Numerical Heat Transfer

（数值传热学）

Chapter 12 How to Use ANSYS FLUENT

Instructor： Ji, Wen-Tao

CFD-NHT-EHT Center

Key Laboratory of Thermo-Fluid Science & Engineering

Xi'an Jiaotong University

Xi'an, 2017-Dec.11

数值传热学

第12章 ANSYS FLUENT软件学习和应用

主讲：冀文涛

西安交通大学能源与动力工程学院
热流科学与工程教育部重点实验室

2017年12月11日，西安

Chapter 12 How to Use ANSYS FLUENT

12. 1. Numerical Heat Transfer Software

ANSYS Fluent, CFX, COMSOL, STAR-CD,

ABAQUS, PHOENICS, ADINA, NASTRAN…..

Market share: Fluent>CFX> others

Accuracy: case-dependent

Technical documentation available:

Fluent>CFX> others

Advantage of commercial NHT Software:

Easy to use!

However, it can not solve all the problems!!

Advantage of Self-programming for NHT:

It is rather important for research!

We can understand the basic procedures and mechanisms in NHT.

12.1.2 ANSYS Fluent software

Fluid flow

Conduction / Convection / Radiation Heat Transfer

Turbulence Modeling

Multiphase Flow

Fluid-Structure Interaction

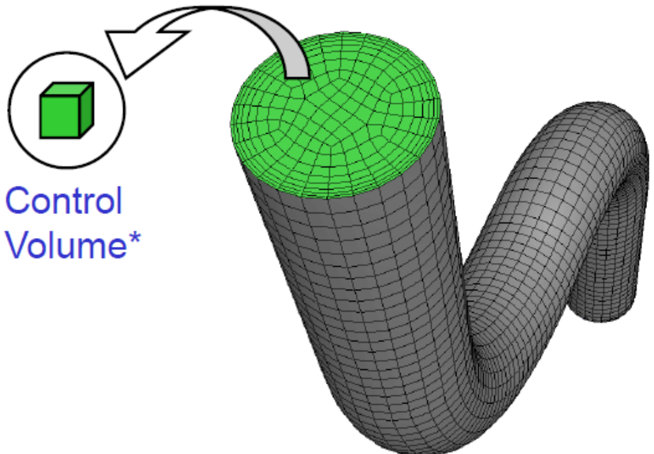
12.1.3 How Does NHT Software Work?

- ANSYS Fluent solvers are based on the finite volume method.

1) Domain is discretized into a finite set of control volumes.

2) General conservation equations for mass, momentum, energy, etc. are solved on this set of volumes.

3) Partial differential equations are discretized into a system of algebraic equations.



4) All algebraic equations are then solved numerically to render the solution field.

Fluid region of pipe flow is discretized into a finite set of control volumes

12.2. NHT Modeling Overview

Solver

Pre-Processing

- Modeler
- Mesh Generator

Equations solved on mesh

- **Transport Equations**
  - **mass**
  - **momentum**
  - **energy**
- **Supporting Physical Models**

- Physical Models
  - Turbulence
  - Combustion
  - Radiation
  - Multiphase
  - Phase Change
  - Moving Zones
  - Moving Mesh

- **Material Properties**
- **Boundary Conditions**
- **Initial Conditions**

- Solver Settings

- Post-Processing

12.2.1 NHT Analysis: Basic Steps

- Problem Identification and Pre-Processing

1. Define your modeling goals.
2. Identify the domain you will model.
3. Design and create the grid.(网格生成)

- Solver Execution

4. Set up the numerical model.(算法和格式选择)
5. Compute and monitor the solution.(方程求解)

- Post-Processing

6. Examine the results.
7. Consider revisions to the model.

1. Define Your Modeling Goals

1) What results are you looking for?

- What are your modeling options?

  - What physical models will need to be included in your analysis?
  - What simplifying assumptions do you have to/can make?
  - Do you want to develop a model which is not included in FLUENT?
  - User-defined functions (written in C) in FLUENT

2) What degree of accuracy is required?

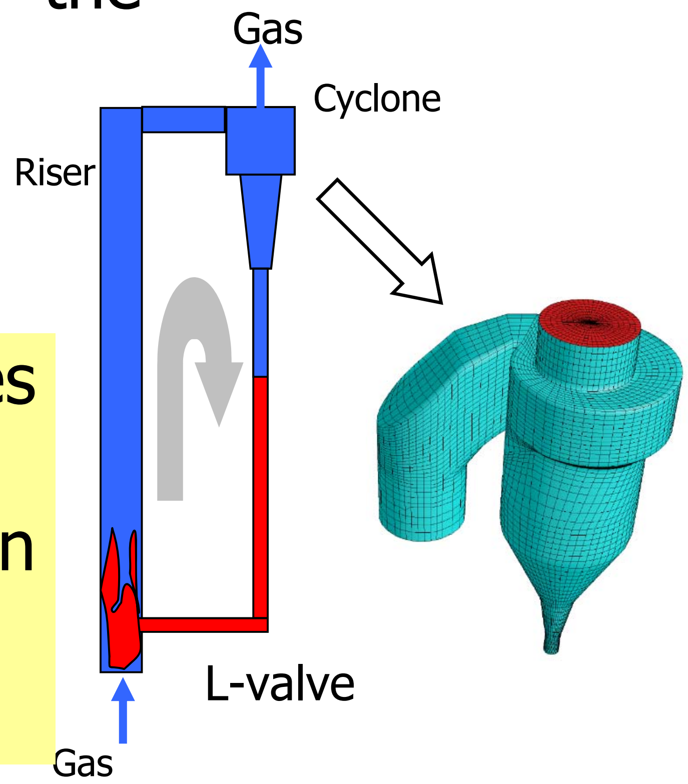
3) How quickly do you need the results?

2.Identify the Domain You Will Model

1) How will you isolate a piece of the complete physical system?

2) Where will the computational domain begin and end?

- Are the boundary condition types appropriate?
- Do you have boundary condition information at these boundaries?
- Is the domain appropriate?



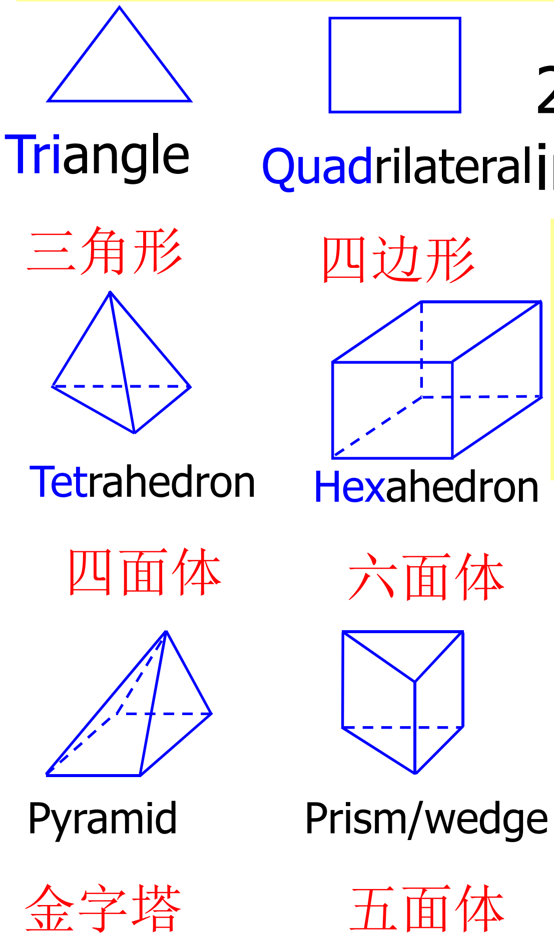
3) Can it be simplified or approximated as a 2D or axisymmetric problem?

Example: Cyclone Separator

3.Design and Create the Grid

1) Can you use a quad/hex (四边形的/六面体的) grid or should you use a tri/tet (三角形/四面体) grid or hybrid grid?

- How complex is the geometry and flow?



2) What degree of grid resolution is required in each region of the domain?

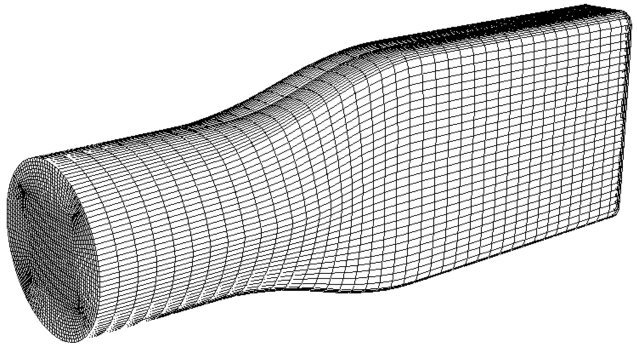
- Is the resolution sufficient for the geometry?
- Can you predict regions with high gradients?
- Will you use adaption to add resolution?

3) Do you have sufficient computer memory?

- How many cells are required?
- How many models will be used?

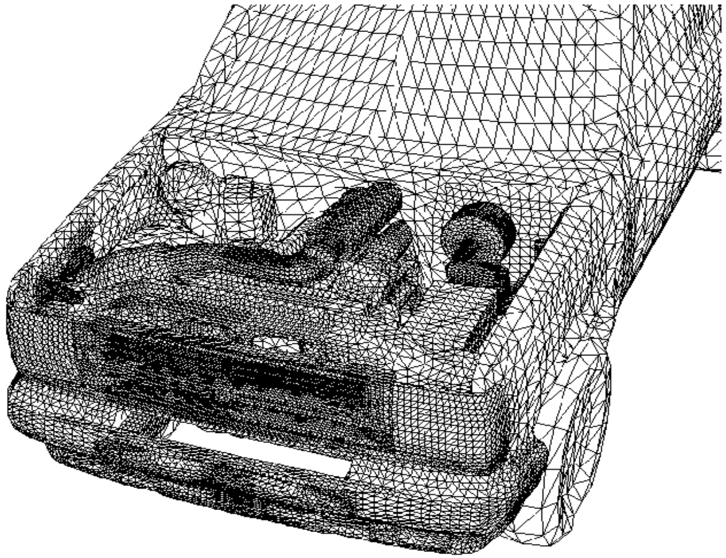
Tri/Tet vs. Quad/Hex Meshes

1) For **simple** geometries, quad/hex meshes can provide higher-quality solutions with fewer cells than a comparable tri/tet mesh.



Align the gridlines with the flow.

2) For **complex** geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh.



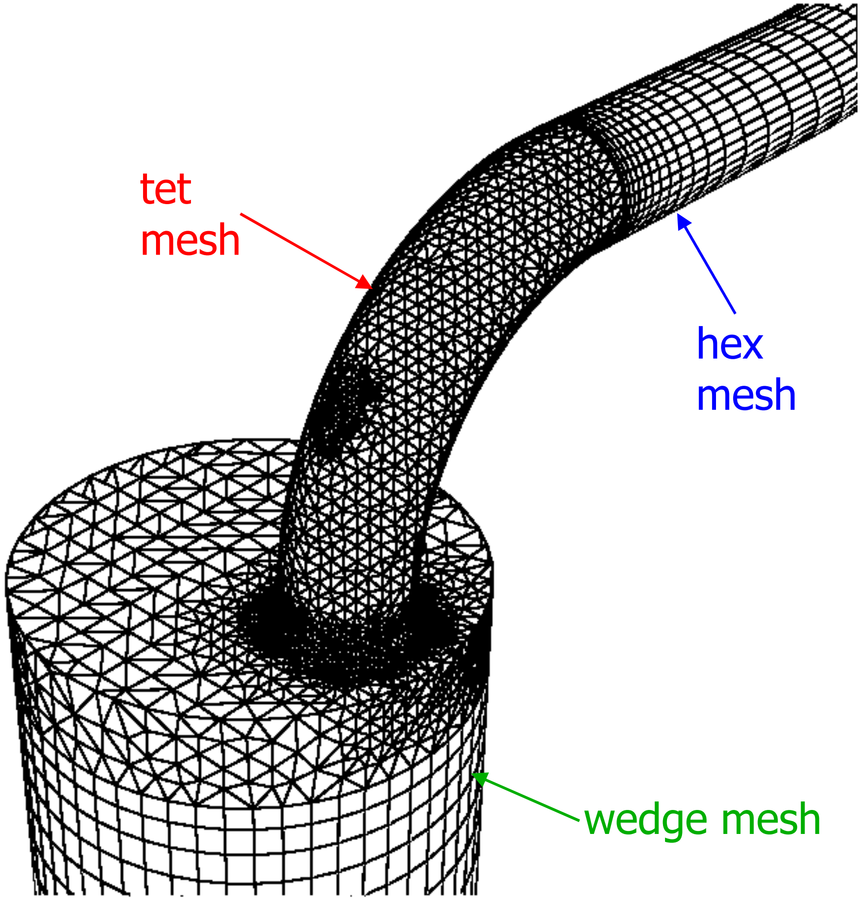
Hybrid Mesh Example

- Valve port grid

1) Specific regions can be meshed with different cell types.

2) Both efficiency and accuracy are enhanced relative to a hexahedral or tetrahedral mesh alone.

3) Tools for hybrid mesh generation are available in Gambit and ICEM.



Hybrid mesh for an engine valve port

4. Set Up the Numerical Model

- For a given problem, you will need to:
  - Select appropriate physical models.
    - Turbulence, combustion, multiphase, etc.
  - Define material properties.
    - Fluid/Solid/Mixture
  - Prescribe operating conditions.
  - Prescribe boundary conditions at all boundary zones.
  - Provide an initial solution.
  - Set up solver controls.
  - Set up convergence monitors.

Solving initially in 2D will provide valuable experience with the models and solver settings for your problem in a short amount of time.

5. Compute the Solution

- The discretized conservation equations are solved *iteratively*.

  A number of iterations are usually required to reach a converged solution.

- Convergence is reached when:

  Changes in solution variables from one iteration to the next are negligible.
  Residuals provide a mechanism to help monitor this trend.
  Overall property conservation is achieved.

- The accuracy of a converged solution is dependent upon:

  Appropriateness and accuracy of physical models.
  Grid resolution and independence
  Problem setup

6. Examine the Results

- Examine the results to review solution and extract useful data.

1) Visualization Tools can be used to answer such questions as:

- What is the overall flow pattern?
- Is there separation?
- Where do shear layers form?
- Are key flow features being resolved?

2) Numerical Reporting Tools can be used to calculate quantitative results:

- Forces and Moments
- Average heat transfer coefficients
- Surface and Volume integrated quantities
- Flux Balances

Examine results to ensure property conservation and correct physical behavior. High residuals may be attributable to only a few cells of poor quality.

7. Consider Revisions to the Model

1) Are physical models appropriate?

- Is flow turbulent? Is flow unsteady?
- Are there compressibility effects? Are there 3D effects?

2) Are boundary conditions correct?

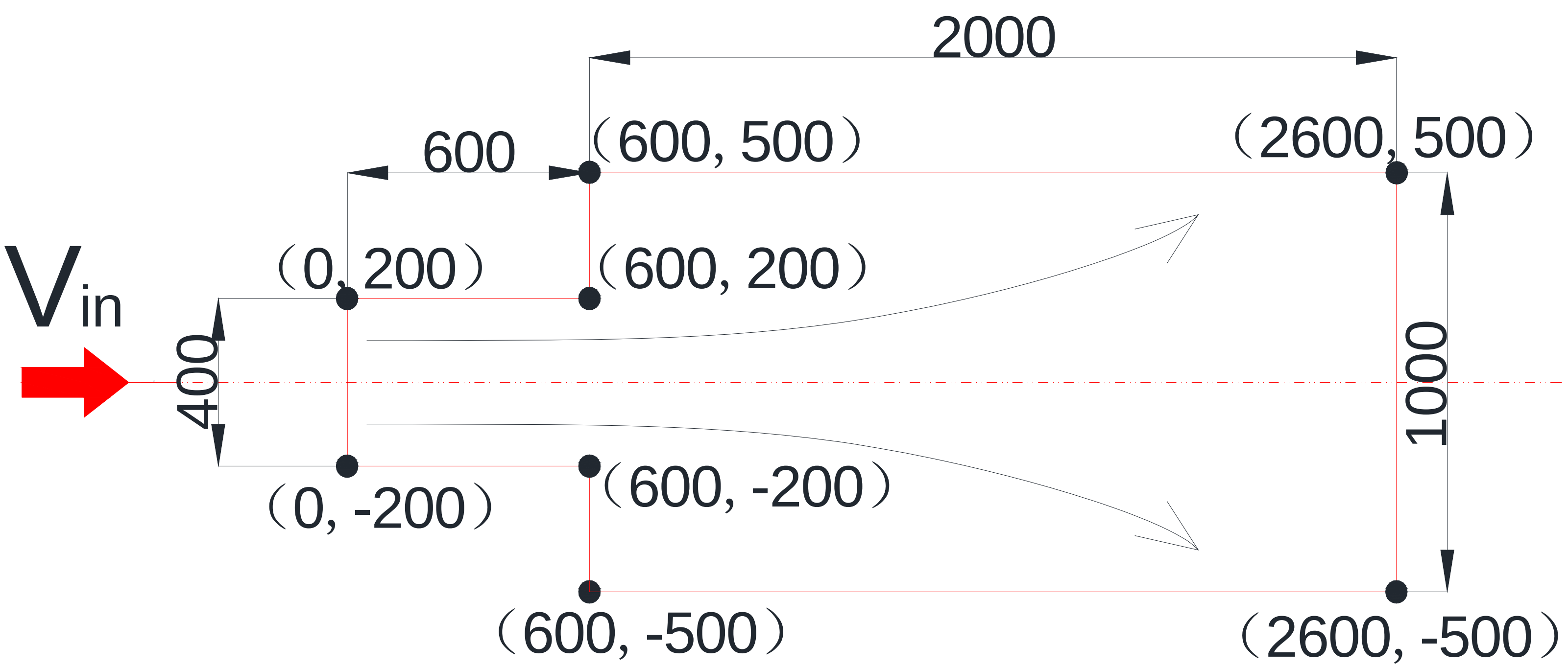
- Is the computational domain large enough?
- Are boundary conditions appropriate?
- Are boundary values reasonable?

