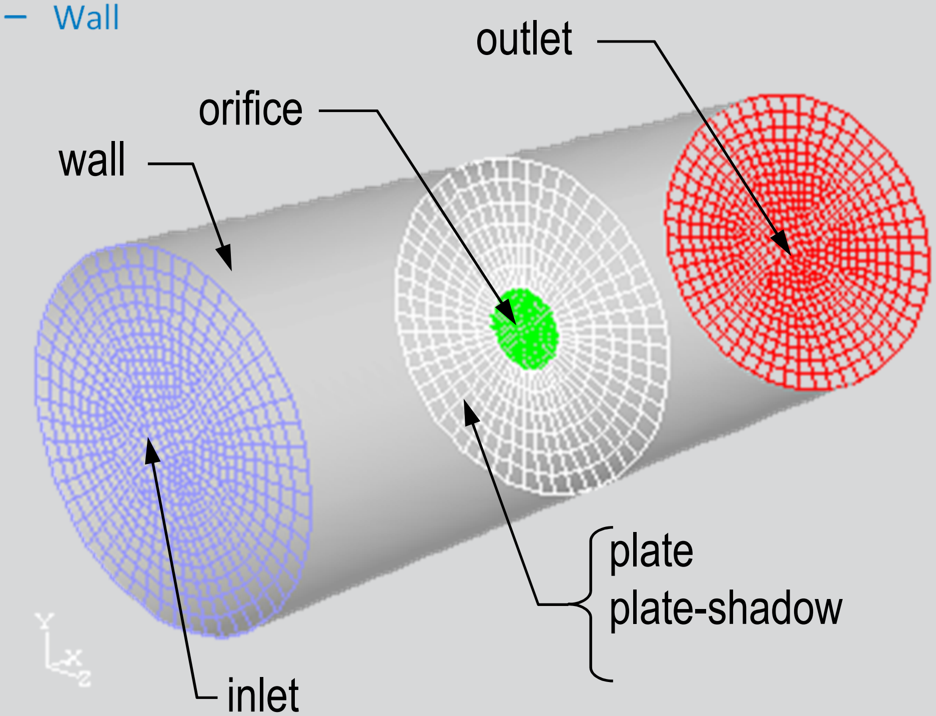
- **Defining boundary conditions involves:**
  - Identifying types (e.g., inlets, walls, symmetry)
  - Identifying location
  - Supplying required data depending on type, location and physical model

- **Choice depends on:**
  - Geometry
  - Availability of data
  - Numerical considerations

# 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



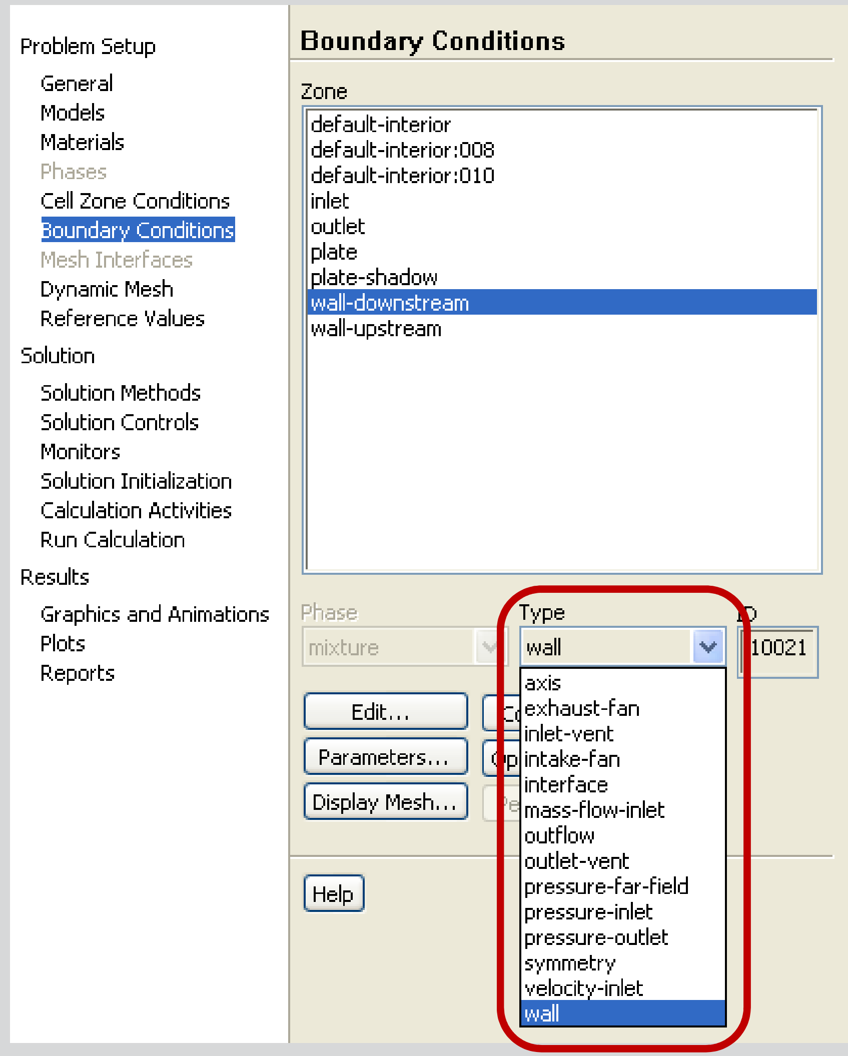

  

ANSYS

# Changing Boundary Condition 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

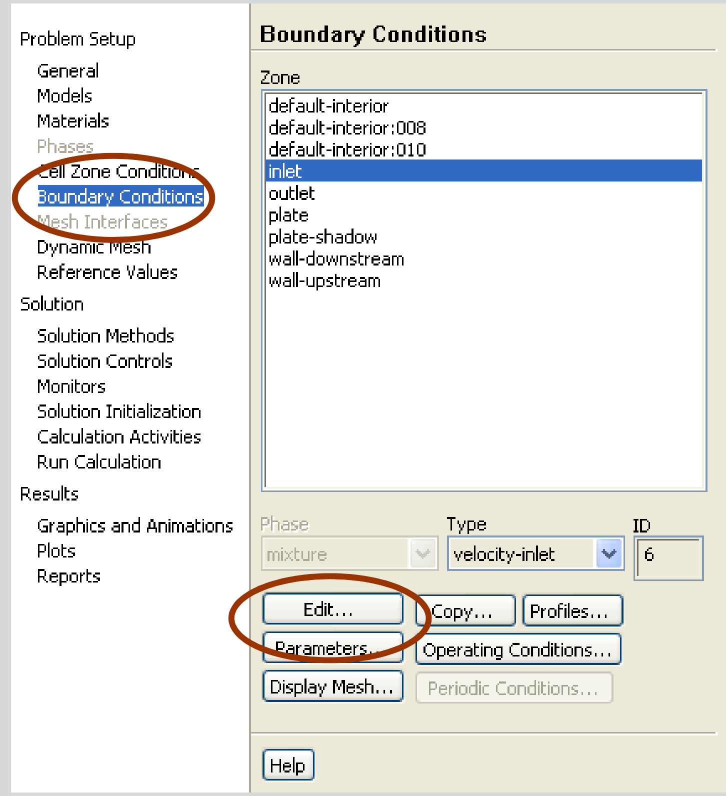




# Setting Boundary Condition Data

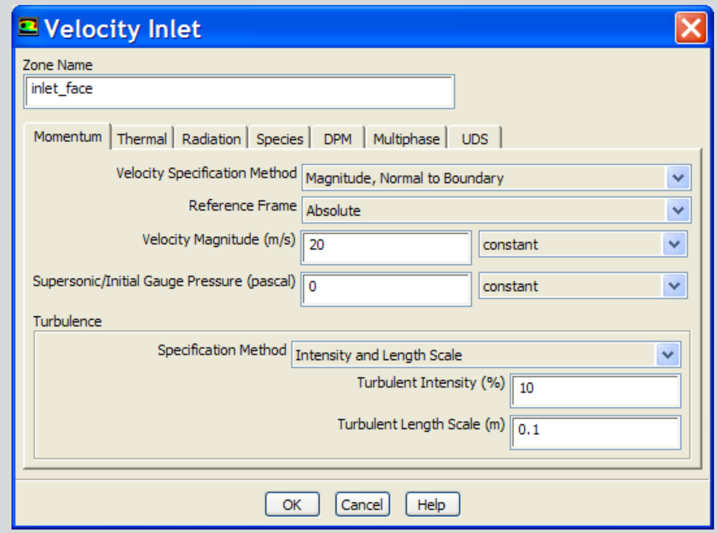
- **Explicitly assign data in BC panels**
  - To set boundary conditions for particular zone:
    - Select Boundary Conditions in the project tree
    - Choose the boundary name in the Zone list
    - Click the Edit... button
  - Boundary condition data can be copied from one zone to another




- **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
    - See Lecture 11 for details of UDFs
    - See Appendix for details of using profiles