3) Is grid adequate?

- Can grid be adapted to improve results?
- Does solution change significantly with adaption, or is the solution grid independent?
- Does boundary resolution need to be improved?

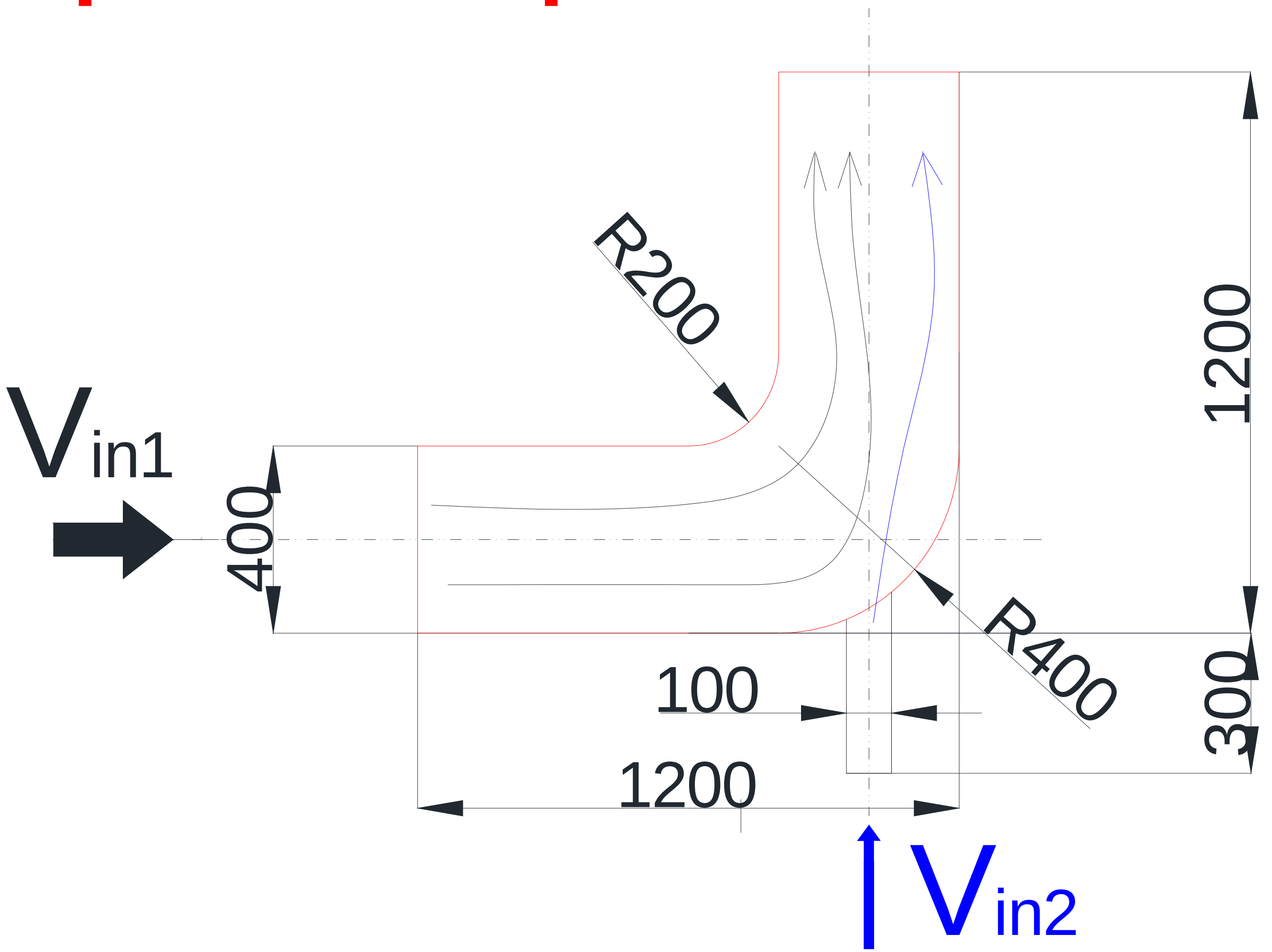
12.3.Simple Examples to Using FLUENT

Example 1: Sudden Expansions



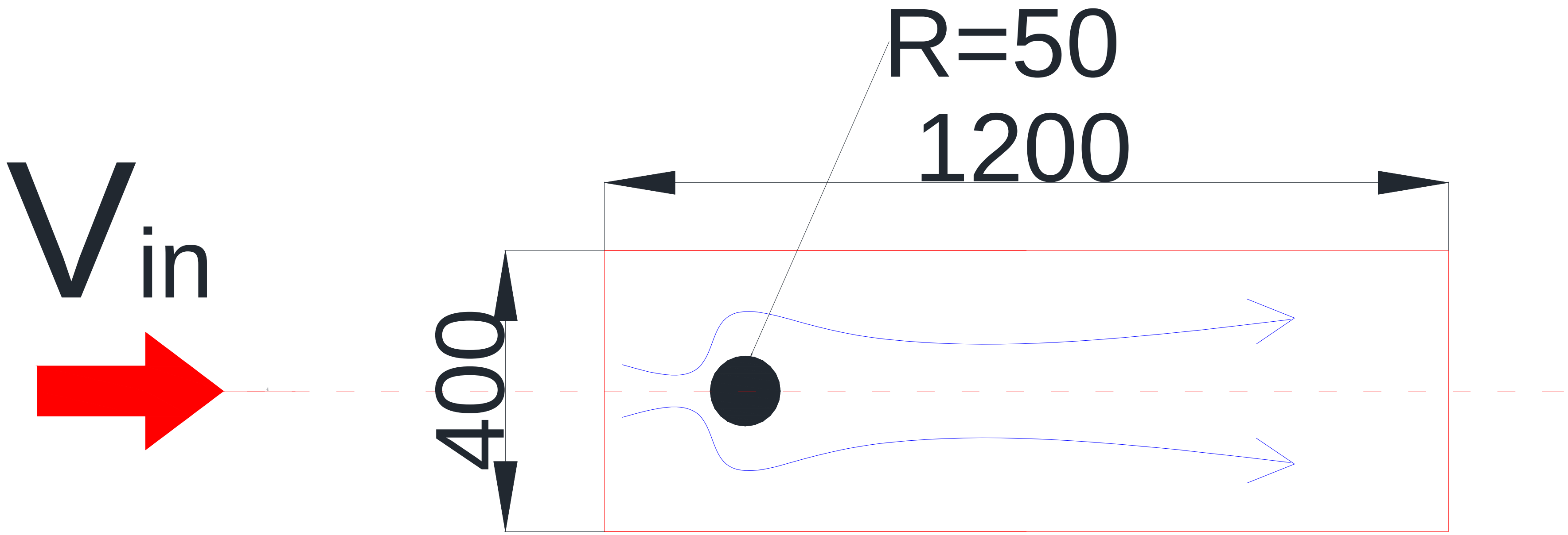
$V_{in}=2m/s$

Example 2: 2D Pipe Junction



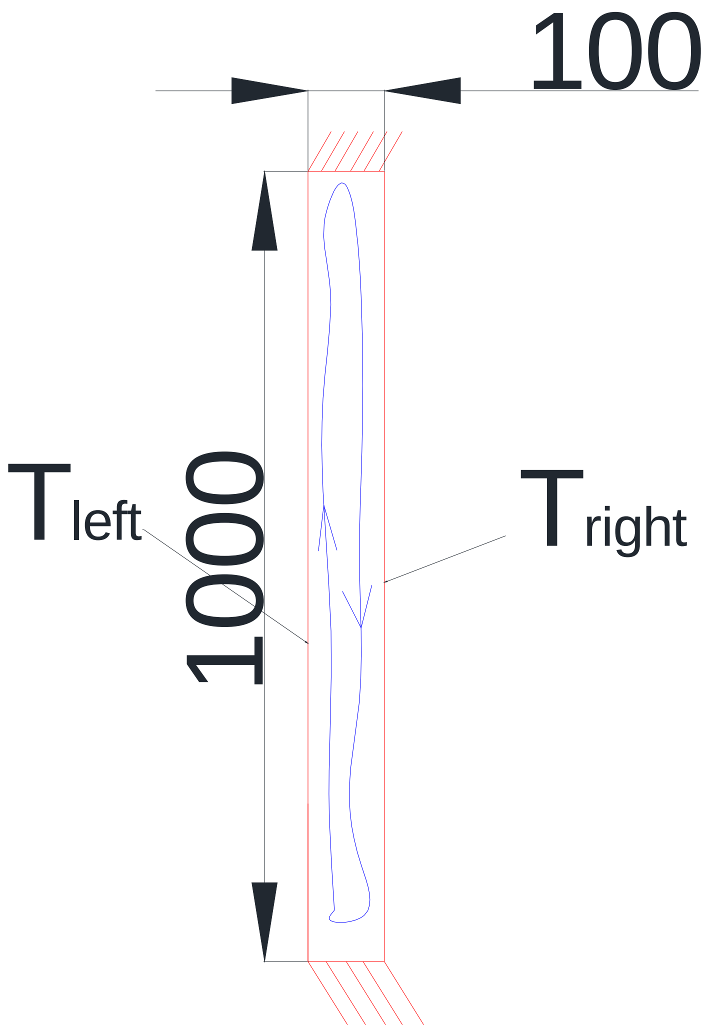
T_{in1}=300K V_{in1}=1m/s T_{in2}=350K V_{in1}=5m/s

Example 3: Flow over a cylinder



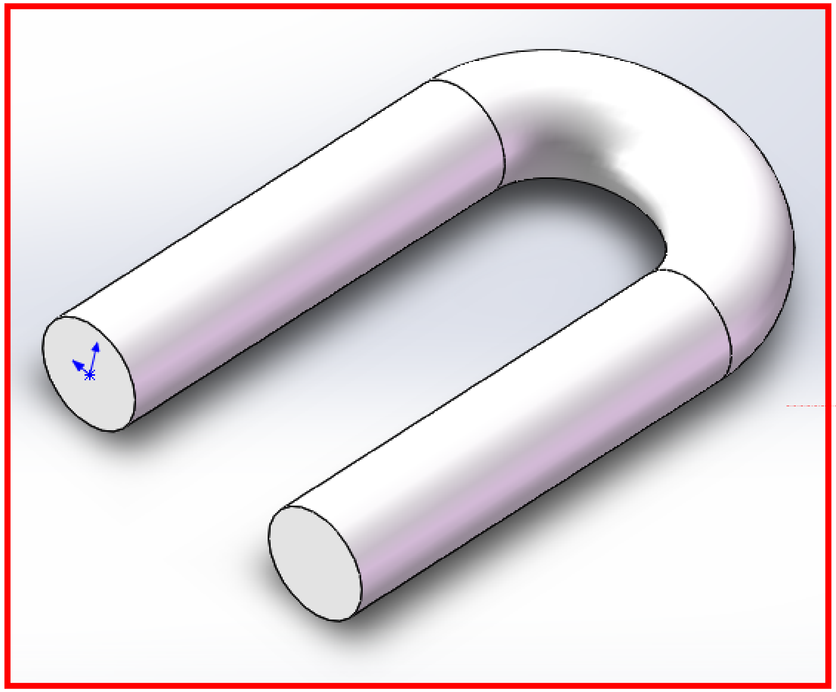
V_{in1}=0.01m/s

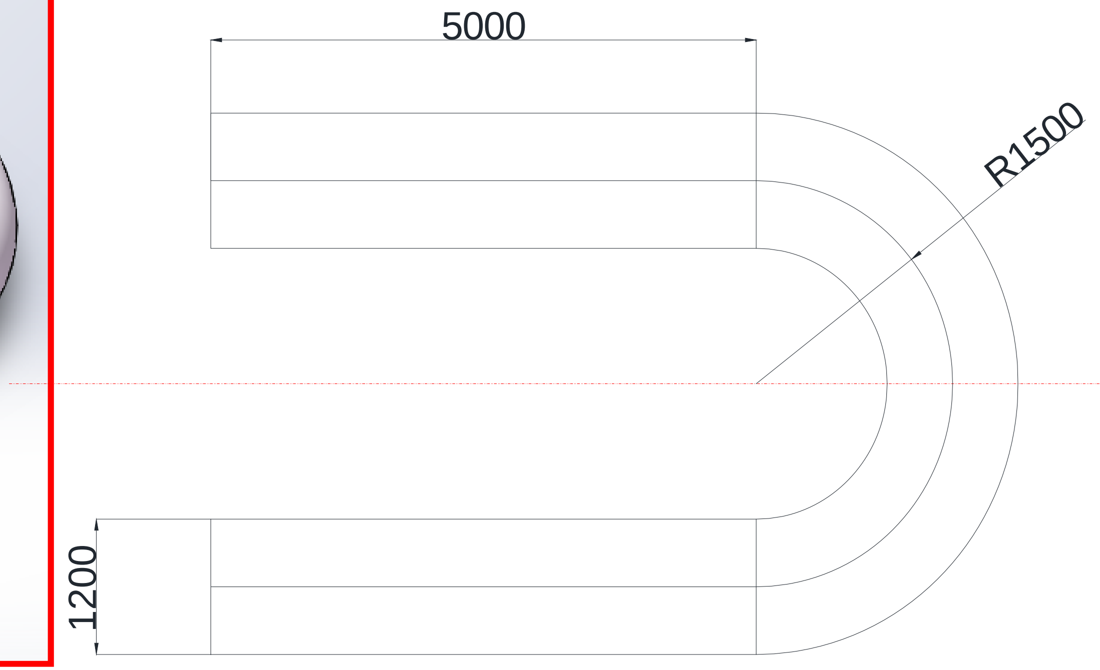
Example 4: Natural convection in a slot



T_{left}=320℃, T_{right}=300℃

Example 5: Flow in a U turn

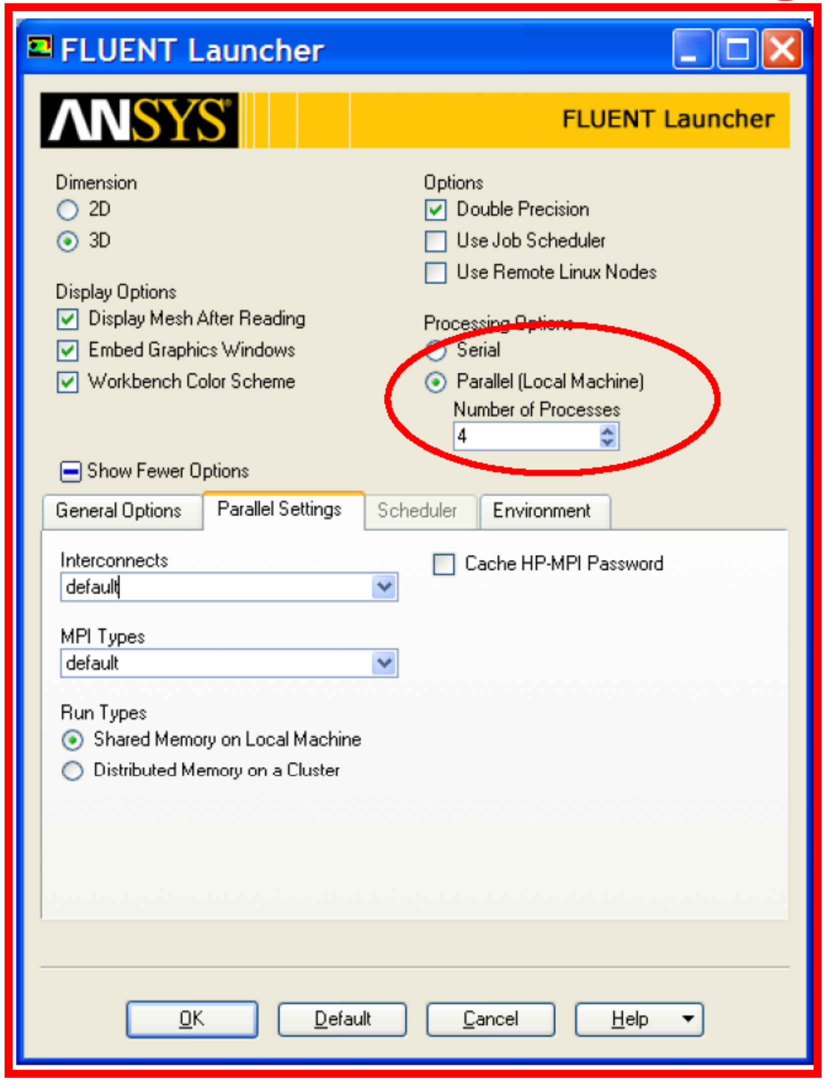




Modeling with Solidworks

Uin=2.0m/s

12.4.Procedures of Using FLUENT



Fluent launcher interface

◆ Parallel Processing（并行处理）

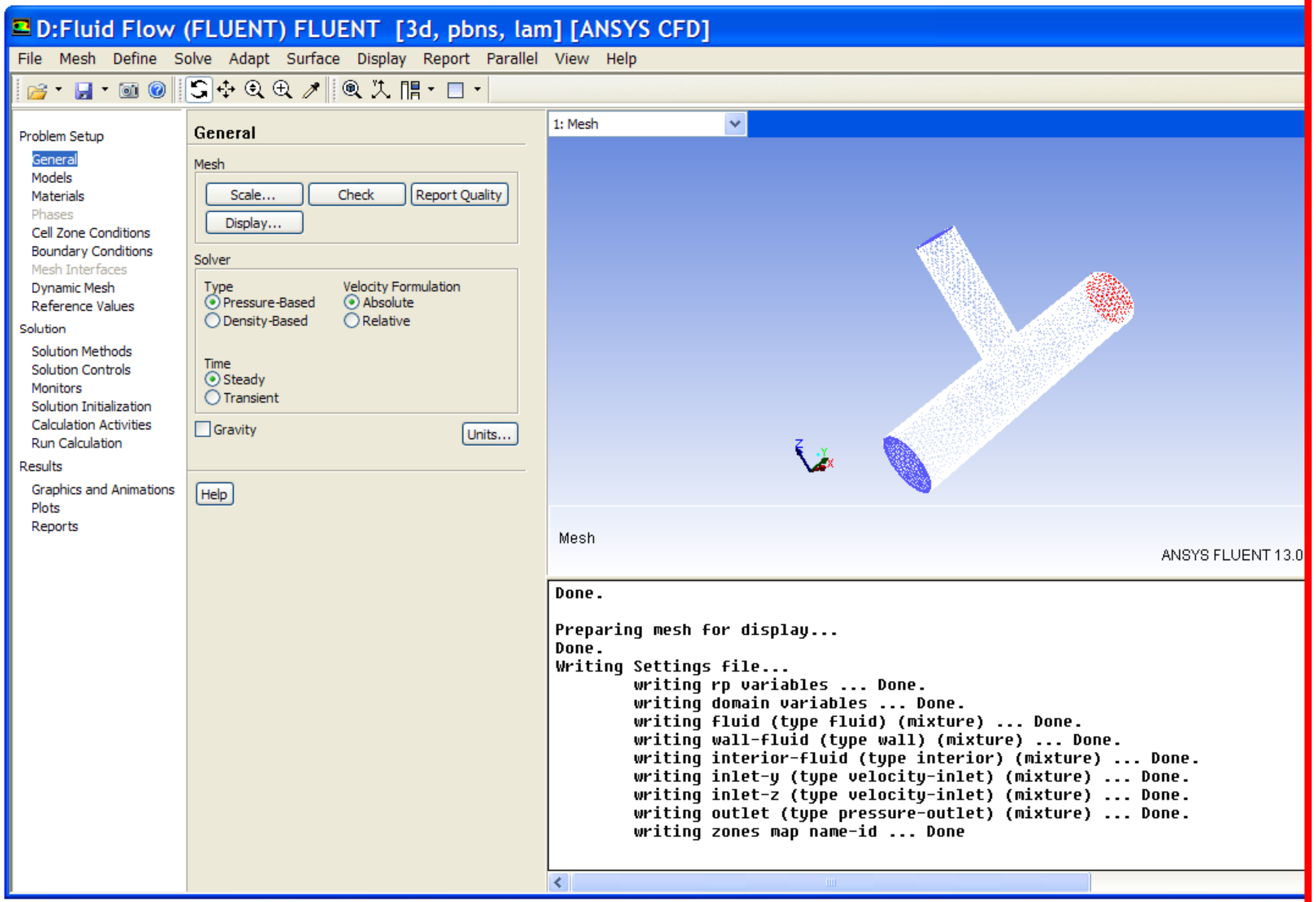
FLUENT can readily be run across many processors in parallel. This will greatly speed up the simulation time.

1) It is common for modern computers to have several processors, or ‘cores’ per processor. Each one of these can be a ‘node(节点)’ for the FLUENT simulation.

2) The mesh is automatically partitioned, and different blocks of the mesh are assigned to the different compute nodes.

3) Alternatively a distributed parallel cluster(集群) can be set up, and the simulation can run across many machines simultaneously.

- The FLUENT Graphical User Interface **(GUI,图形界面)** is arranged such that the tasks are generally arranged from top to bottom in the project setup tree.

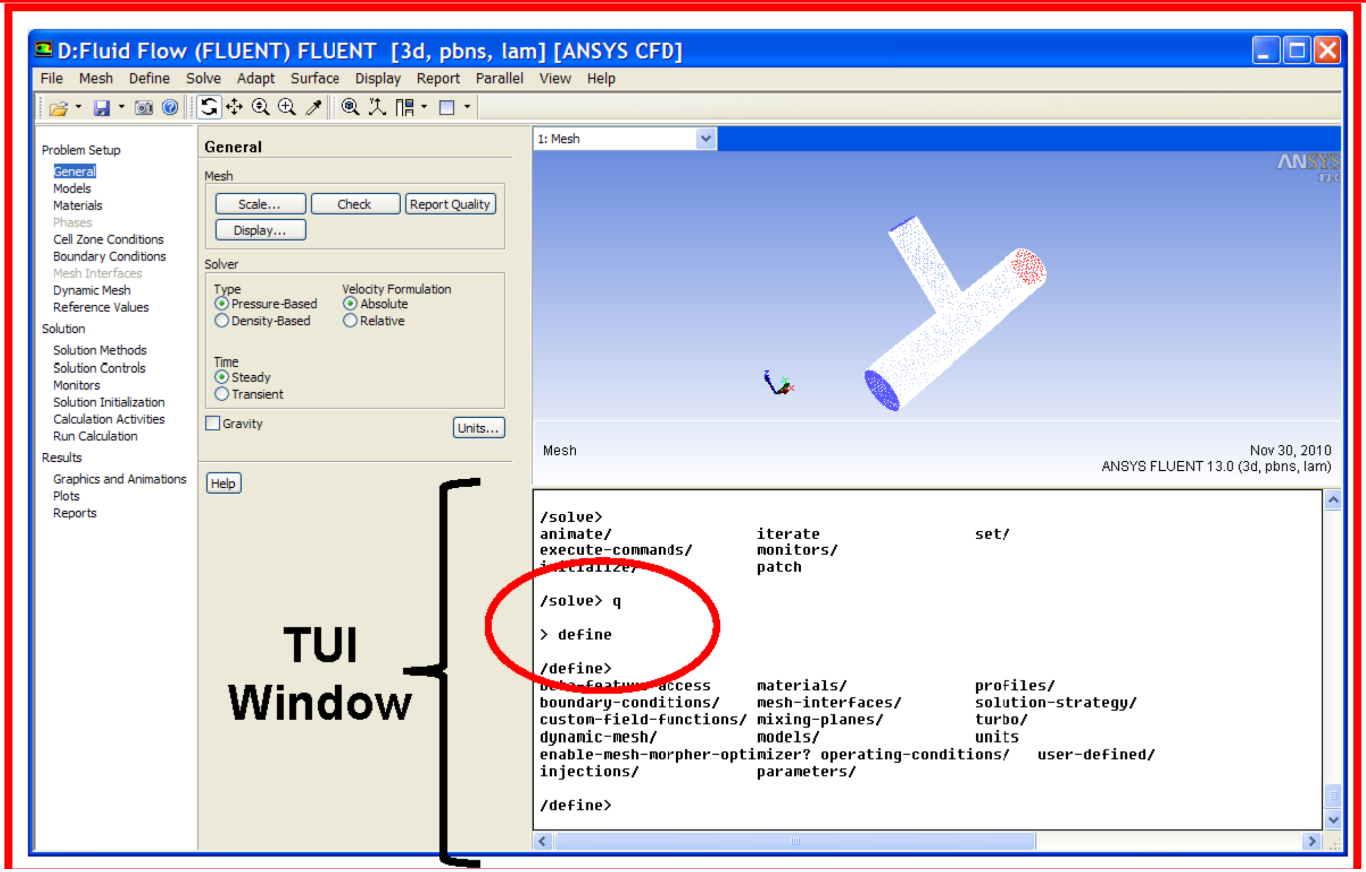


◆ Text User Interface

Most GUI commands have a corresponding TUI command.

1) Press the Enter key to display the command set at the current level.

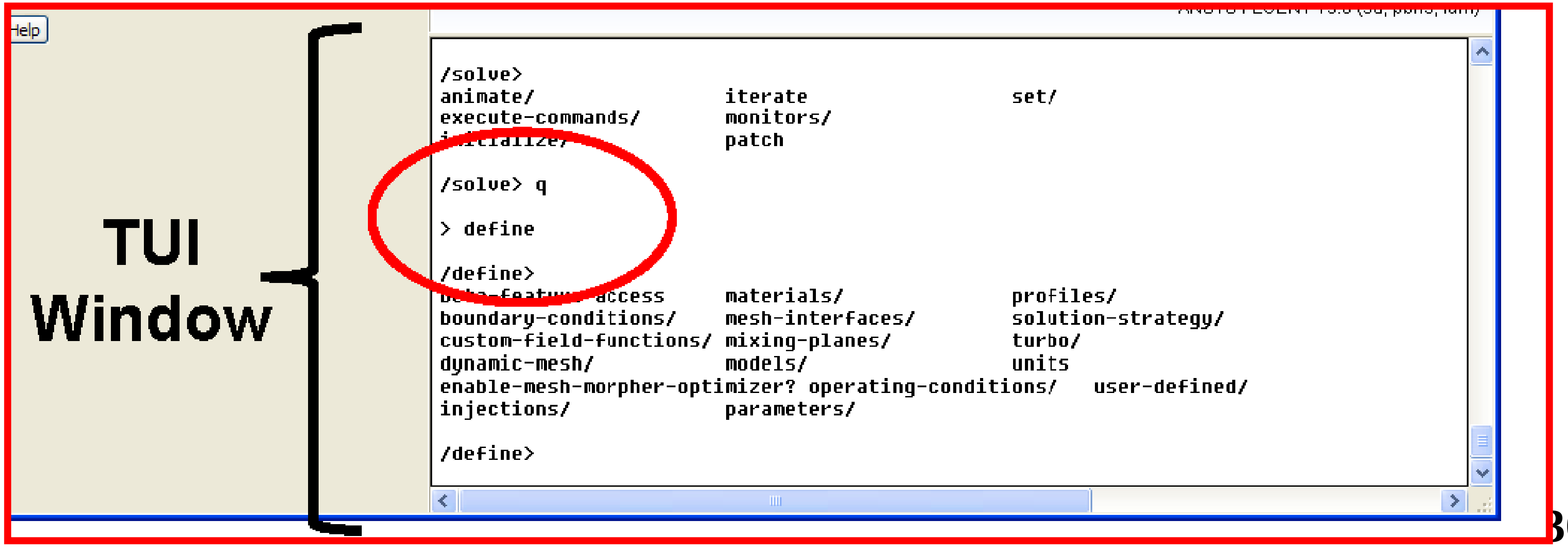
2) Some advanced commands are only available through the TUI.



The TUI offers many very valuable benefits:

1) Journal files can be constructed to automate repetitive tasks.

2) FLUENT can be run in batch mode(批处理模式), with TUI journal scripts set to automate the loading/modification/solver execution and post processing.



- 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

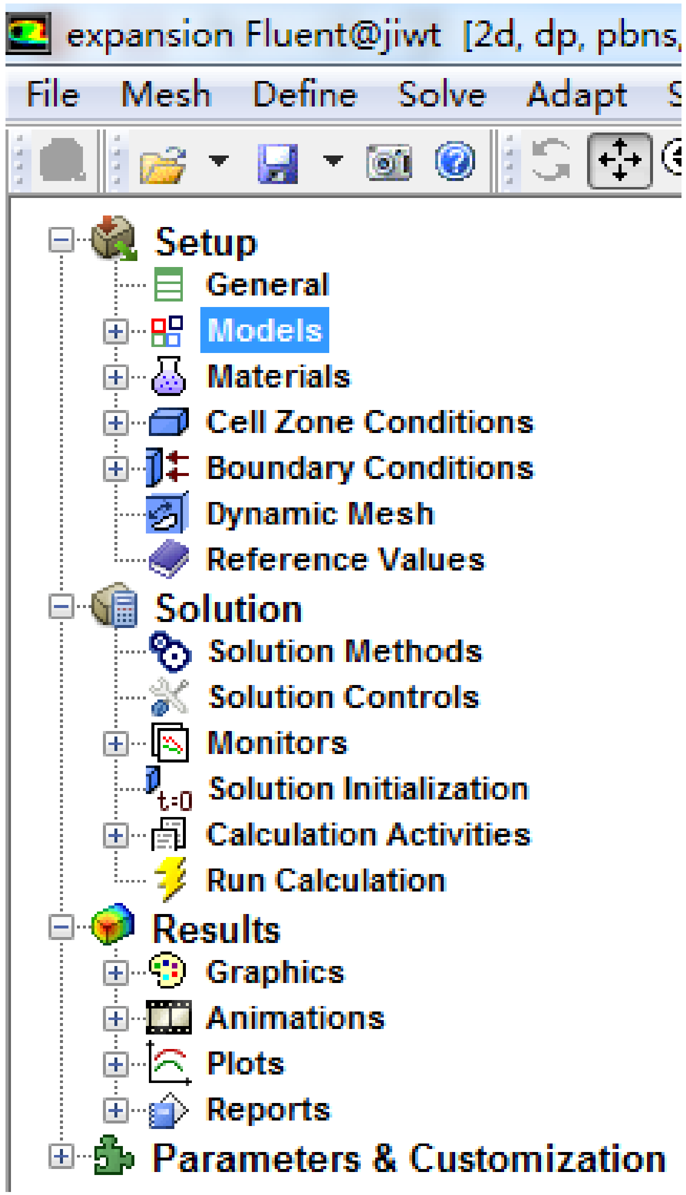
Sample Journal File

```
; 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
```