# 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



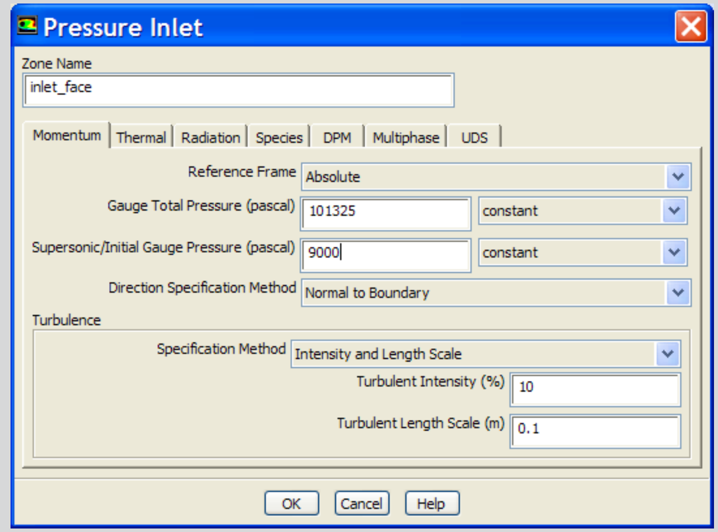

> - Applies a uniform velocity profile at the boundary ,unless UDF or profile is used
> - Velocity Magnitude input can be negative, implying that you can prescribe the exit velocity

- **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**

**Incompressible:**

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

**Compressible:**

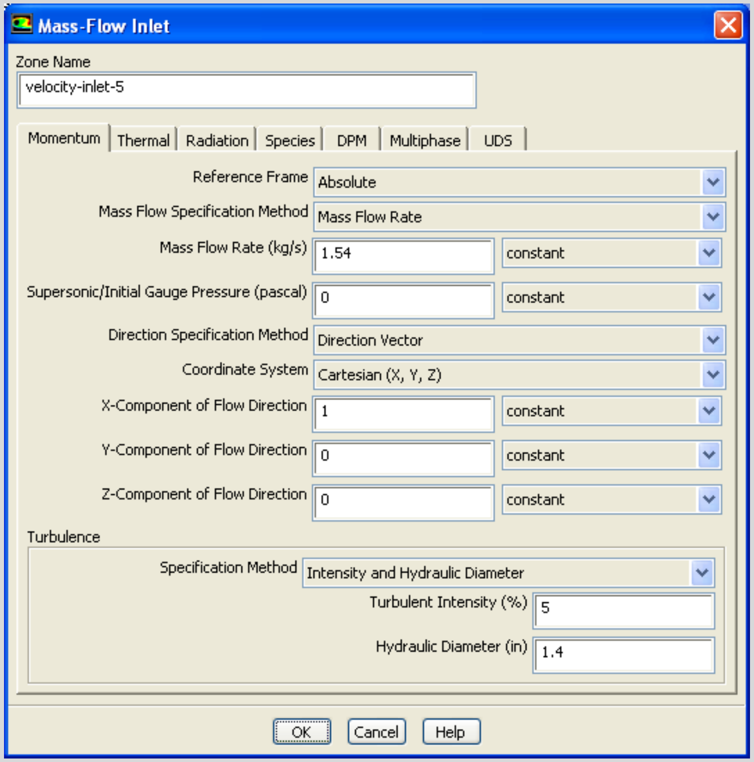
$$p_{\text{total,abs}} = p_{\text{static,abs}} \left(1 + \frac{k-1}{2} M^2\right)^{\frac{k}{k-1}}$$

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

# Mass Flow Inlet

- **<u>Required information</u>**
  - 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

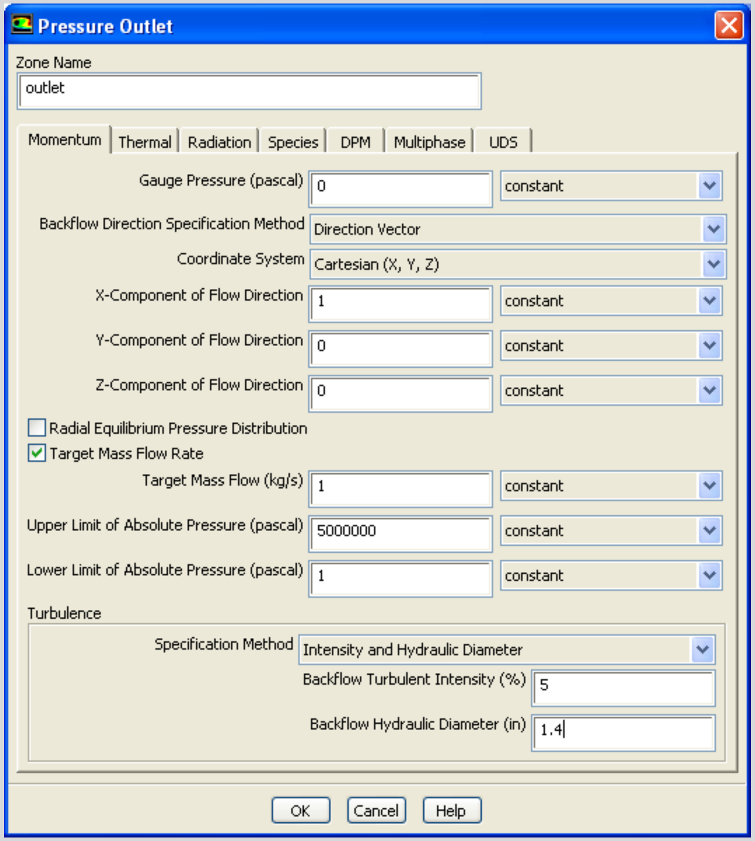



- **Mass flow inlets are intended for compressible flows; however, they can be used for incompressible flows**

# Pressure Outlet

- **Required information**
  - 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)**

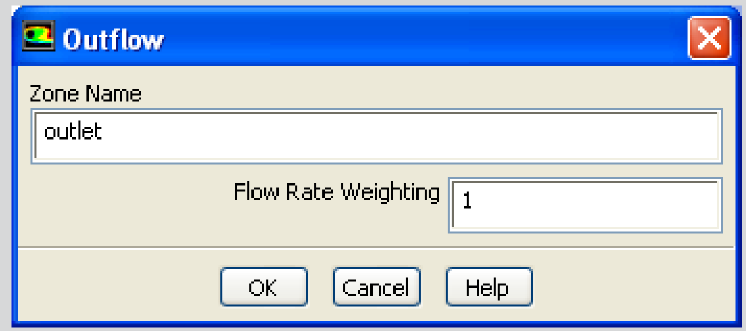
- Can be used as a “free” boundary in an external or unconfined flow
- Target Mass Flow Rate Option can be applied




- **Suitable for compressible and incompressible flows**
  - For ideal gas (compressible) flow, non-reflecting outlet boundary conditions (NRBC) are available with Density-Based Coupled solvers

# 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




- **<u>Limitations :</u>**
  - **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**

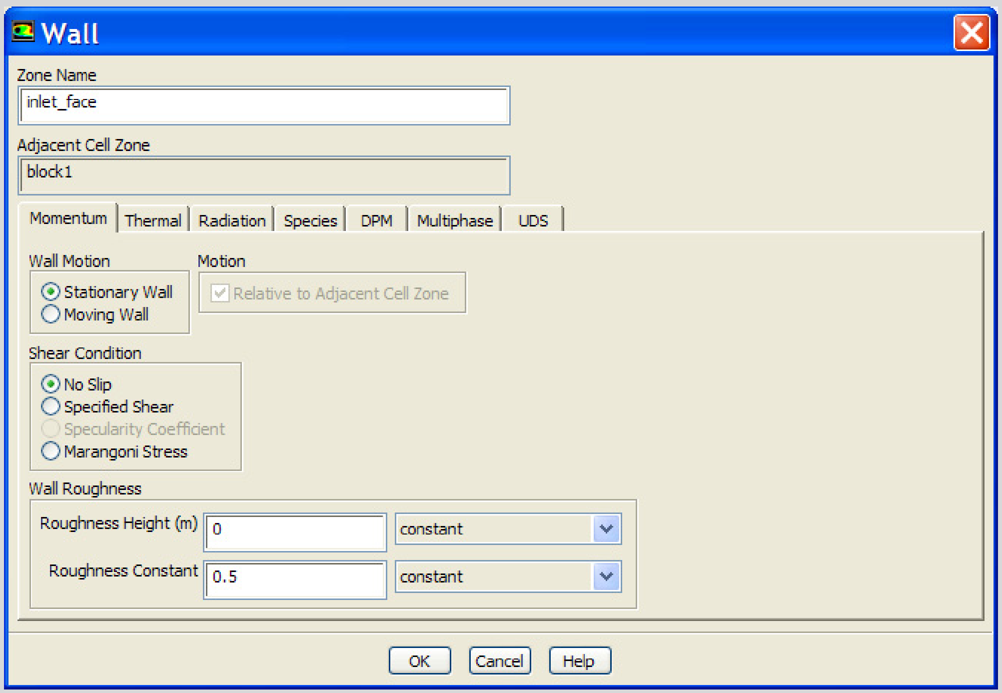
- **The outflow boundary is intended for use with incompressible flows**

- **Poor rate of convergence when backflow occurs during iterations**
  - Cannot be used if backflow is expected in the final solution

# Wall Boundaries

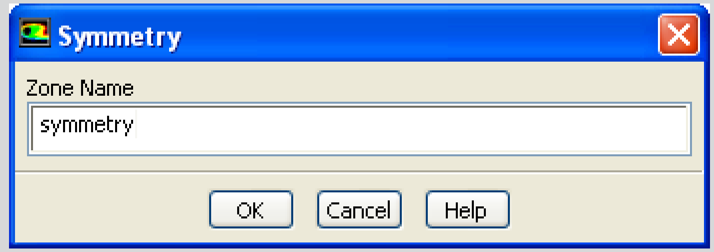
- **In viscous flows, no-slip conditions are applied at walls**
  - Shear stress can be applied
  - Wall roughness can be defined for turbulent flows
    - Modification of the Logarithmic Standard Wall Function