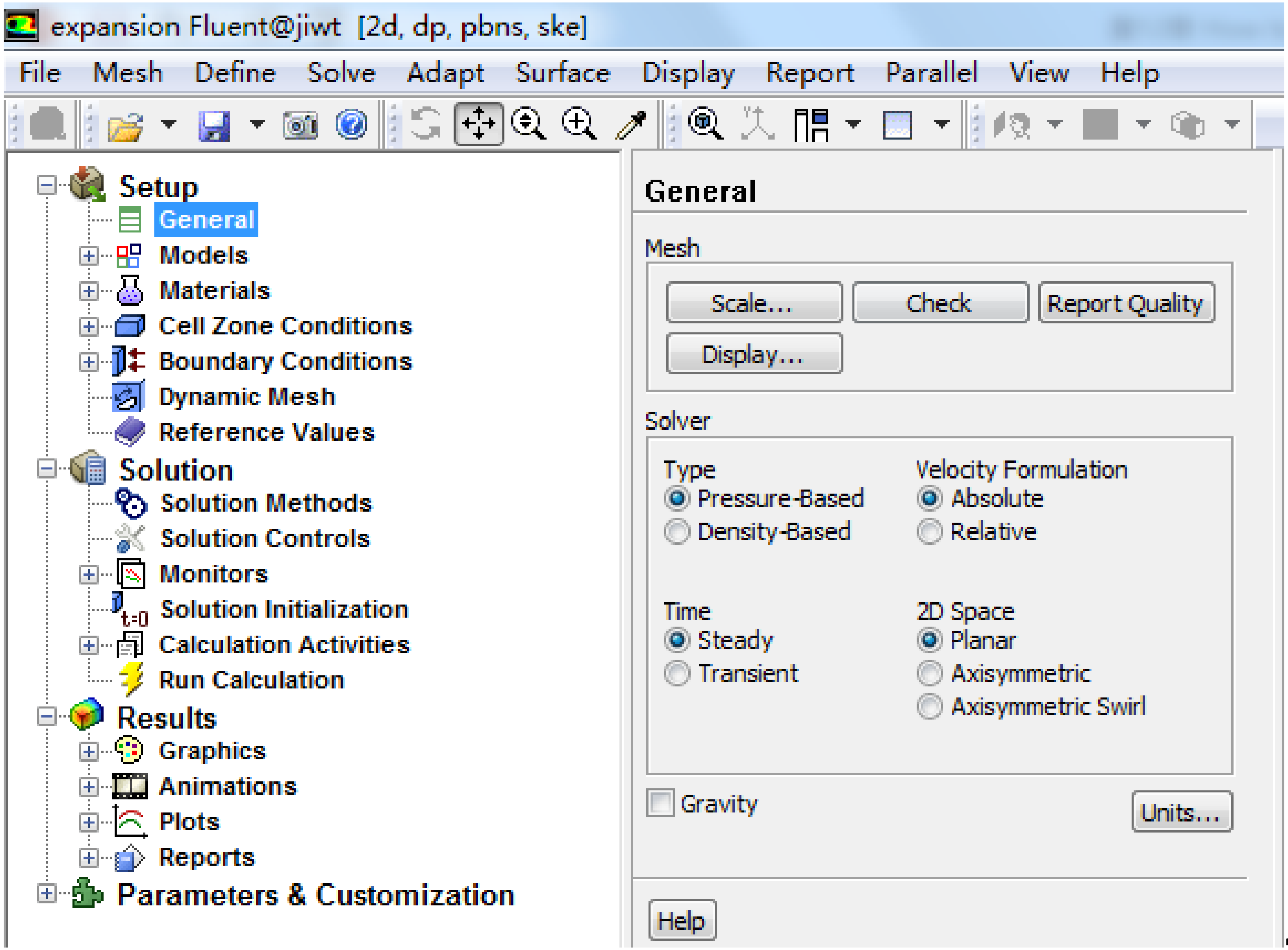
• FLUENT 16 GUI Navigation

Selecting an item in the tree opens the relevant input items in the center pane.

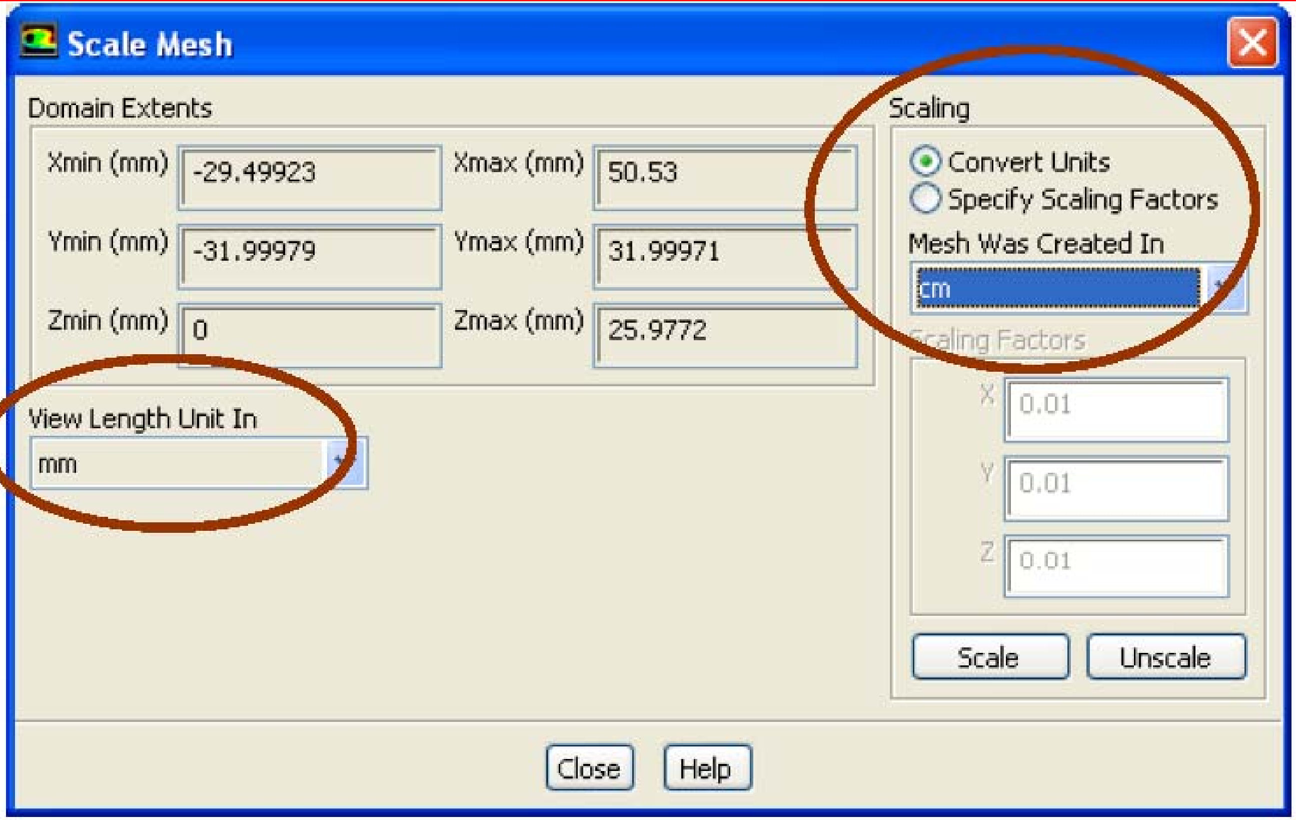
- General
- Models
- Materials
- Boundary Conditions
- Solver Settings
- Initialization and Calculation
- Post processing



1. General setting



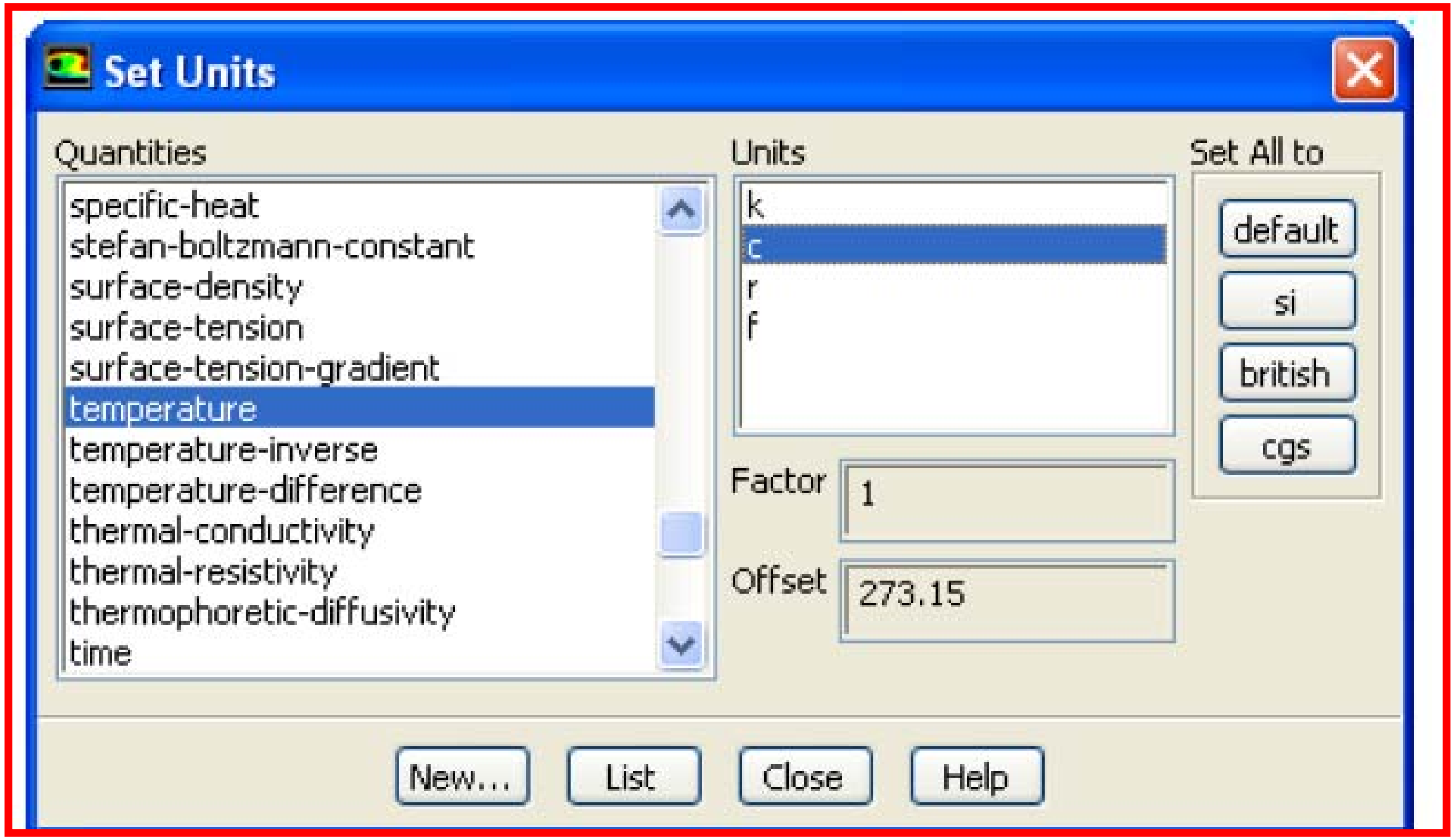
◆ Scaling the Mesh and Selecting Units



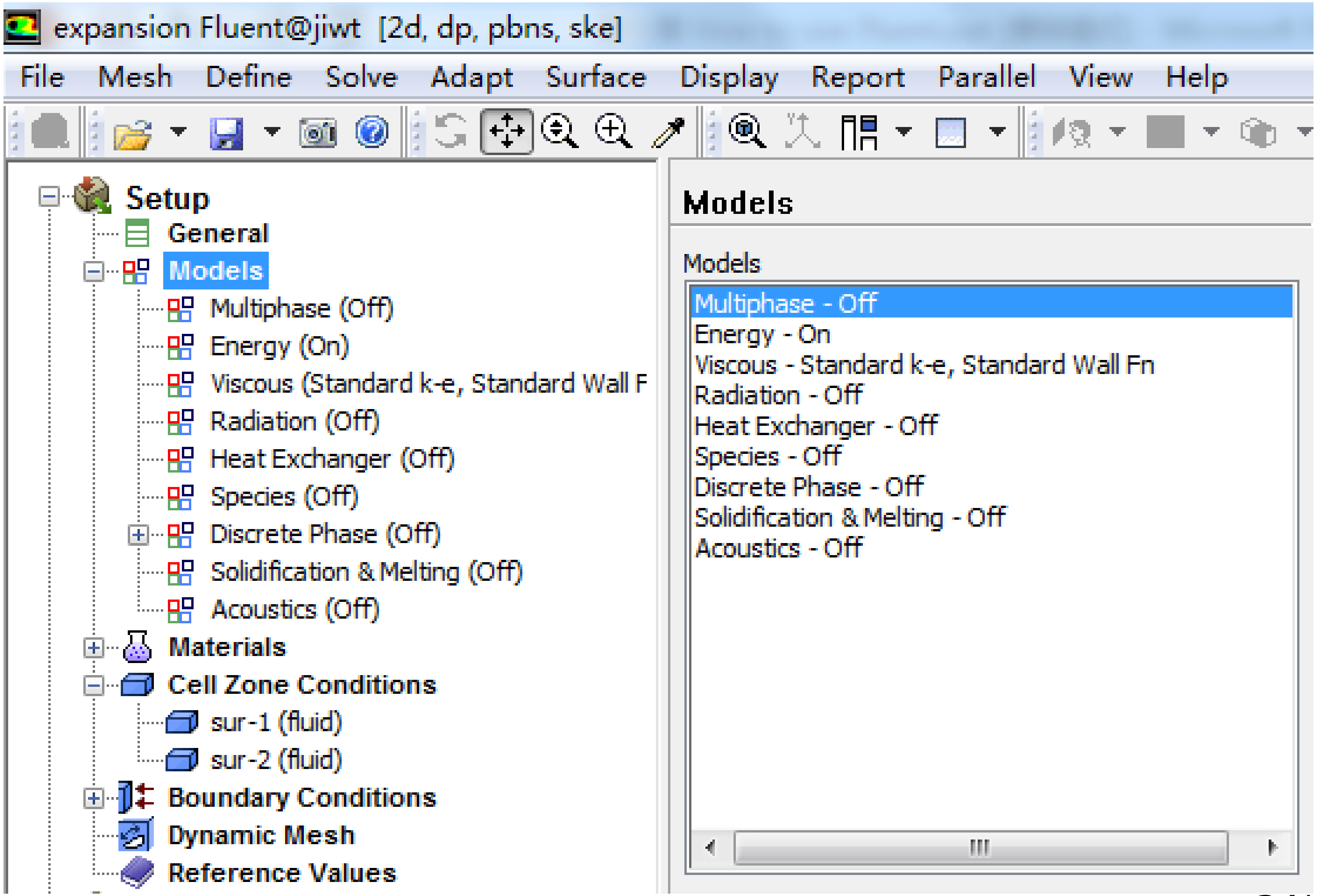
- When FLUENT reads a mesh file(.msh), all dimensions are assumed to be in units of meters.
  - If your model was not built in meters, then it must be scaled.
  - Always verify that the domain extents are correct.

Any "mixed" units system can be used if desired.

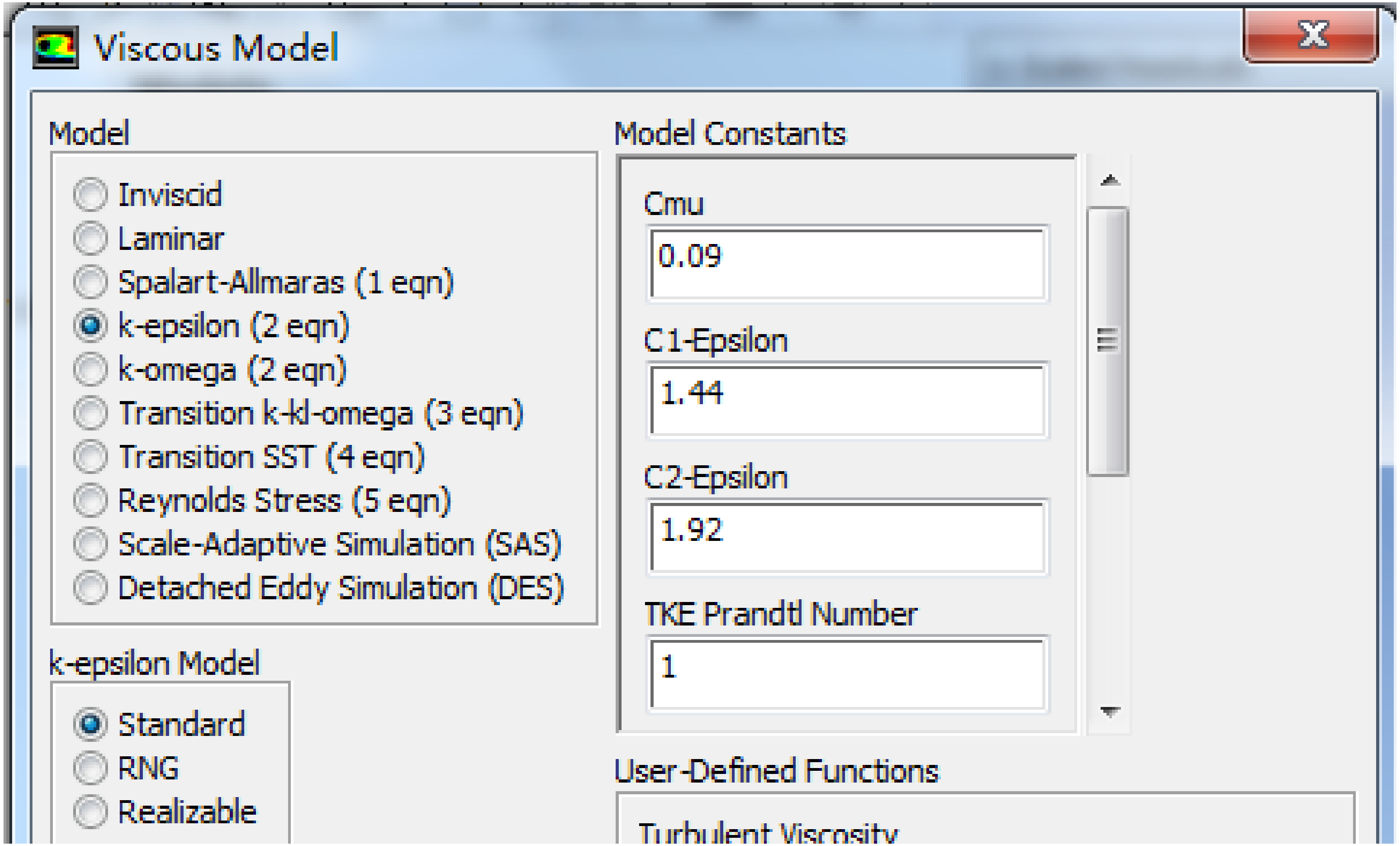
– By default, FLUENT uses the international system of units (SI).
– Any units can be specified in the Set Units panel, accessed from the top menu.



2. Setup the Models



◆ Turbulence Models Available in FLUENT(Chapter 9)



One-Equation Model
Spalart-Allmaras
Two-Equation Models
Standard k–ε
RNG k–ε
Realizable k–ε
Standard k–ω
SST k–ω
4-Equation v2f *
Reynolds Stress Model
***k–kl–ω* Transition Model**
SST Transition Model

Detached Eddy Simulation

Large Eddy Simulation

RANS based models

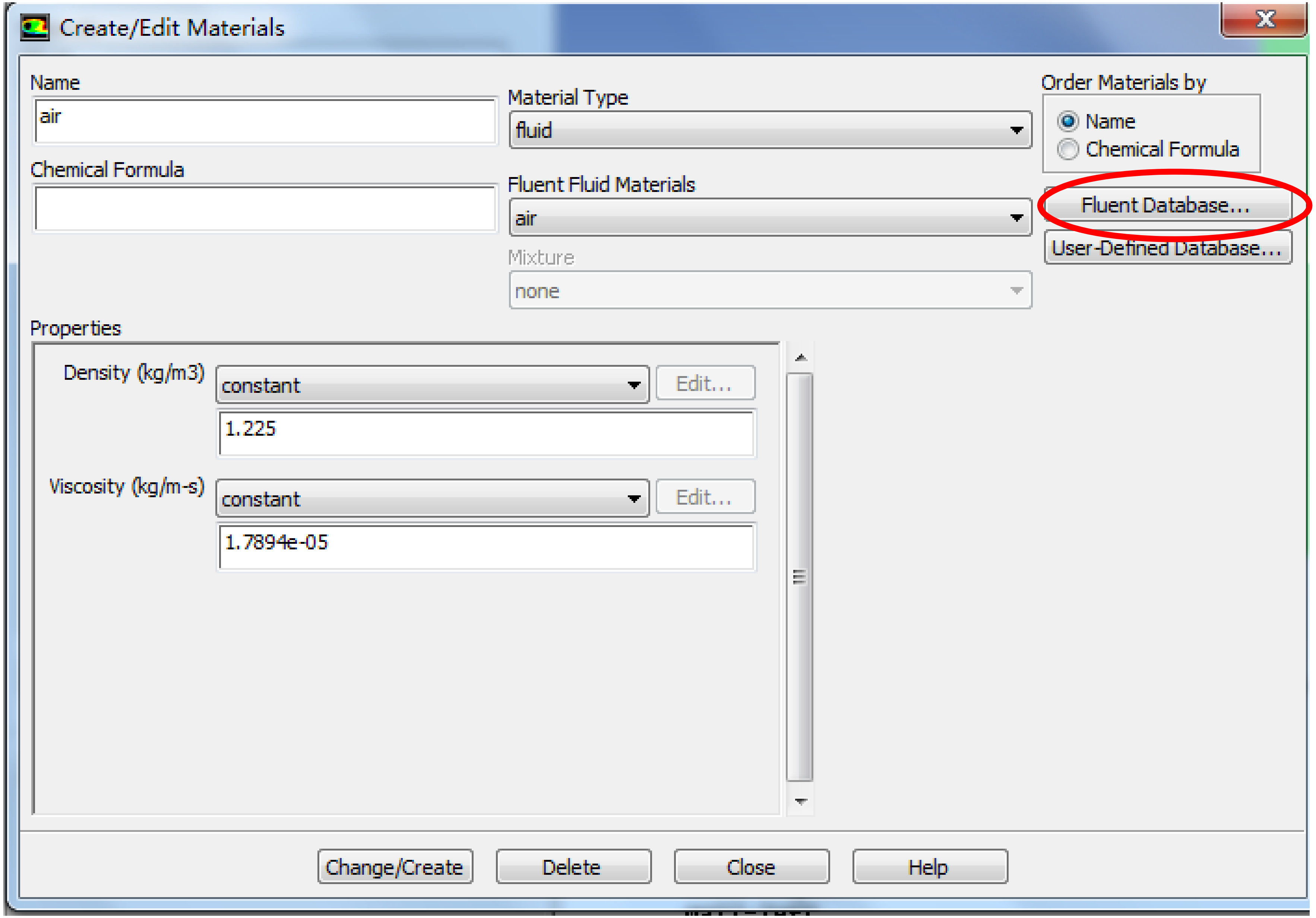
Chapter 9

Increase in Computational Cost Per Iteration

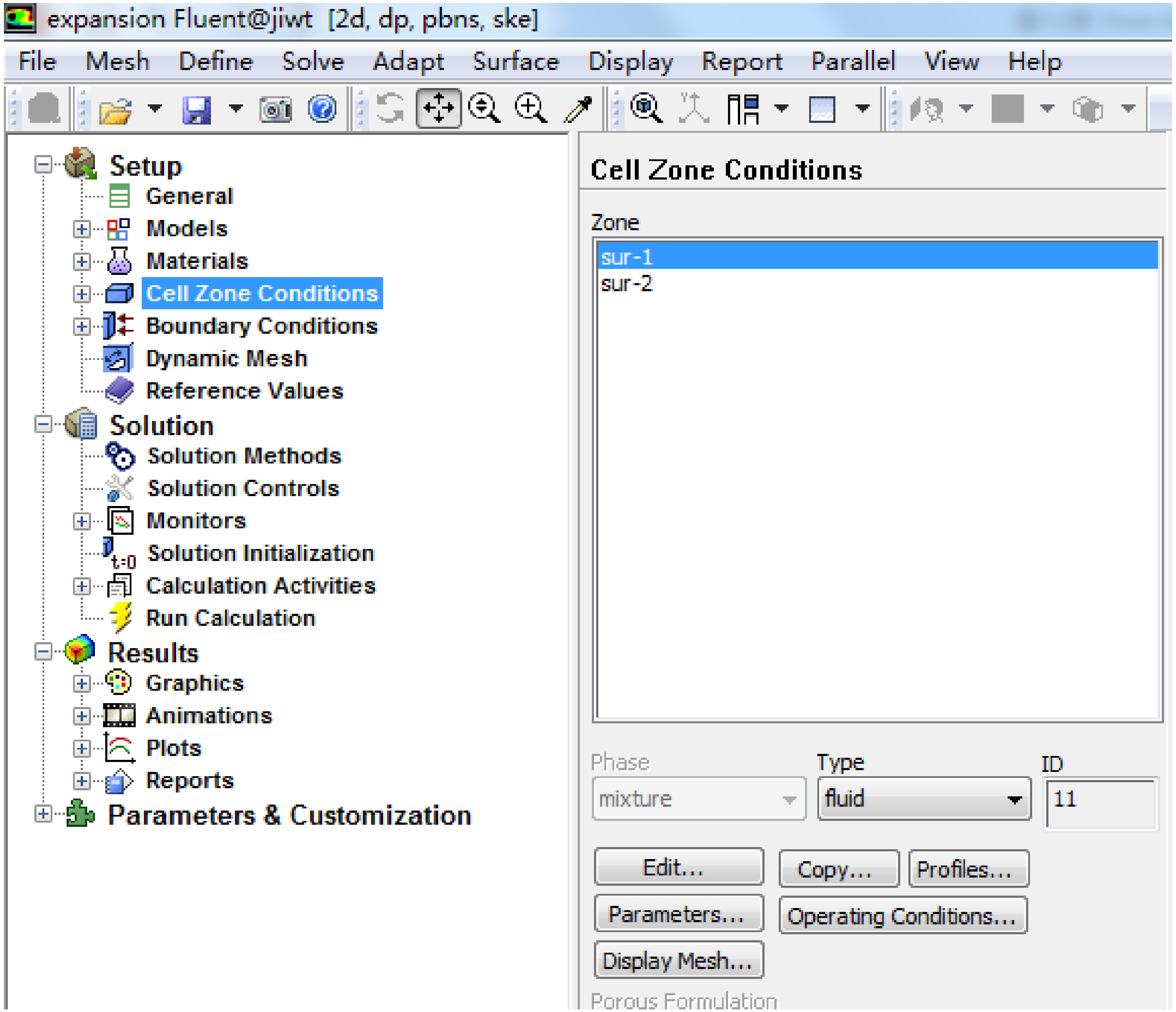
3. Material Properties

1）Material properties need to be defined for all fluids and solids to be simulated
2）The parameters depend on the models selected for the simulation
3）Many common materials are already defined in the 'FLUENT Database' and can easily be copied to the model

Note that these values may be either:
- Constants
- Functions of temperature
- Other built in functions following common relationships
- Defined by the user in a UDF.


Create/Edit Materials
Name
air
Chemical Formula
Material Type
fluid
Fluent Fluid Materials
air
Mixture
none
Order Materials by
Name
Chemical Formula
Fluent Database...
User-Defined Database...
Properties
Density (kg/m3)
constant
Edit...
1.225
Viscosity (kg/m-s)
constant
Edit...
1.7894e-05
Change/Create
Delete
Close
Help

4.Cell Zone Conditions

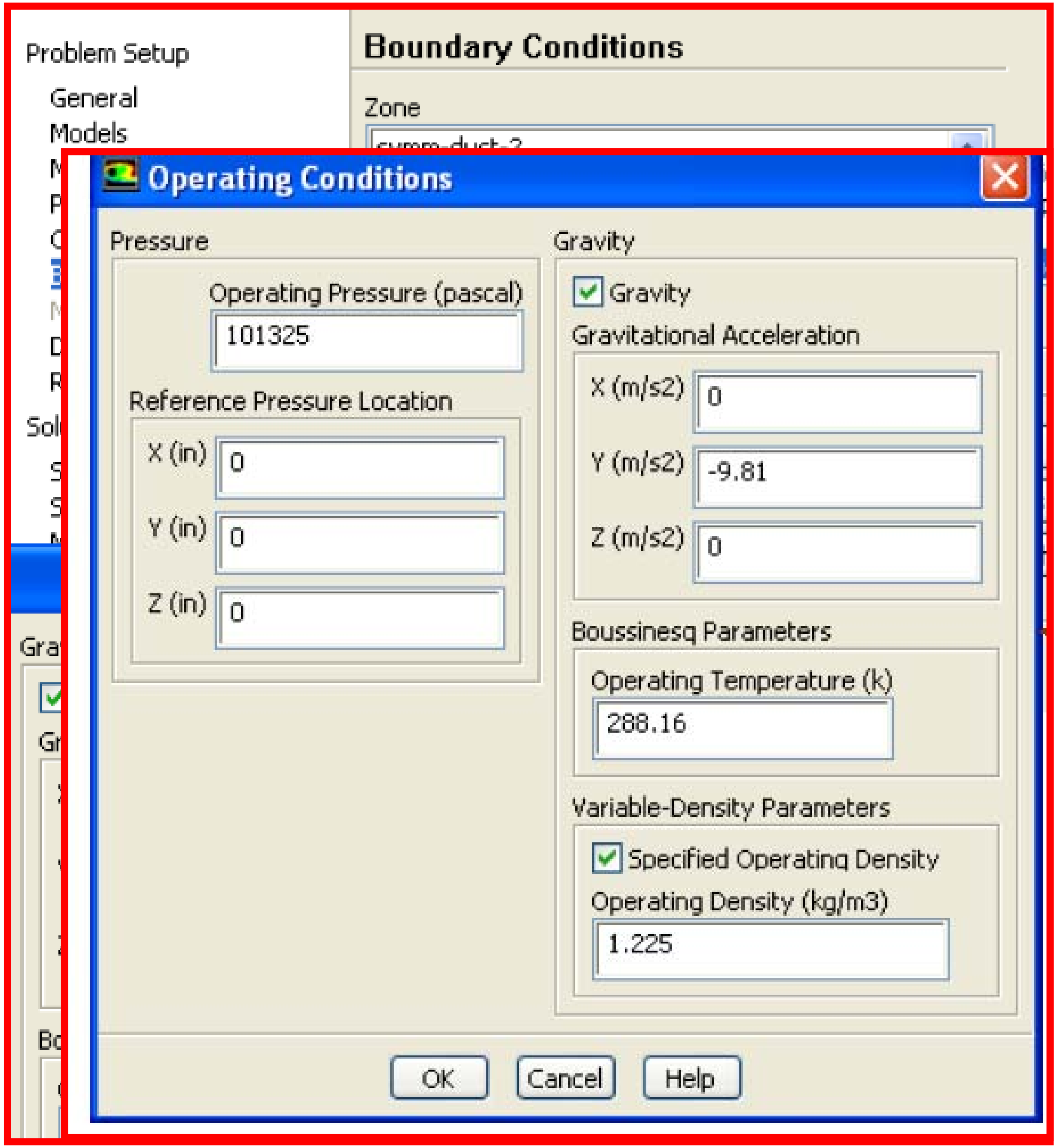


◆Operating Conditions

① The Operating Pressure with a Reference Pressure Location sets the reference value that is used in computing gauge pressures.

② The Operating Temperature sets the reference temperature (used when computing buoyancy forces).

• Specified Operating Density sets the reference value for flows with widely varying density.


Problem Setup
General
Models
Boundary Conditions
Zone
Operating Conditions
Pressure
Operating Pressure (pascal)
101325
Reference Pressure Location
X (in) 0
Y (in) 0
Z (in) 0
Gravity
Gravity
Gravitational Acceleration
X (m/s2) 0
Y (m/s2) -9.81
Z (m/s2) 0
Boussinesq Parameters
Operating Temperature (k)
288.16
Variable-Density Parameters
Specified Operating Density
Operating Density (kg/m3)
1.225
OK
Cancel
Help

◆ Defining Cell Zones and Boundary Conditions

To properly define any NHT problem, you must define:

1）Cell zones

– These relate to the middle of the grid cells

– Typically this always involves setting up which material (fluid) is in that cell

– Other values（heat sources，etc)

2）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.

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

◆ Cell Zones – Fluid

① A fluid cell zone is a group of cells for which all active equations are solved.

② The material in the cell zone must be declared.