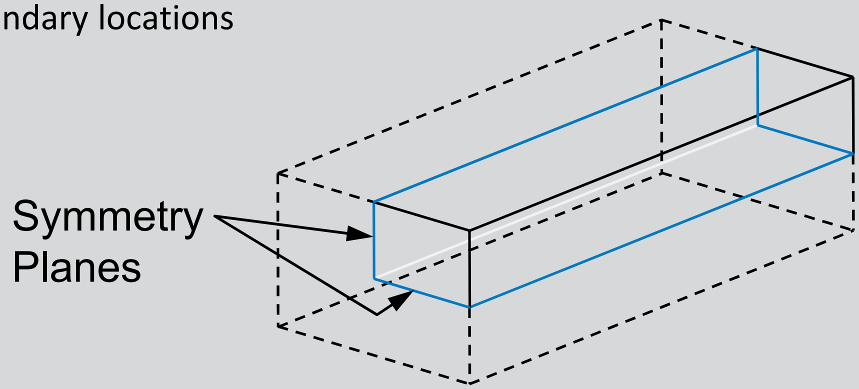



- **More information in moving zone and heat transfer presentation**

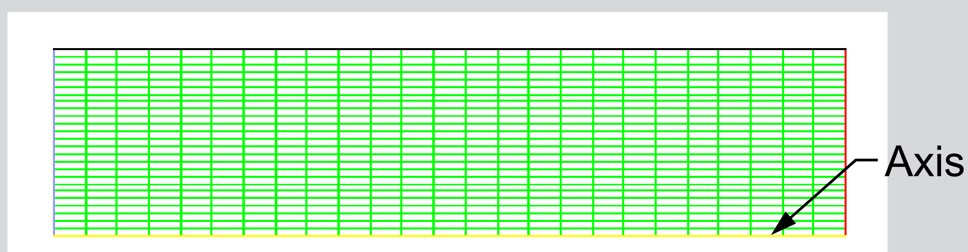
# Symmetry and Axis Boundaries

- **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



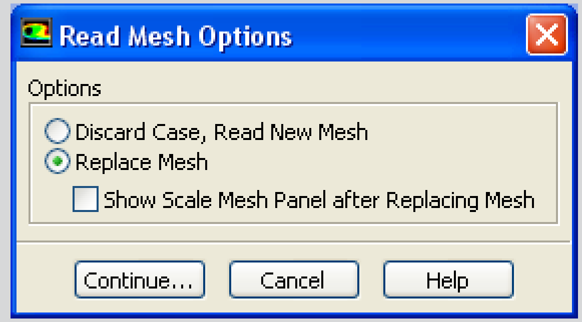




- **Axis Boundary**
  - Used at the center line for 2d axisymmetric problems
  - No user inputs required
  - The axis boundary must coincide with the x-axis

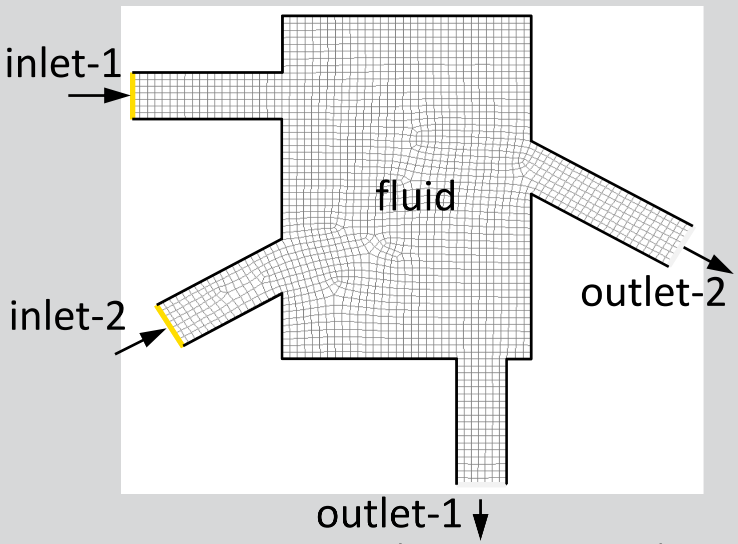


# Case Setup Replication

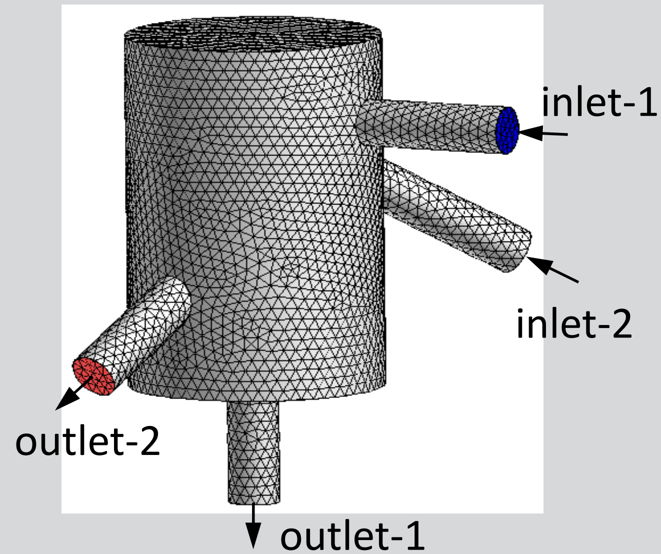
- **To replicate a case setup:**
  - Open the actual case file and go to File/Read/Mesh
    - Choose the option : Replace Mesh




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




2D Flow Domain (approximation)




Actual 3D Flow Domain

**The text user interface (TUI) commands `/file/write-settings` and `/file/read-settings` can also be used for the same purpose (see Appendix)**

# General Guidelines

- **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**
  - Introduction of an error

# Specifying Well Posed Boundary Conditions

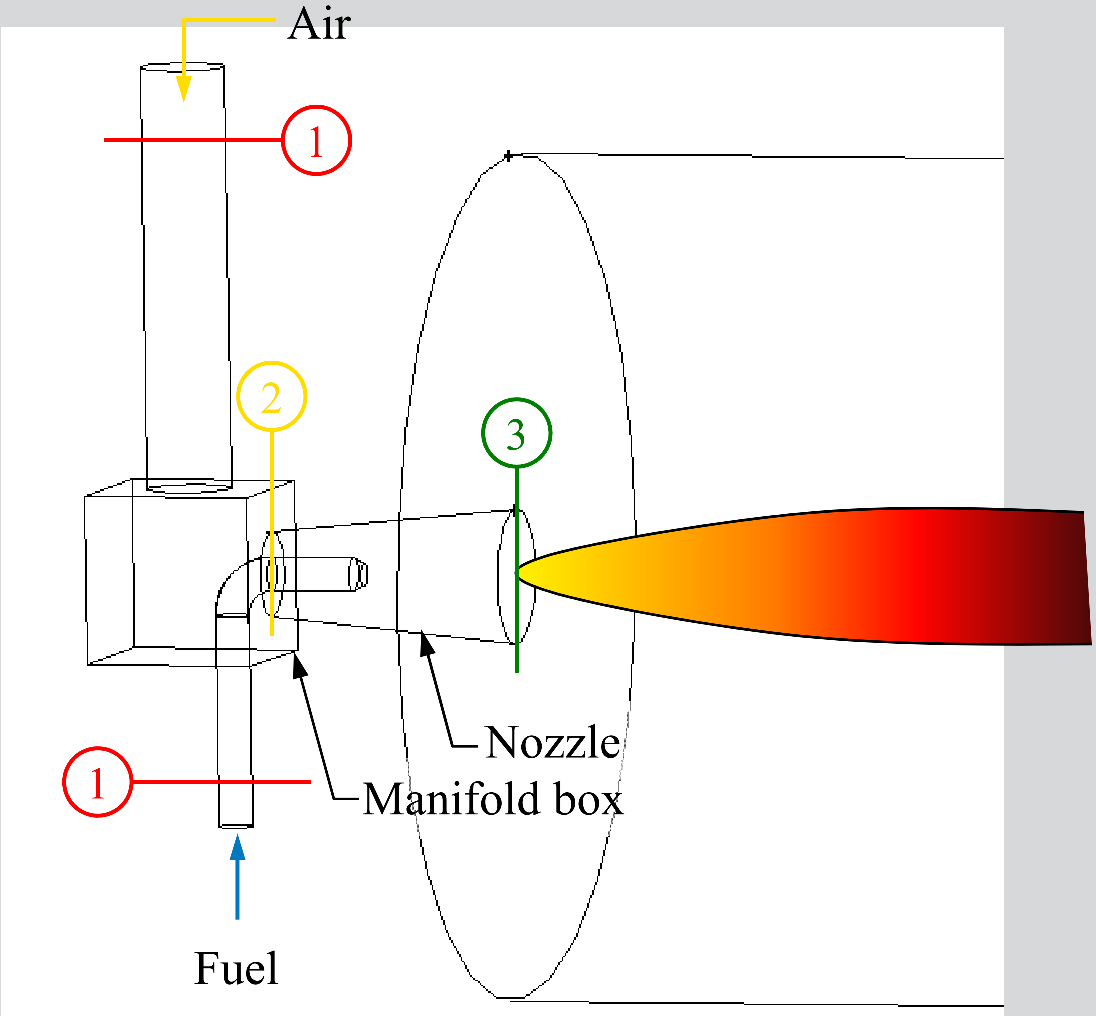
- Consider the following case which contains separate air and fuel supply pipes