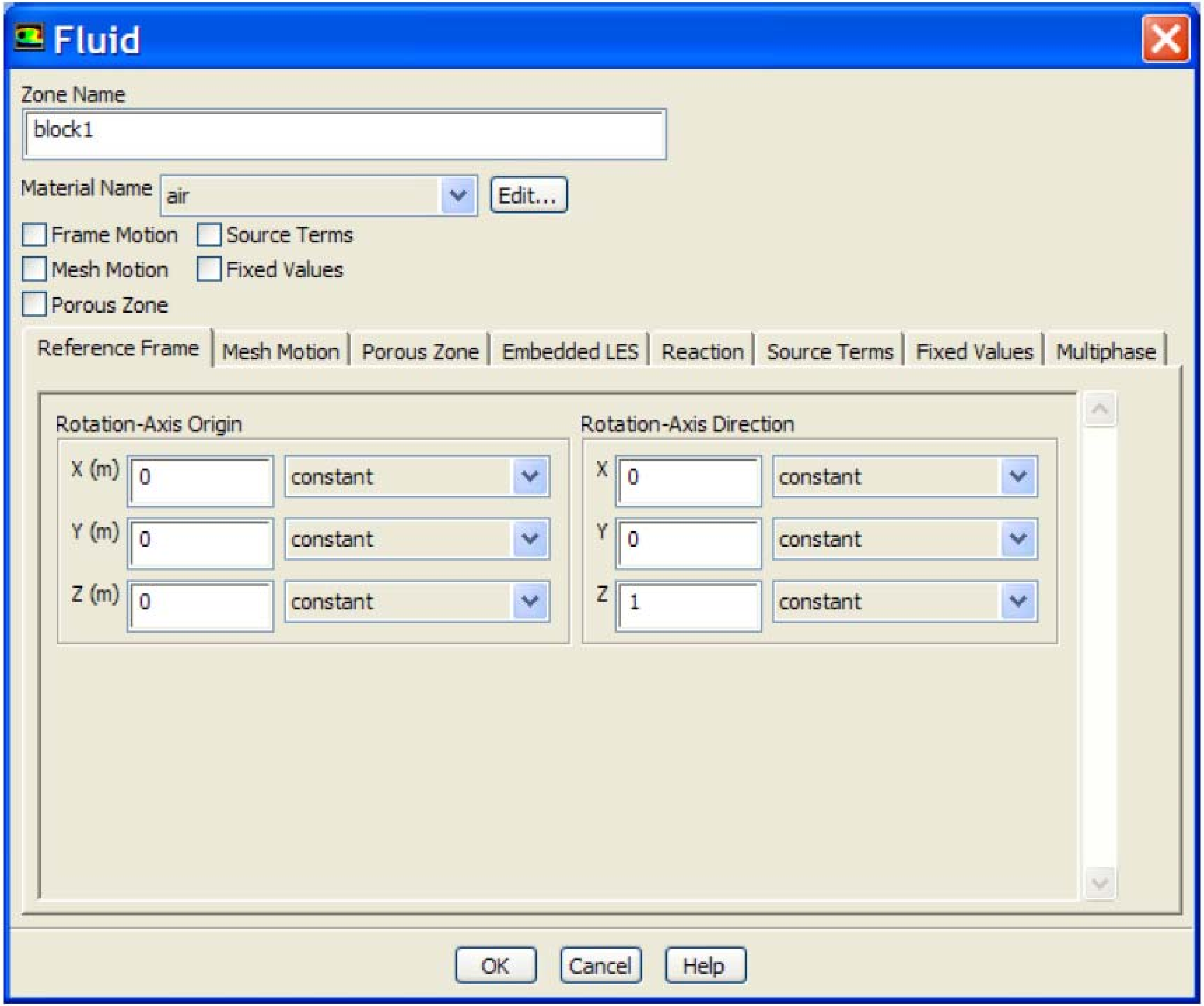
- Optional inputs

① Moving zones

② Porous region

③ Source terms

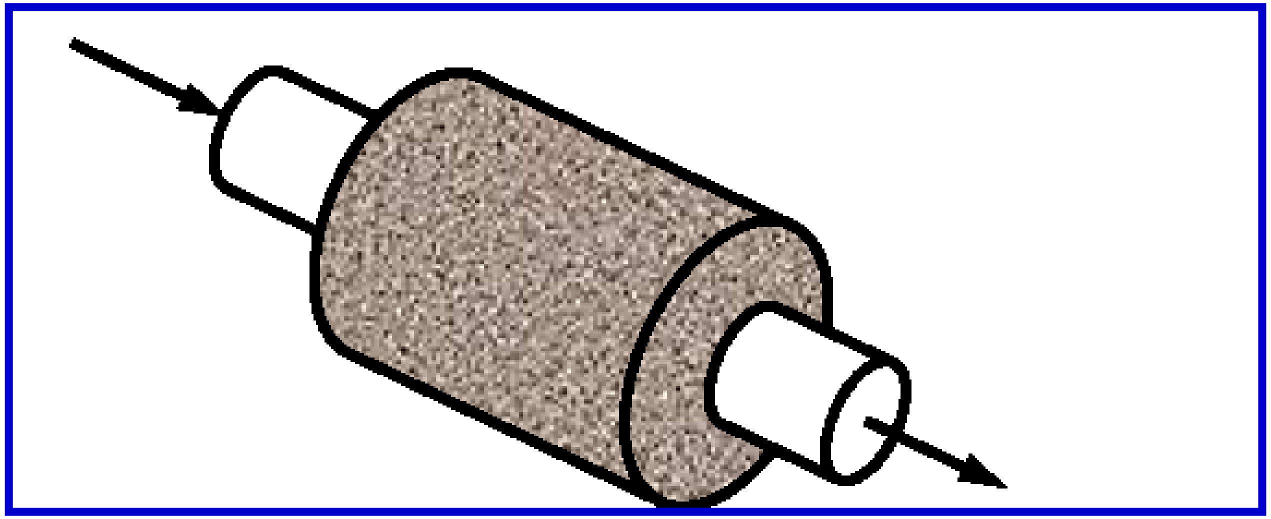
④ Fixed Values

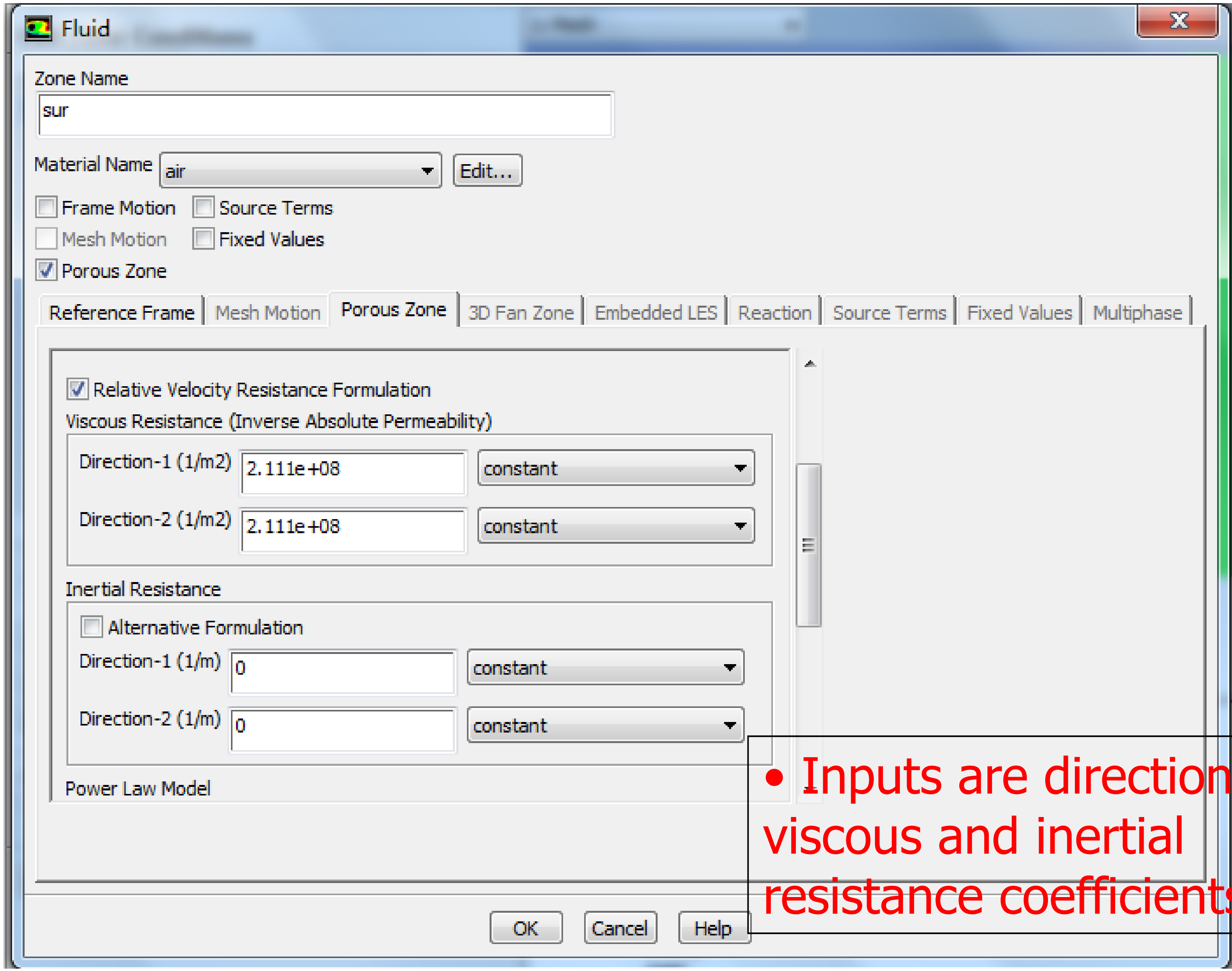

Fluid
Zone Name
block1
Material Name
air
Edit...
Frame Motion
Source Terms
Mesh Motion
Fixed Values
Porous Zone
Reference Frame
Mesh Motion
Porous Zone
Embedded LES
Reaction
Source Terms
Fixed Values
Multiphase
Rotation-Axis Origin
X (m)
0
constant
Y (m)
0
constant
Z (m)
0
constant
Rotation-Axis Direction
X
0
constant
Y
0
constant
Z
1
constant
OK
Cancel
Help

◆Cell Zones - Porous Media

• Some fluid regions are obviously porous and impossible to resolve exactly in a mesh:

– Packed beds，metal foam





- Inputs are directional viscous and inertial resistance coefficients.

◆ Cell Zones – Solid

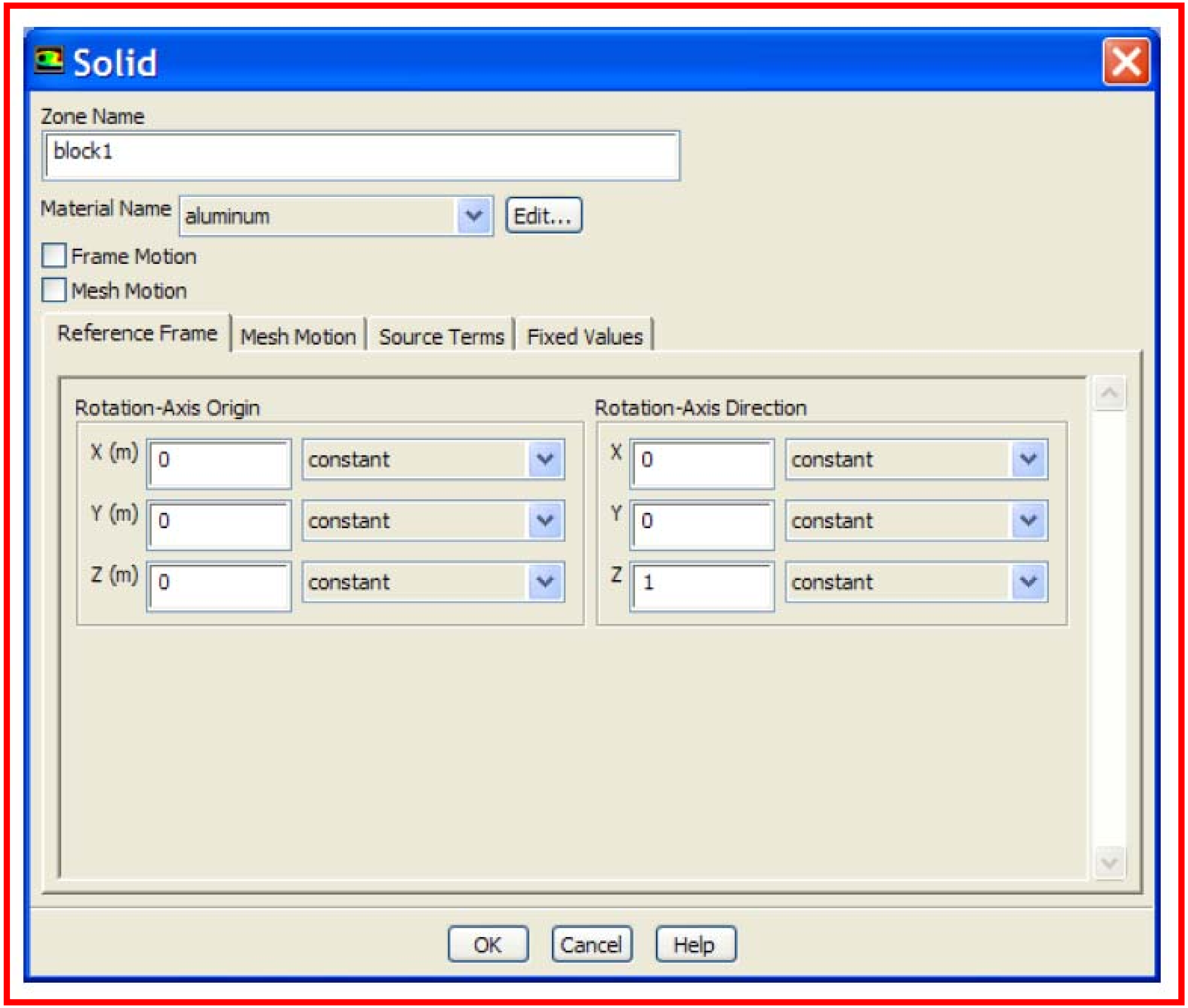
1) A solid zone is a group of cells for which only the energy equation is solved.

2) Only required input is the material name (defined in the Materials panel).

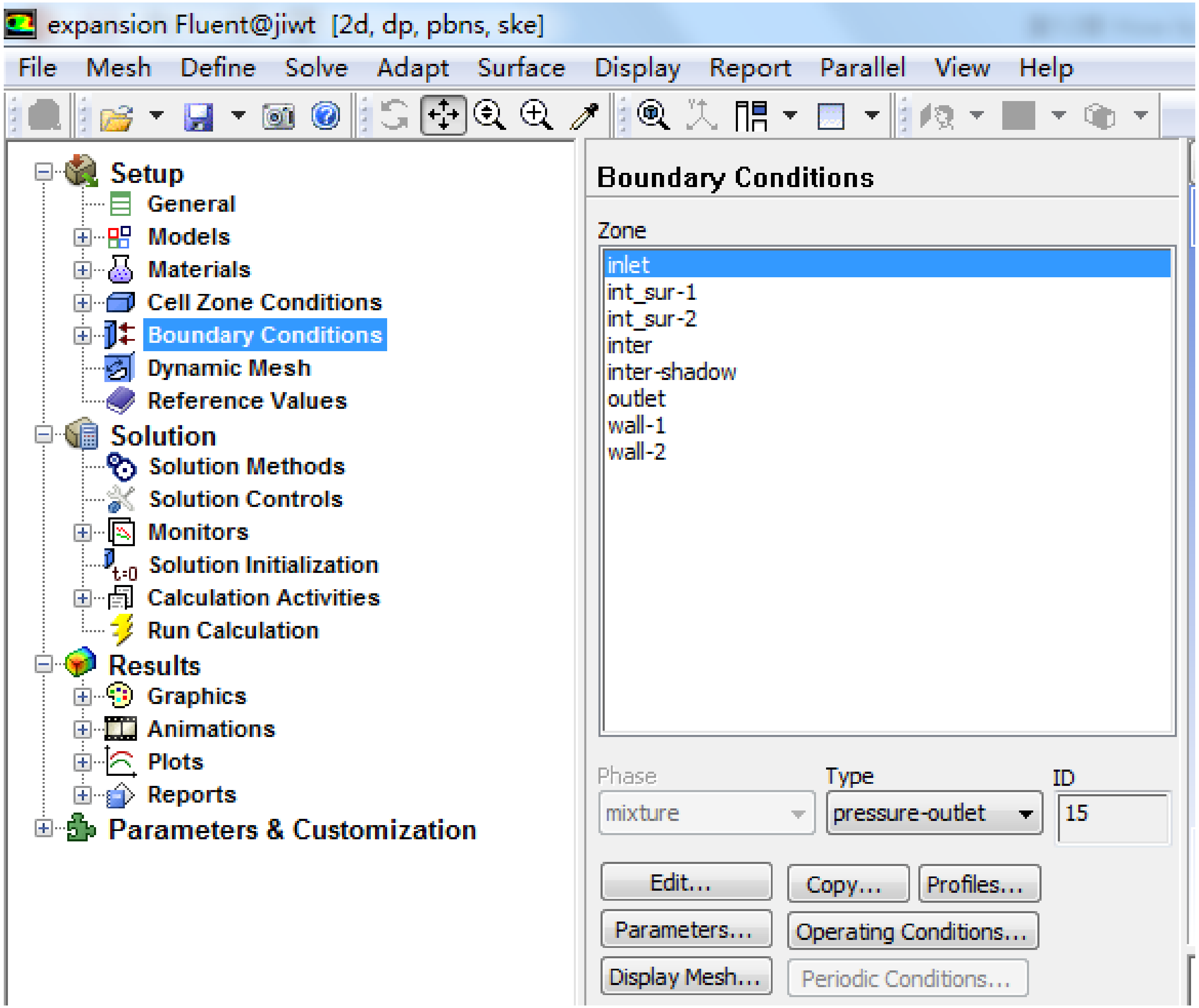
3) Optional inputs allow you to set volumetric heat generation rate(heat source).

4) Need to specify rotation axis if rotationally periodic boundaries adjacent to solid zone.

5) Can define motion for a solid zone


Solid
Zone Name
block1
Material Name
aluminum
Edit...
Frame Motion
Mesh Motion
Reference Frame
Mesh Motion
Source Terms
Fixed Values
Rotation-Axis Origin
X (m)
0
constant
Y (m)
0
constant
Z (m)
0
constant
Rotation-Axis Direction
X
0
constant
Y
0
constant
Z
1
constant
OK
Cancel
Help

5. Boundary Conditions



Boundary Conditions - Available Types

External Boundaries

1） General

- Pressure Inlet
- Pressure Outlet

2） Incompressible

- Velocity Inlet
- Outflow

3） Compressible

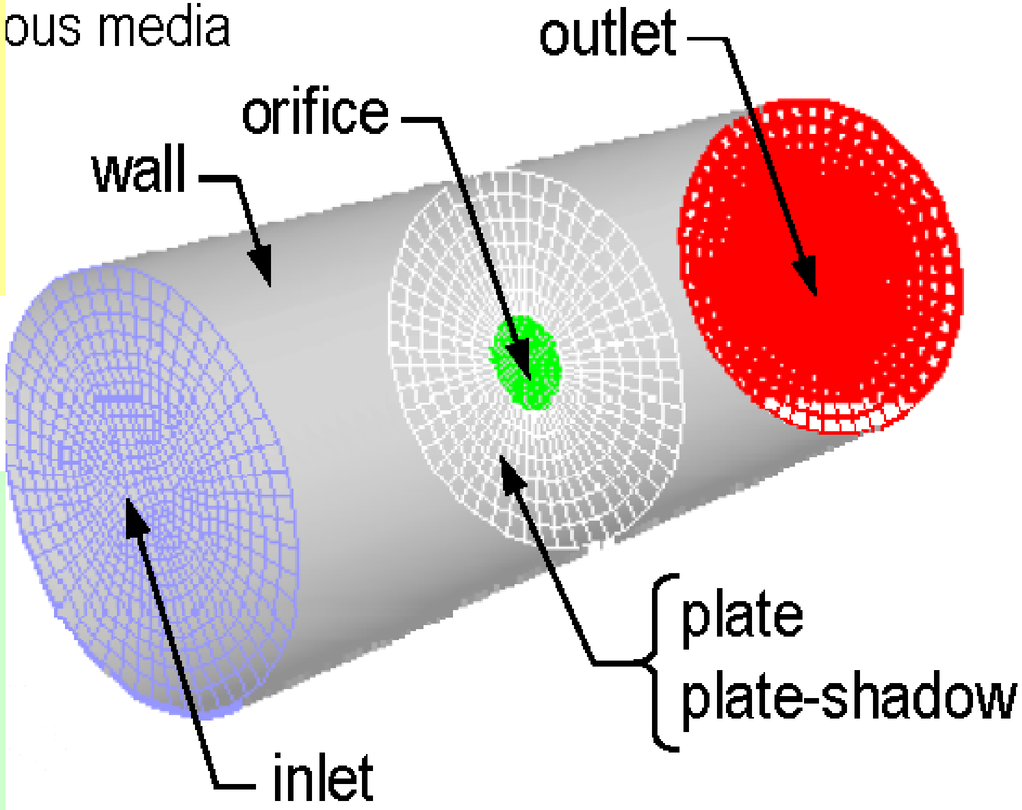
- Mass Flow Inlet
- Pressure Far Field

4） Other

- Wall
- Symmetry
- Axis
- Periodic

- Internal Boundaries
  - Fan
  - Interior
  - Radiator
  - Wall

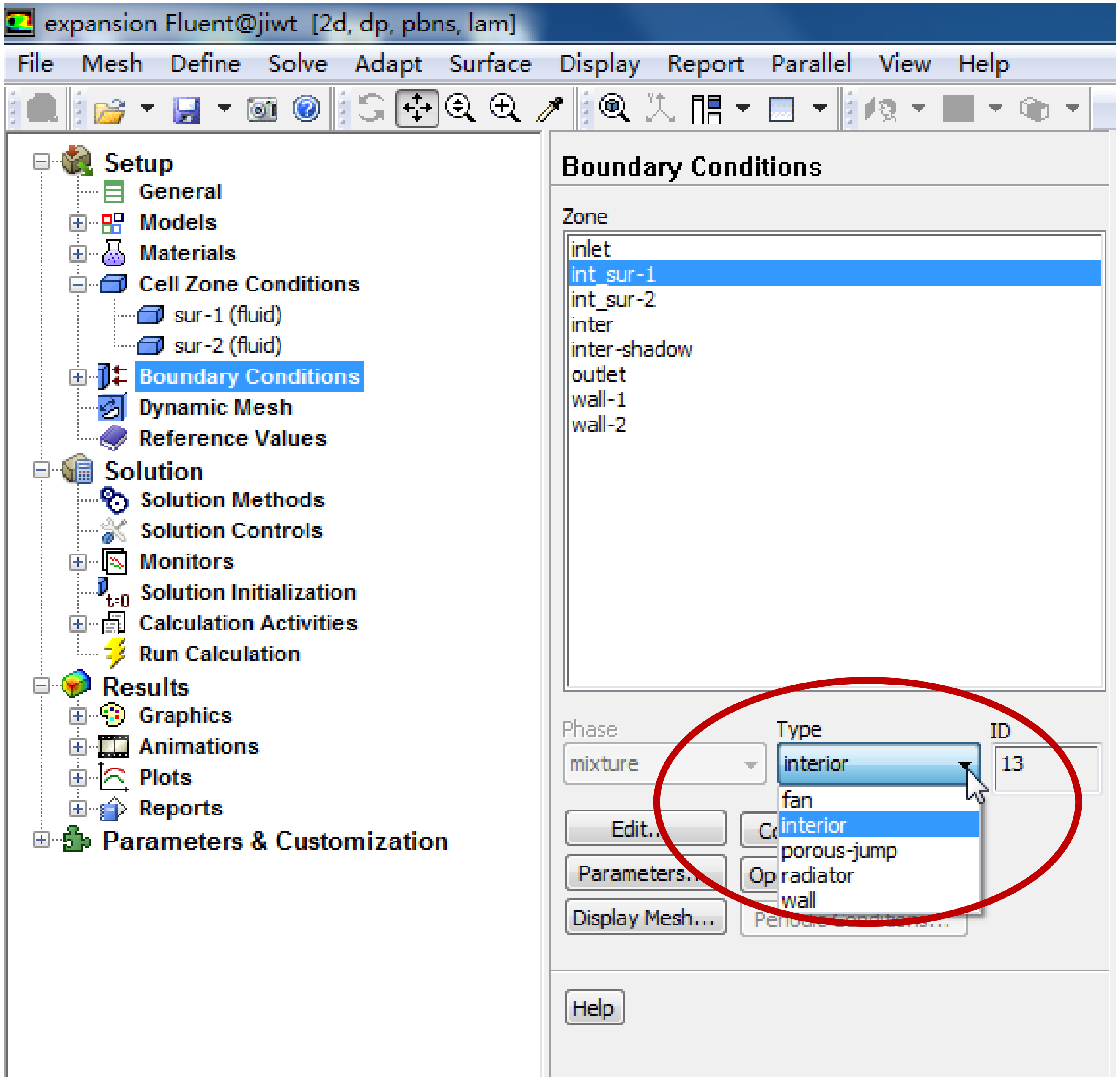
Cell (Continuum) zones
- Fluid
- Solid
- Porous media



◆ Boundary Conditions – Changing the Types

• Zones and zone types are initially defined in the preprocessing phase(ICEM).

• 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.


expansion Fluent@jiwt [2d, dp, pbns, lam]
File Mesh Define Solve Adapt Surface Display Report Parallel View Help
Setup
General
Models
Materials
Cell Zone Conditions
sur-1 (fluid)
sur-2 (fluid)
Boundary Conditions
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation
Results
Graphics
Animations
Plots
Reports
Parameters & Customization
Boundary Conditions
Zone
inlet
int_sur-1
int_sur-2
inter
inter-shadow
outlet
wall-1
wall-2
Phase
mixture
Type
interior
ID
13
fan
interior
porous-jump
radiator
wall
Edit...
Parameters...
Display Mesh...
Help

◆ Boundary Conditions - Velocity Inlet

1) Velocity Specification Method

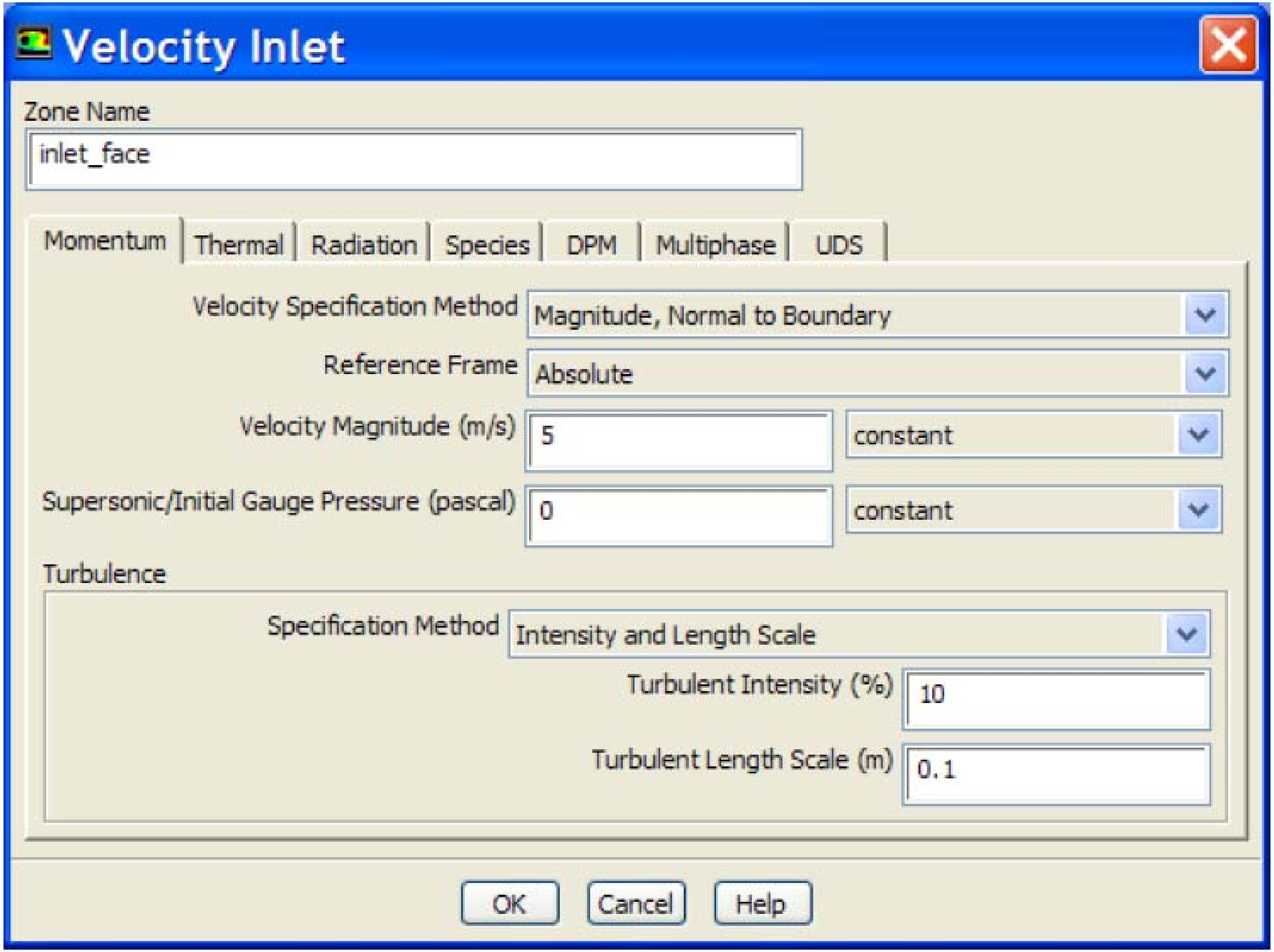
– Magnitude, Normal to Boundary

– Magnitude and Direction

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

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

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


Velocity Inlet
Zone Name
inlet_face
Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Velocity Specification Method
Magnitude, Normal to Boundary
Reference Frame
Absolute
Velocity Magnitude (m/s)
5
constant
Supersonic/Initial Gauge Pressure (pascal)
0
constant
Turbulence
Specification Method
Intensity and Length Scale
Turbulent Intensity (%)
10
Turbulent Length Scale (m)
0.1
OK
Cancel
Help

◆ Boundary Conditions - Pressure Inlet

1) Pressure inlets are suitable for both compressible and incompressible flows.

– FLUENT calculates static pressure and velocity at inlet (Dynamic pressure)

2) Required inputs

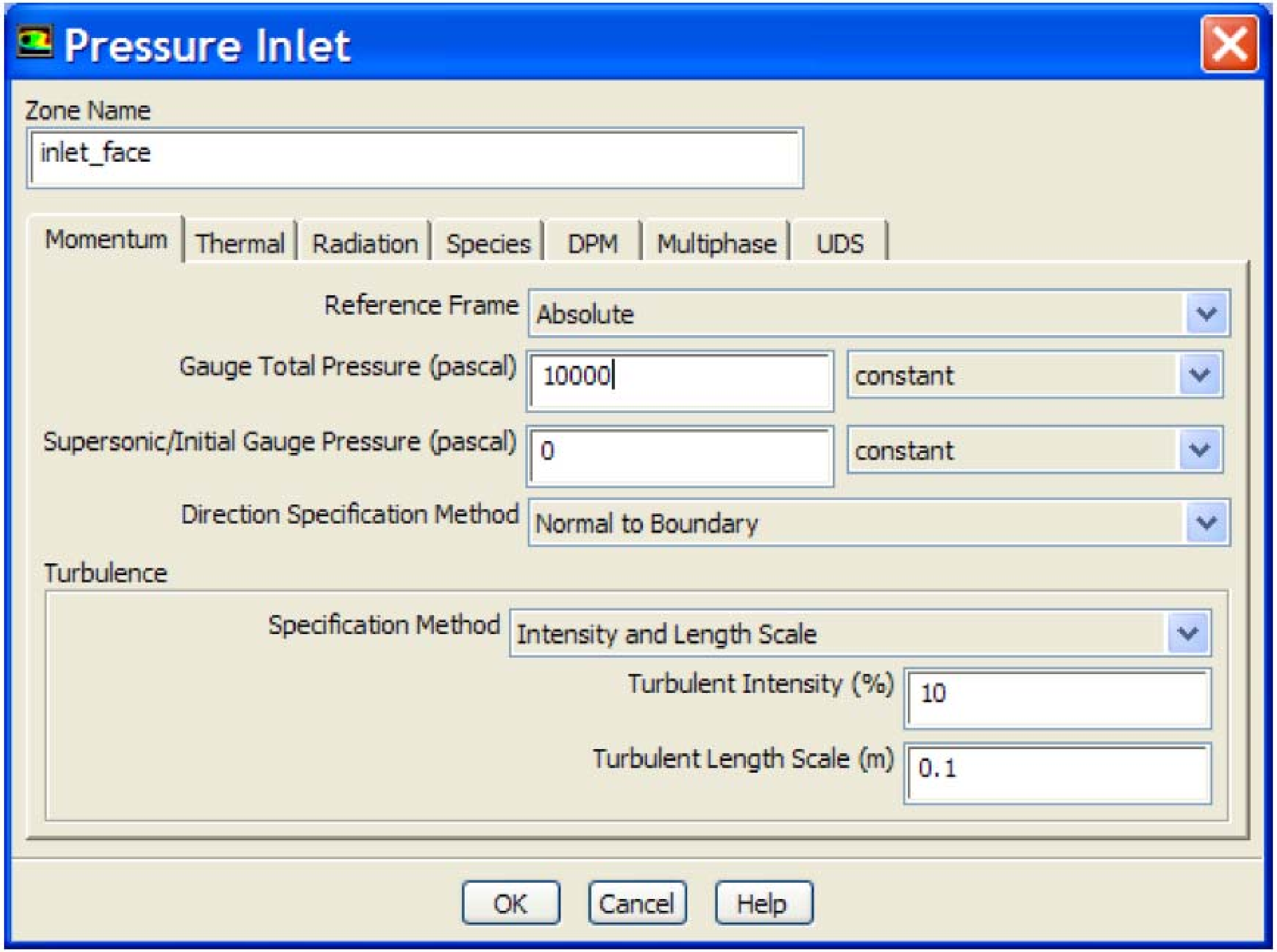
– Gauge Total Pressure

– Supersonic/Initial Gauge Pressure

– Inlet flow direction

– Turbulence quantities(if applicable)

– Total temperature (heat transfer or compressible).