- 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

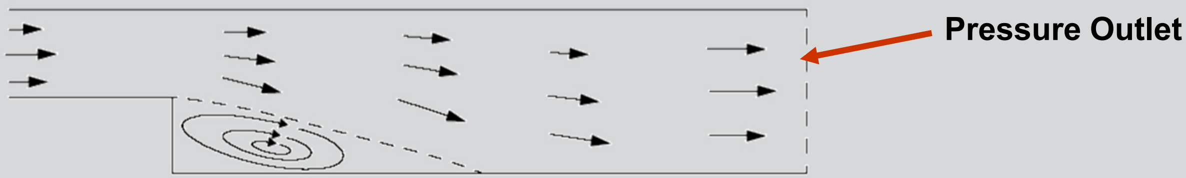
  2 Nozzle inlet plane
    - Requires accurate velocity profile data for the air and fuel

  3 Nozzle outlet plane
    - Requires accurate velocity profile data and accurate profile data for the mixture fractions of air and fuel

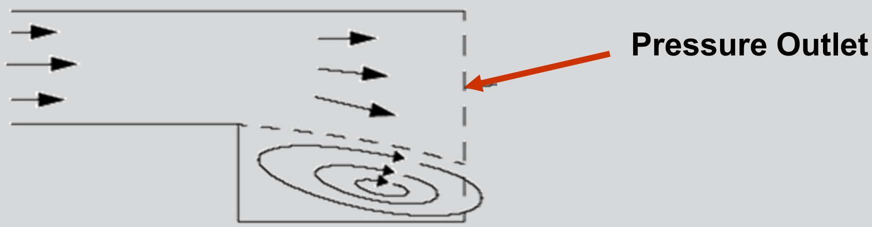


# Specifying Well Posed Boundary Conditions

- **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: Apply meaningful backflow conditions at pressure outlet for turbulence, temperature, species, … (difficult)




Introduction | Material Properties | Cell Zone Conditions | **Boundary Conditions** | Summary

# 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

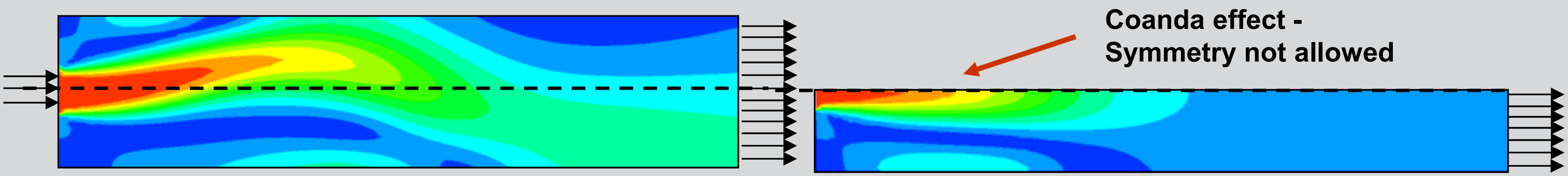
# Specifying Well Posed Boundary Conditions

- **Turbulence at Inlet:**
  - If you have no idea, you can use the following parameters:
    - Turbulence Intensity = 5%
    - Eddy 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 to become developed
  - Addressed in ***Lecture 7: Turbulence***

# Specifying Well Posed Boundary Conditions

- **Symmetry Plane and the Coanda Effect:**
  - Symmetric geometry does not necessarily mean symmetric flow
  - Example: *The Coanda effect*. Above a certain Reynolds number, a jet entering at the center of a symmetrical duct will tend to attach to one side wall

**Jet Instability**

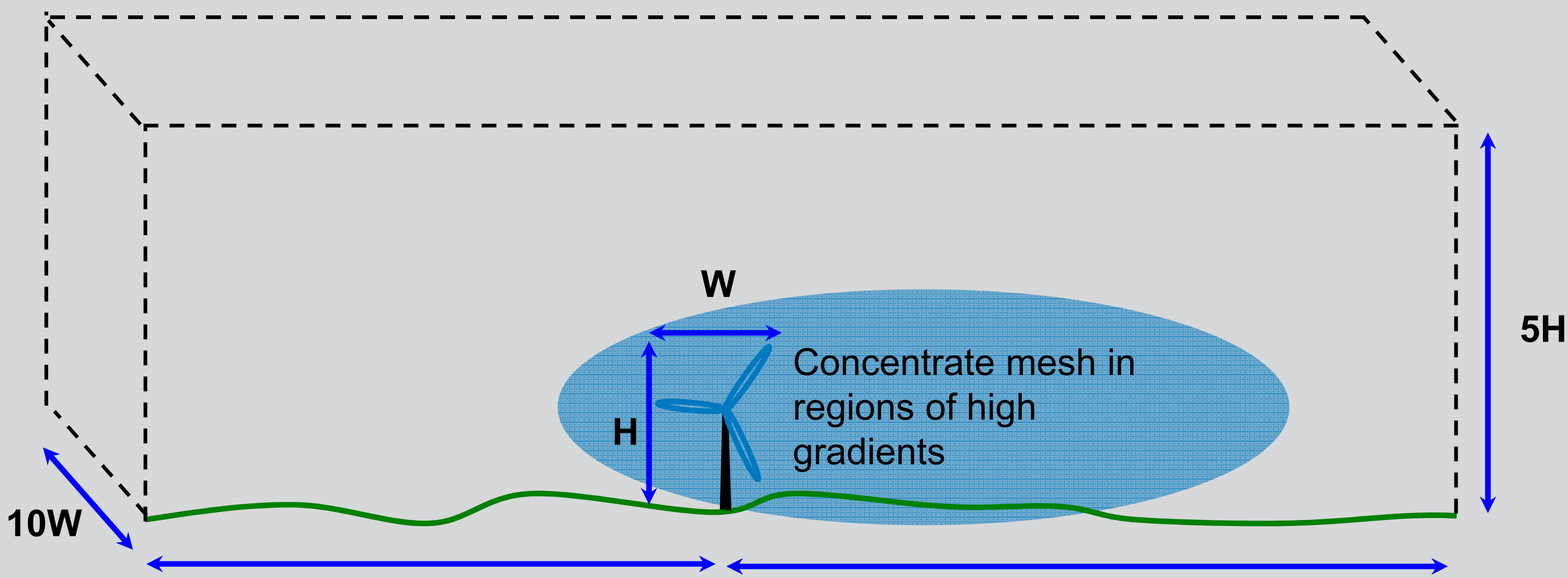



**No Symmetry Plane** **With Symmetry Plane**

**<u>Coanda Effect</u>**
-Coanda effect is the phenomenon in which a jet flow attaches itself to a nearby surface
- The proximity of the surface to the jet restricts the entrainment in that region. As flow accelerates, a pressure difference develops across the jet to balance the momentum transfer and the jet is deflected closer to the surface - eventually attaching to it

# Specifying Well Posed Boundary Conditions

- **External Flow**
  - In general, if the building 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 10 H 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



# Summary

- **Cell zones (or continuum 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 assign boundary conditions at external and internal boundaries**
  - Many different boundary types exist for prescribing boundary information

- **Locations and types of boundary condition are extremely important for good convergence and accurate results**

# Appendix

# Auxiliary Operations

- The definition of material properties, boundary conditions and cell zone conditions is a fundamental part of setting up any CFD simulation in FLUENT
- There are some additional auxiliary operations that may be generally very useful when setting up a simulation in FLUENT
  - Polyhedral mesh conversion
  - Text User Interface (TUI)
  - Journal files
  - Reading and writing data profiles

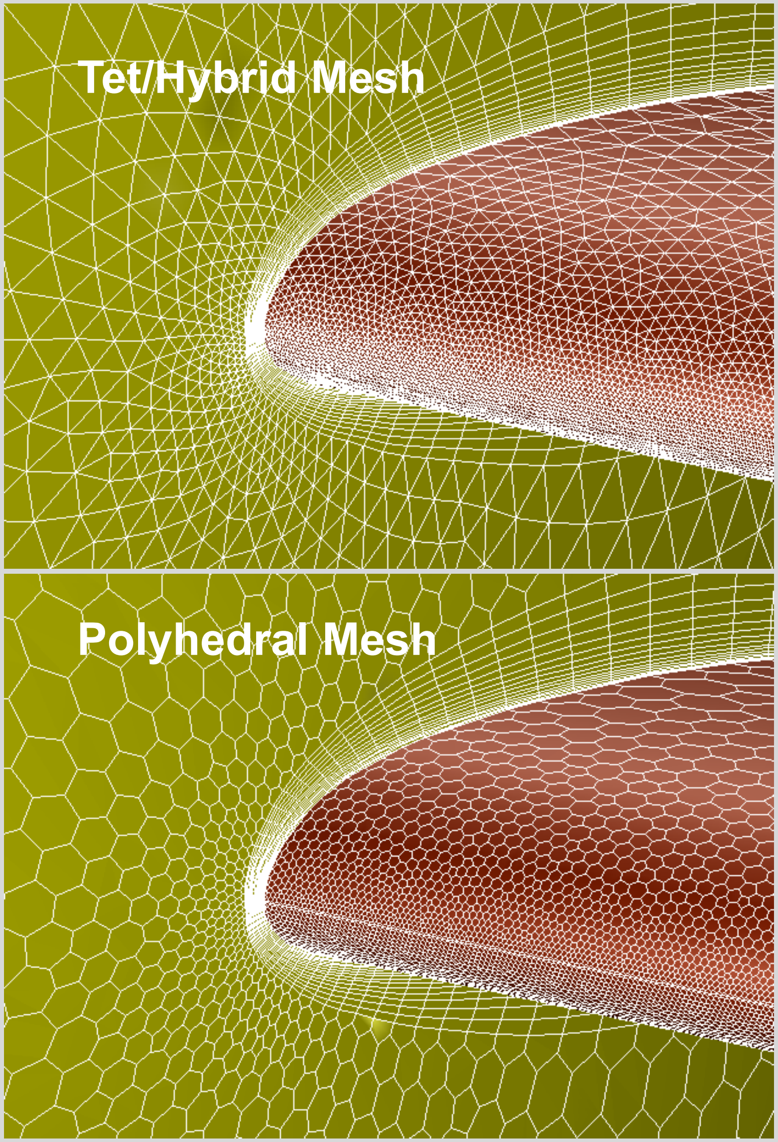
# Polyhedral Mesh Conversion

- A tetrahedral or hybrid grid can be converted to polyhedra in the FLUENT GUI (not in ANSYS Meshing).
  - Generate a tetrahedral mesh then convert inside FLUENT.
  - Advantages
    - Improved mesh quality.
    - Can reduce cell count significantly.
    - User has control of the conversion process.
  - Disadvantages:
    - Cannot be adapted or converted again.
    - Cannot use tools such as smooth, swap, merge and extrude to modify the mesh.

- Two conversion options are available in the Mesh menu:

  **`Mesh > Polyhedra > Convert Domain`**
  - Convert all cells in the domain (except hex cells) to polyhedra
    - Cannot convert adapted meshes with hanging nodes
  - Convert only highly skewed cells to polyhedra

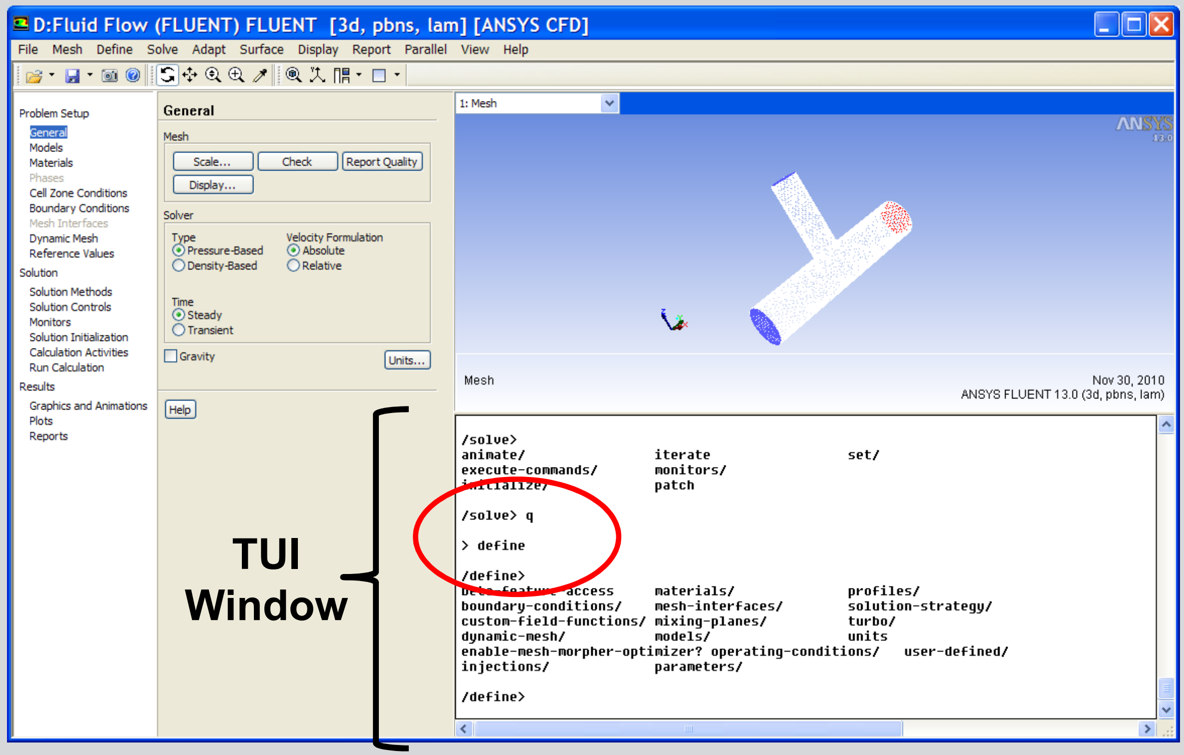
  **`Mesh > Polyhedra > Convert Skewed Cells`**



# Text User Interface

- Most GUI commands have a corresponding TUI command.
  - Press the **Enter** key to display the command set at the current level.
  - **q** moves up one level.
  - Some advanced commands are only available through the TUI.




- The TUI offers many valuable benefits:
  - Journal (text) files can be constructed to automate repetitive tasks.
  - FLUENT can be run in batch mode, with TUI journal scripts set to automate the loading / modification / solver execution and postprocessing.
  - Very complex models can be set using a spreadsheet to generate the TUI commands.

# Sample FLUENT Journal

- A journal file is a text file which contains TUI commands which FLUENT will execute sequentially.

- Note that the FLUENT TUI accepts abbreviations of the commands for example,
  - **`rcd`** Reads case and data files
  - **`wcd`** Writes case and data files

**Sample Journal File**

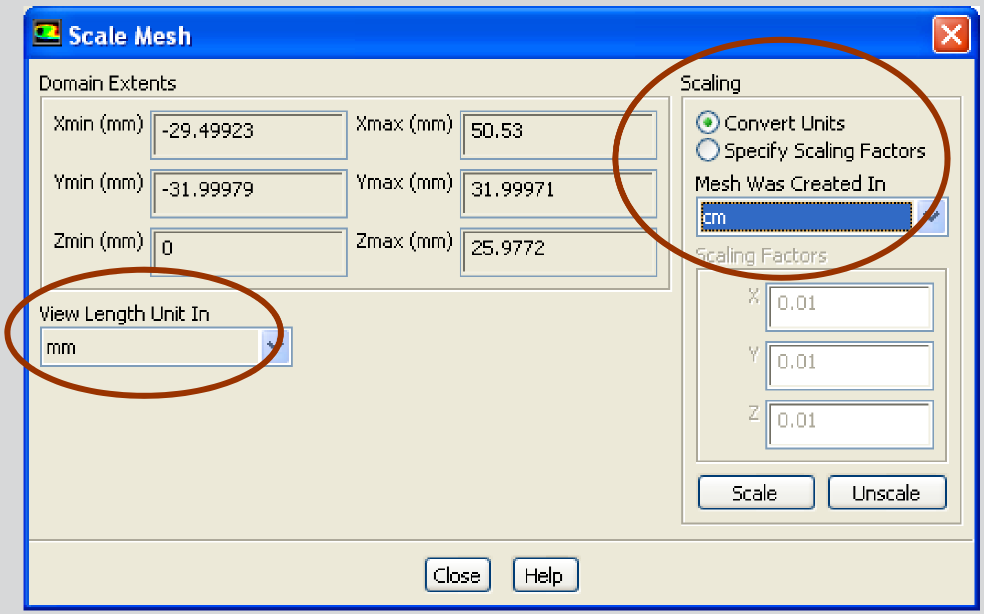
```
; Read case file
rc example.cas.gz
; Initialize the solution
/solve/initialize/initialize-flow
; Calculate 50 iterations
it 50
; Write data file
wd example50.dat.gz
; Calculate another 50 iterations
it 50
; Write another data file
wd example100.dat.gz
; Exit FLUENT
exit
yes
```

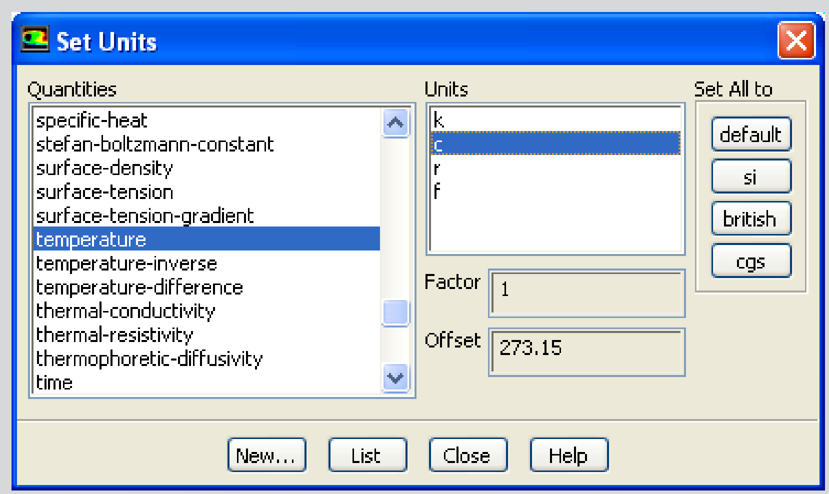
# Scaling the Mesh and Selecting Units

- When FLUENT reads a mesh file (.msh), all dimensions are assumed to be in units of meters.
  - If your model was not built in meters, then it must be scaled.
  - **Always verify that the domain extents are correct.**

- When importing a mesh under Workbench, the mesh does not need to be scaled; however, the units are set to the default MKS system.

- Any “mixed” units system can be used if desired.
  - By default, FLUENT uses the SI system of units (specifically, MKS system).
  - Any units can be specified in the Set Units panel, accessed from the top menu.



# Reordering and Modifying the Grid

- The grid can be reordered so that neighboring cells are near each other in the zones and in memory
  – Improves efficiency of memory access and reduces the bandwidth of the computation
  – Reordering can be performed for the entire domain or specific cell zones.

    **`Mesh > Reorder > Domain`** **`Mesh > Reorder > Zones`**

  – The bandwidth of each partition in the grid can be printed for reference.

    **`Mesh > Reorder > Domain`**

- The face/cell zones can also be modified by the following operations in the Grid menu:
  – Separation and merge of zones
  – Fusing of cell zones with merge of duplicate faces and nodes
  – Translate, rotate, reflect face or cell zones
  – Extrusion of face zones to extend the domain
  – Replace a cell zone with another or delete it
  – Activate and Deactivate cell zones

# Profile Data and Solution Data Interpolation

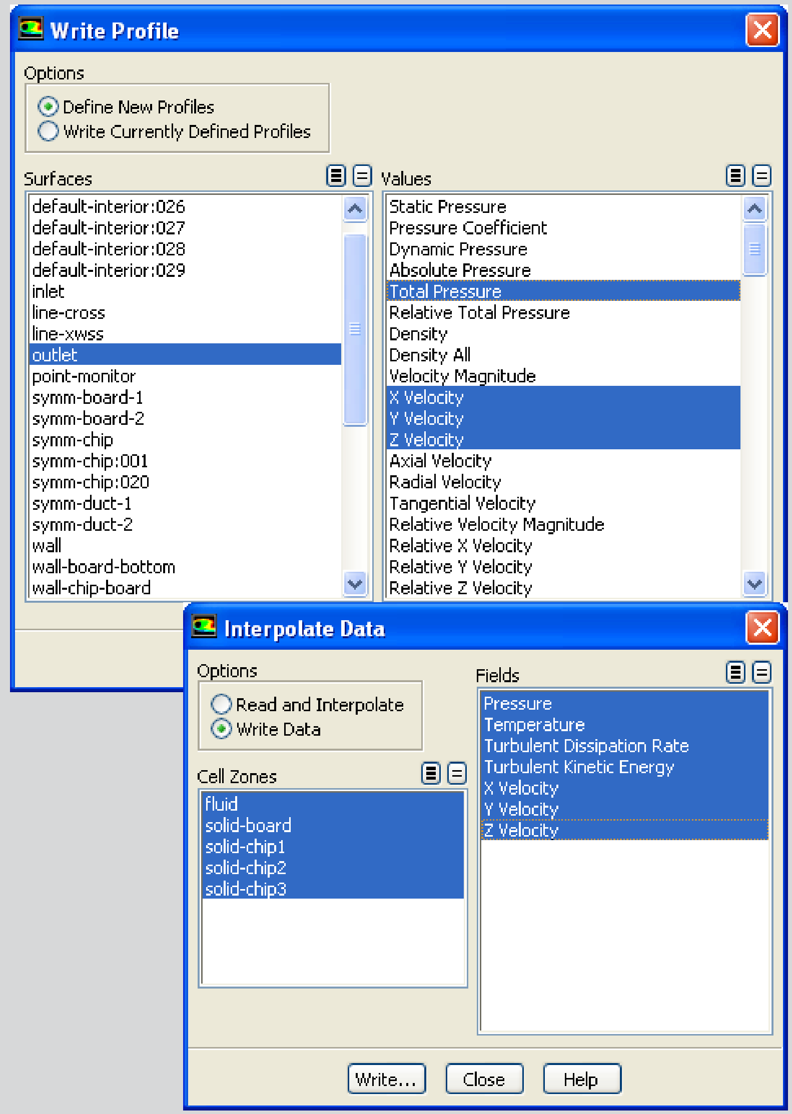
- FLUENT allows interpolation of selected variable data on both face zones and cell zones by using profile files and data interpolation files, respectively.
  - For example, a velocity profile from experimental data or previous FLUENT run at an inlet, or a solution interpolated from a coarse mesh to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

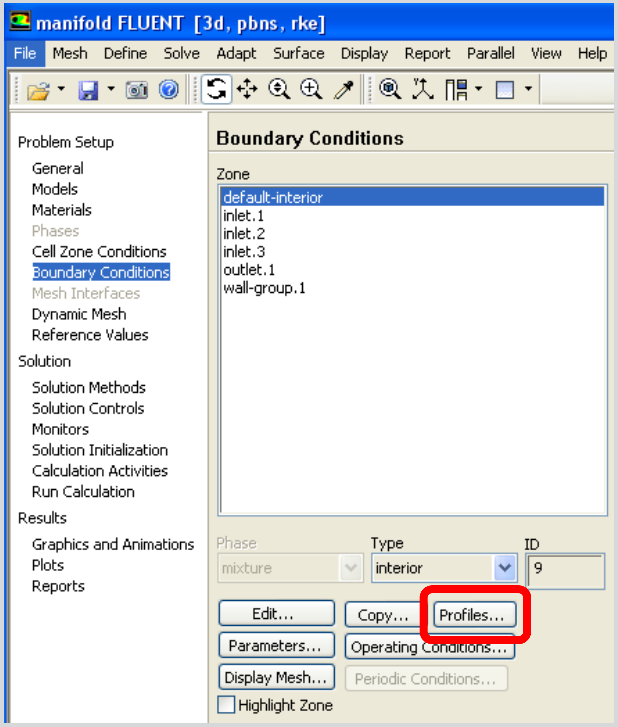
- Similarly, interpolation files contain discrete data for selected field variables on particular cell zones to be written and read into FLUENT.

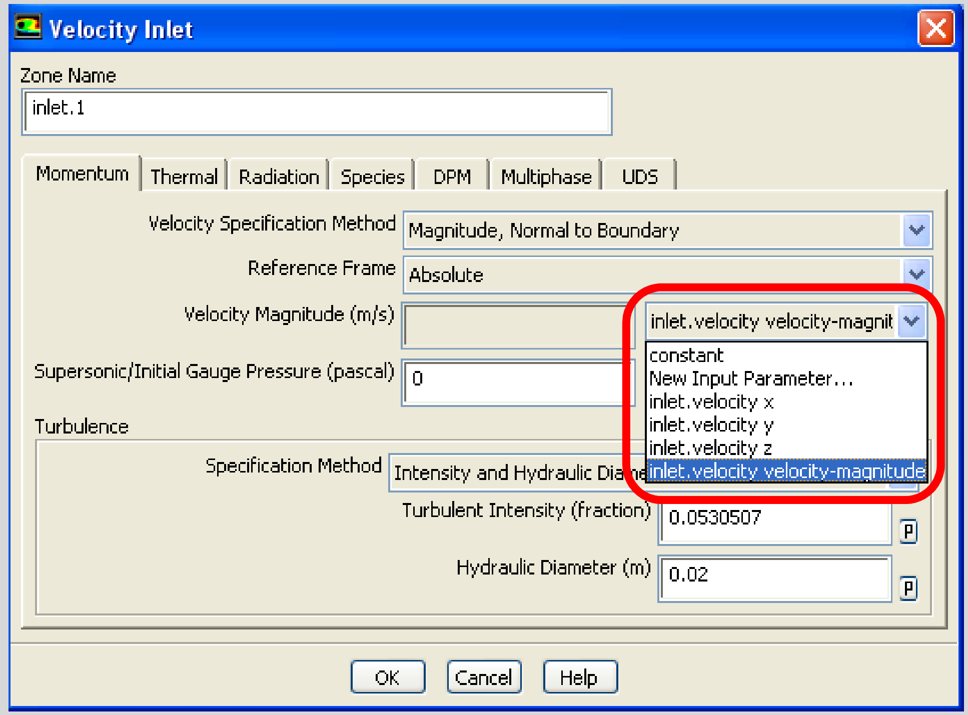
  **File > Interpolate**



# Profile Boundary Conditions



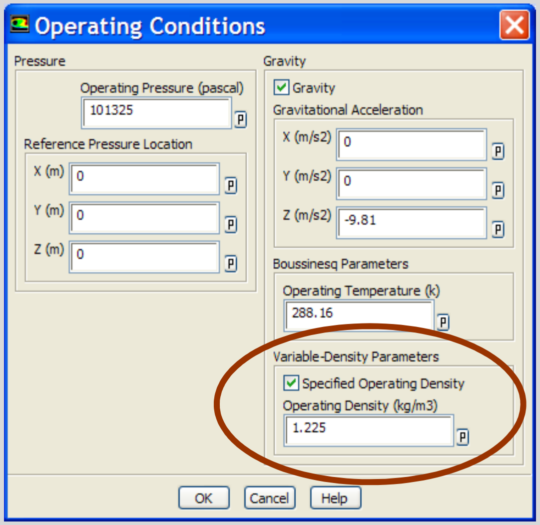

- Select Profiles in the Boundary Conditions panel (left figure)
- After reading the profile, open the panel for the boundary where it is to be applied
- Select the arrow and scroll down in the drop-down list until the desired profile is reached (right figure)
  – The first three items in the list will 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

# Buoyancy

- **Buoyancy Ref. Density**
  - The *Buoyancy Reference Density* is used to avoid round-off errors by solving at an offset level
  - The *Reference Pressure* is used to offset the operating pressure of the domain, while the *Buoyancy Reference Density* should be used to offset the hydrostatic pressure in the domain
    - The pressure solution is relative to $\rho_{ref}\, g\, h$, where $h$ is relative to the Reference Location
    - If $\rho_{ref}$ = the fluid density ($\rho$), then the solution becomes relative to the hydrostatic pressure, so when visualizing ***Pressure*** you only see the pressure that is driving the flow
  - ***Absolute Pressure*** always includes both the hydrostatic and reference pressures

$$P_{abs} = P_{reference} + P_{relative} + \rho_{ref}\, g\, h$$

  - For a non-buoyant flow a hydrostatic pressure does not exist



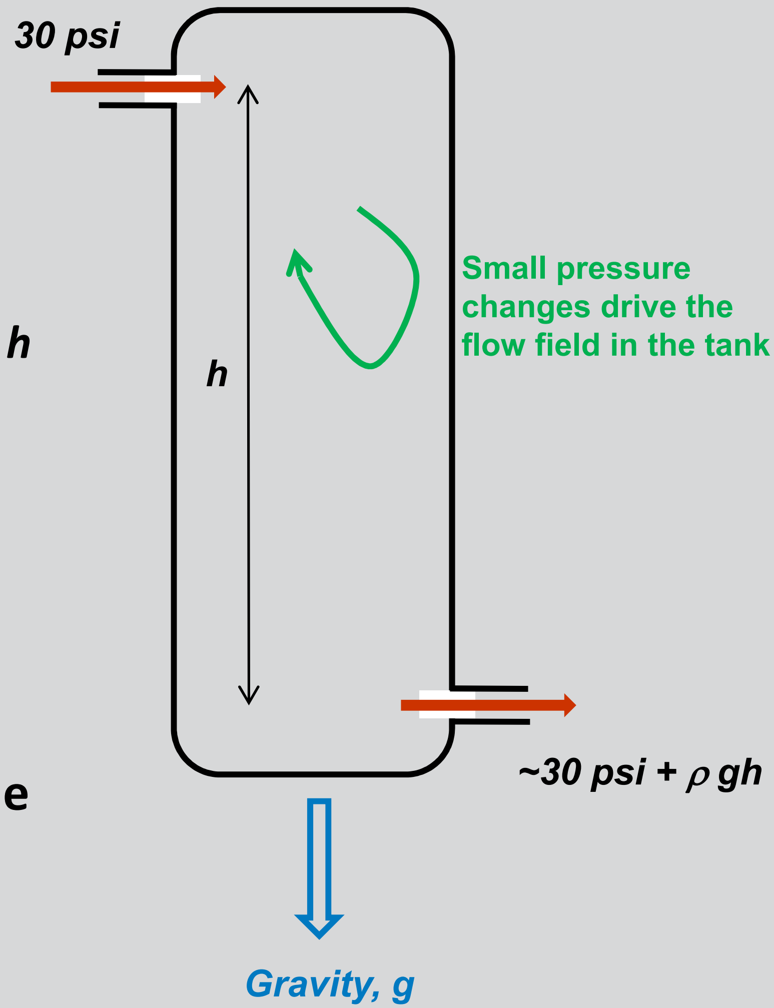

  

# Pressure and Buoyancy Example

- **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***



# 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

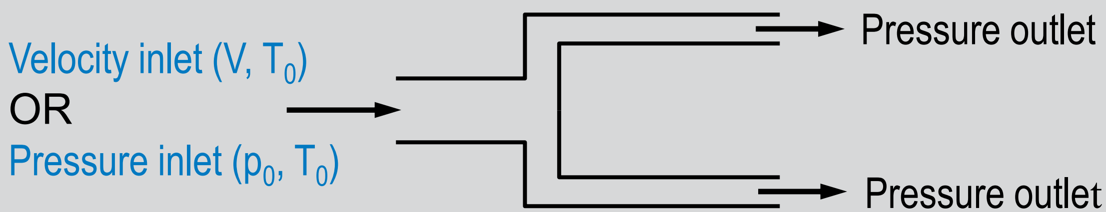
# Other Inlet / Outlet Boundary Conditions

- **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**
  - <u>Pressure outlets:</u> requires knowledge of downstream pressures; FLUENT calculates the fraction of total flow through each branch

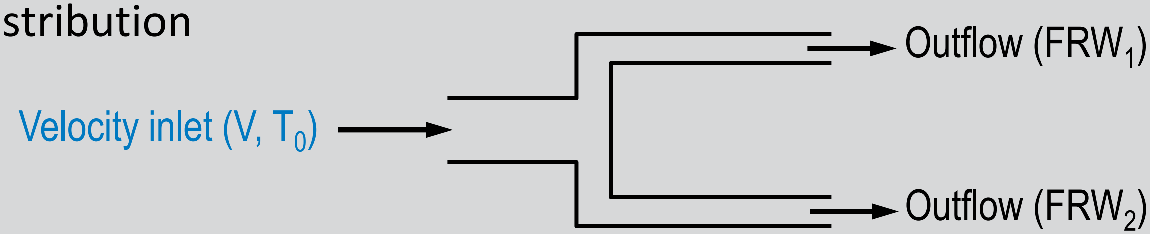



  - <u>Outflow:</u>
    - Mass flow rate fraction determined from Flow Rate Weighting (FRW) by

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

$0 <= \mathrm{FRW}_i <= 1$

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

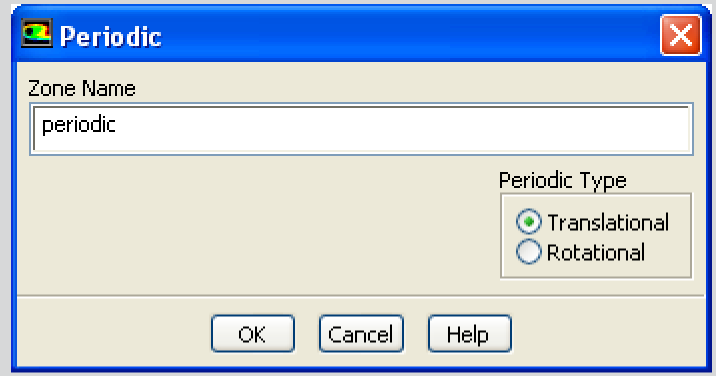


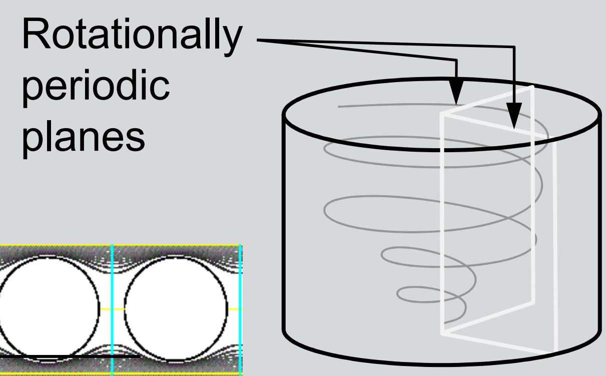
# Periodic Boundaries

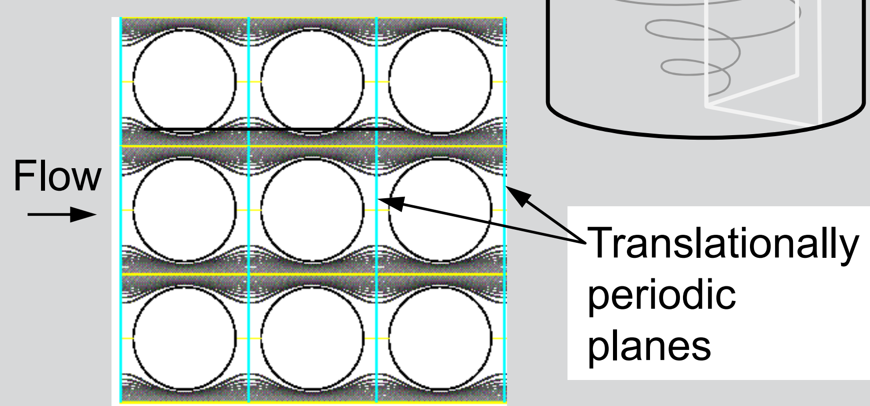
- **Used to reduce the overall mesh size**

- **Flow field and geometry must contain either rotational or translational periodicity**
  - Rotational periodicity
    - 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 boundaries can be either conformal or non-conformal**
  - **See next two slides**



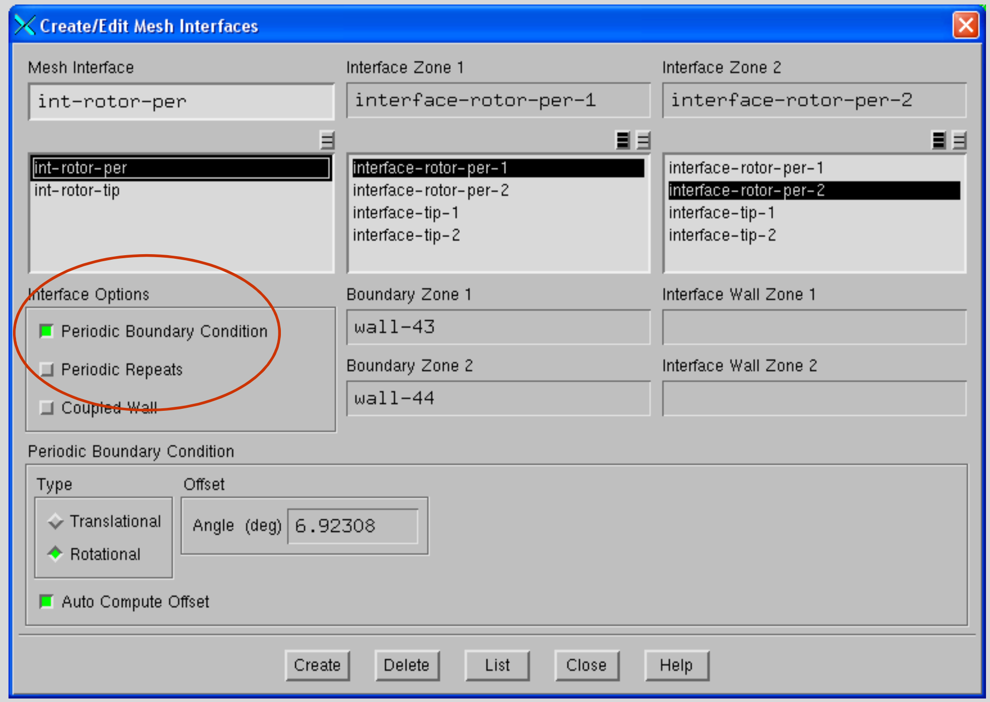




**2D Tube Heat Exchanger**

# Non-conformal Periodic Boundary Conditions

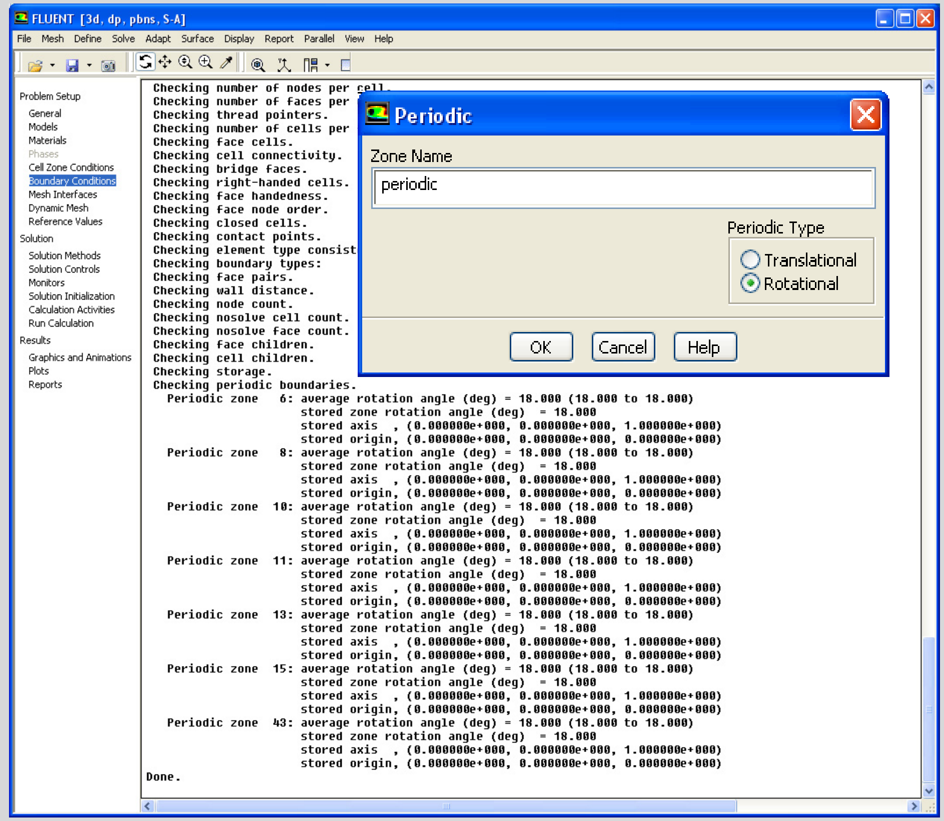
- **Fluent permits the use of non-conformal rotationally periodic BCs**
- **Non-conformal periodics do not require a matching mesh on the boundaries**
  - **Coupling of the periodic zones is accomplished using the same algorithms employed in non-conformal interfaces**
- **Non-conformal periodic can now 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

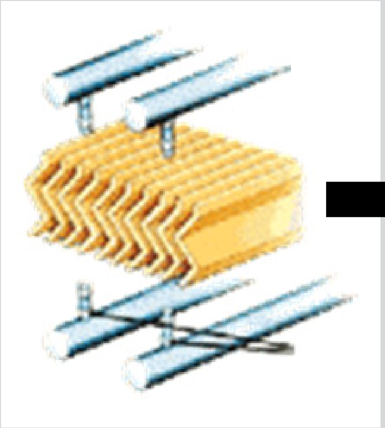
- **Defined on the cell faces only:**
  - Thickness of these internal faces is zero
  - These internal faces provide means of introducing step changes in flow properties

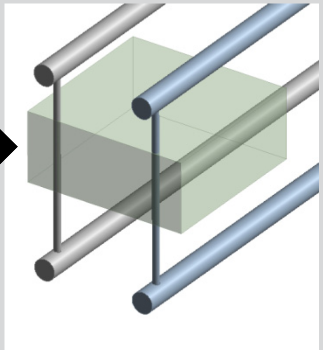
- **Used to implement various physical models including:**
  - Fans
  - Radiators
  - Porous-jump models
    - Preferable over porous media for its better convergence behavior
  - Interior walls

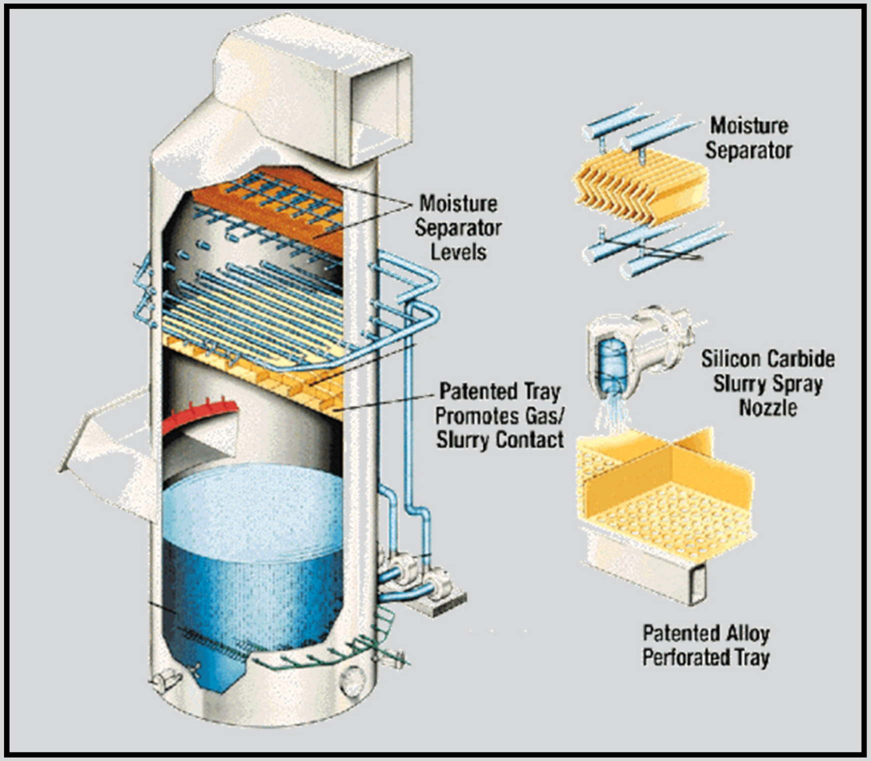
# Porous Domains

- **Used to model flows where the geometry is too complex to resolve with a grid**

- **Instead of including the geometric details, their effects are accounted for numerically**







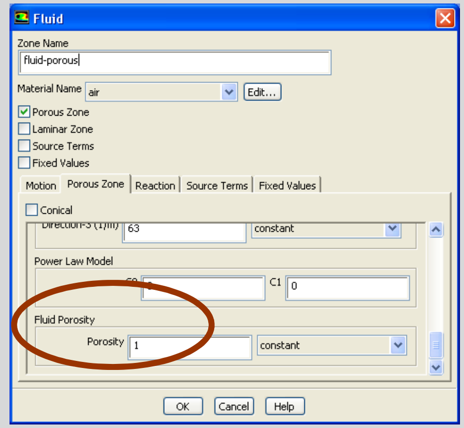

Images Courtesy of Babcock and Wilcox, USA

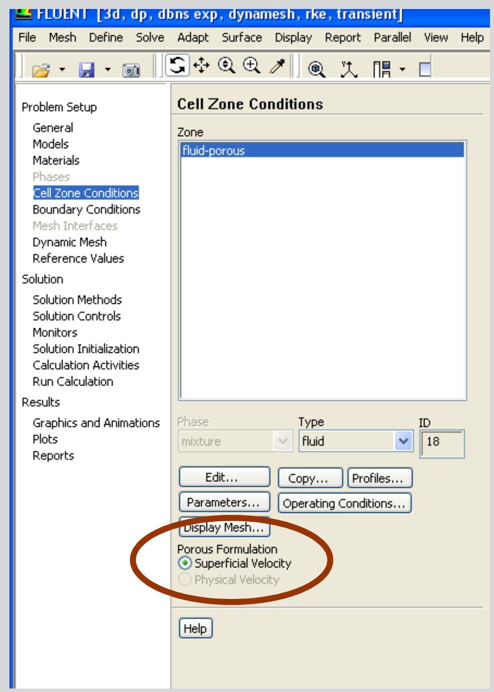
# Porous Domains

- **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 Zones

## Directional Loss

- Different losses are induced in the streamwise and transverse directions
- Honeycombs and Porous plates

## 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