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

◆ Boundary Conditions - Mass Flow Inlet

1)Mass flow inlets are intended for compressible flows; however, they can be used for incompressible flows.

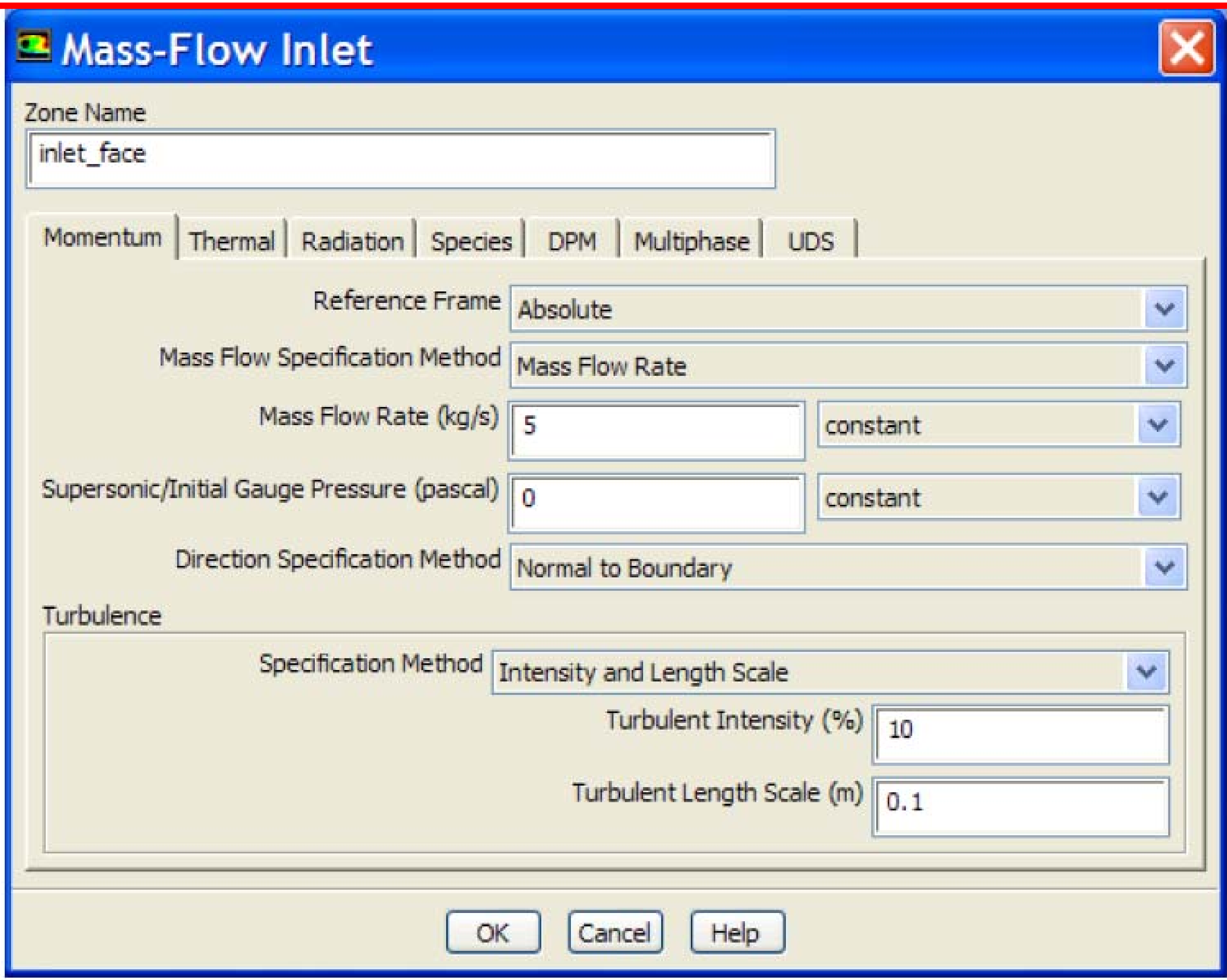
– Total pressure adjusts to accommodate mass flow inputs.

– More difficult to converge than pressure inlet.

2) Required information

– Mass Flow Rate or Mass Flux

– Supersonic/Initial Gauge Pressure


Mass-Flow Inlet
Zone Name
inlet_face
Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Reference Frame
Absolute
Mass Flow Specification Method
Mass Flow Rate
Mass Flow Rate (kg/s)
5
constant
Supersonic/Initial Gauge Pressure (pascal)
0
constant
Direction Specification Method
Normal to Boundary
Turbulence
Specification Method
Intensity and Length Scale
Turbulent Intensity (%)
10
Turbulent Length Scale (m)
0.1
OK
Cancel
Help

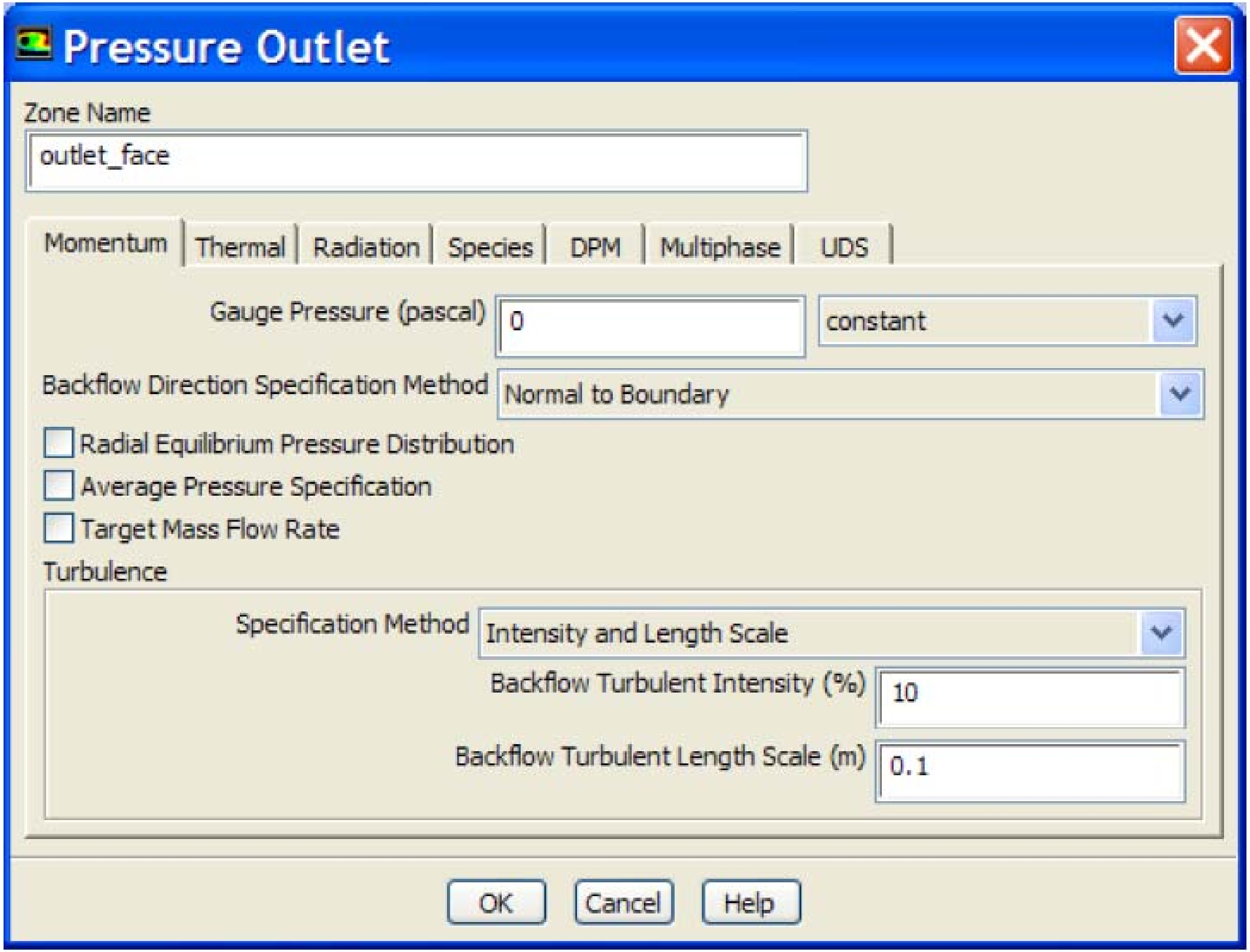
◆ Boundary Conditions - Pressure Outlet

1) Suitable for compressible and incompressible flows.

2) Required information

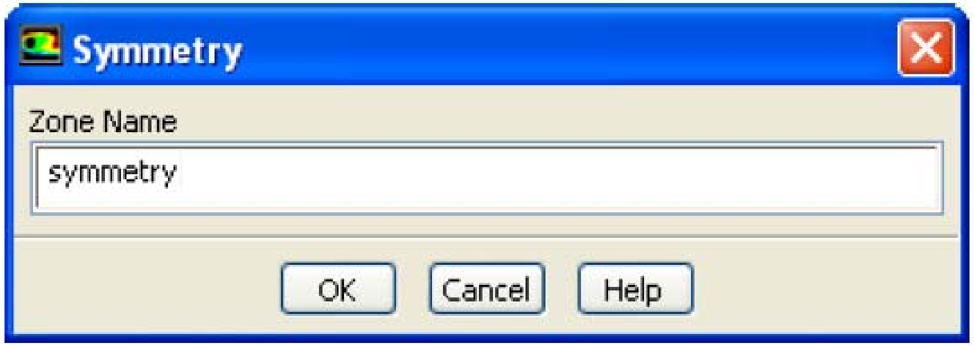
–Gauge Pressure (static)–static pressure of the environment into which the flow exits.

–Backflow quantities–Used as inlet conditions when backflow occurs (outlet acts like an inlet).

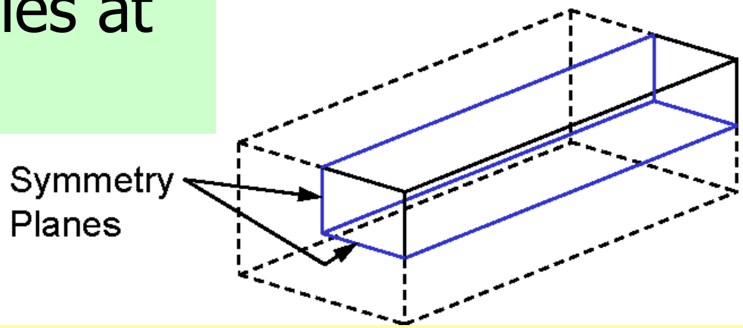

Pressure Outlet
Zone Name
outlet_face
Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Gauge Pressure (pascal)
0
constant
Backflow Direction Specification Method
Normal to Boundary
Radial Equilibrium Pressure Distribution
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method
Intensity and Length Scale
Backflow Turbulent Intensity (%)
10
Backflow Turbulent Length Scale (m)
0.1
OK
Cancel
Help

◆ 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



- Axis Boundary
  - the center axisymmetric problems Used at line for problems.
  - No user inputs required.

◆ Boundary Conditions - Periodic Boundaries

1) Used to reduce the overall mesh size.

2) Flow field and geometry must contain either rotational or translational periodicity.

3) Rotational periodicity

- ΔP=0 across periodic planes.
- Axis of rotation must be defined in fluid zone.

4) Translational periodicity

ΔP can be finite across periodic planes.

5) Rotationally periodic planes

Models fully developed conditions.
Specify either mean ΔP per period or net mass flow rate.

◆ 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

- Preferable over porous media for its better convergence behavior.
  - Interior walls

◆ 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).

◆ Boundary Conditions - Outflow

- 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.

◆ Wall Boundary Conditions

- Five thermal conditions

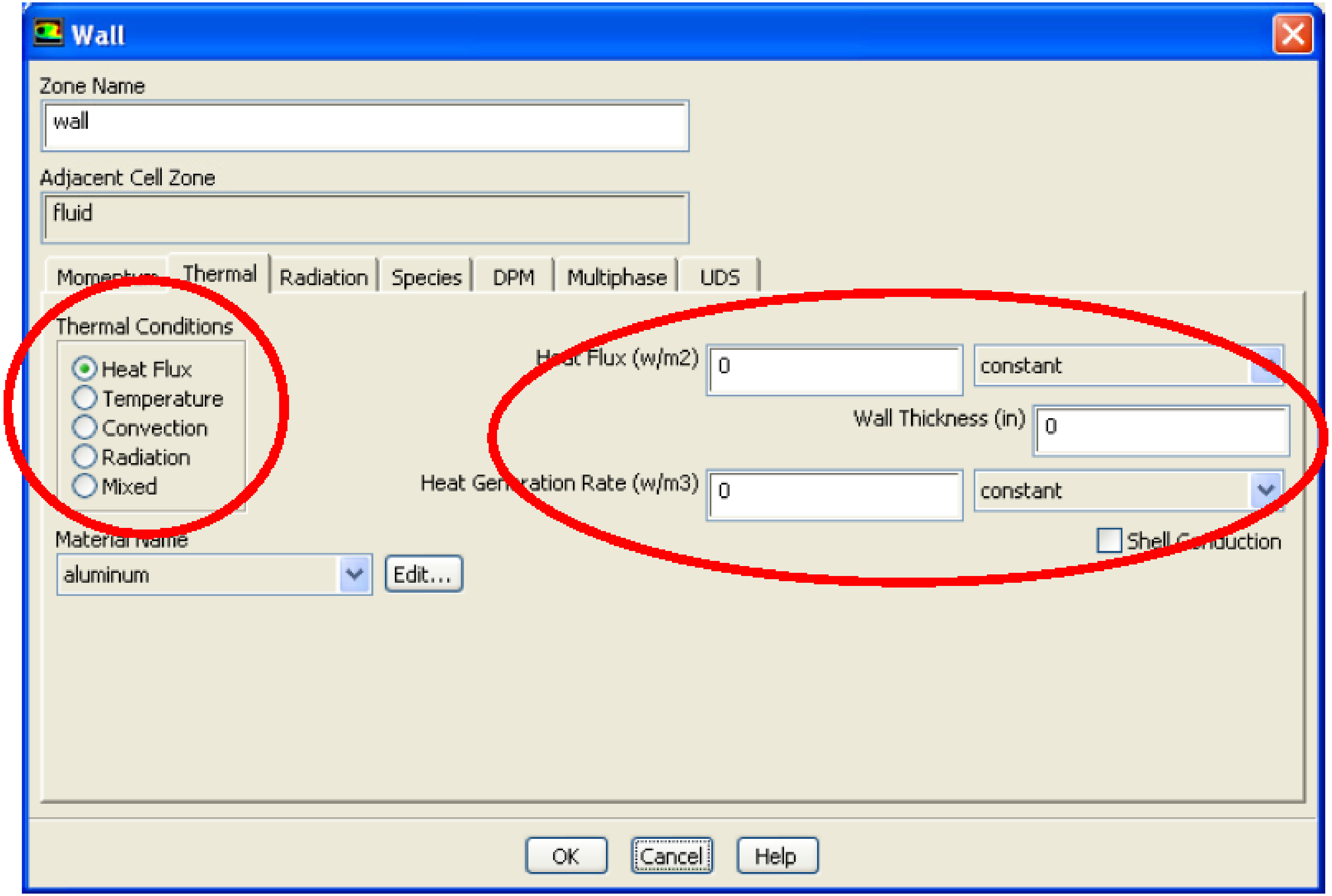
① Heat Flux

② Temperature

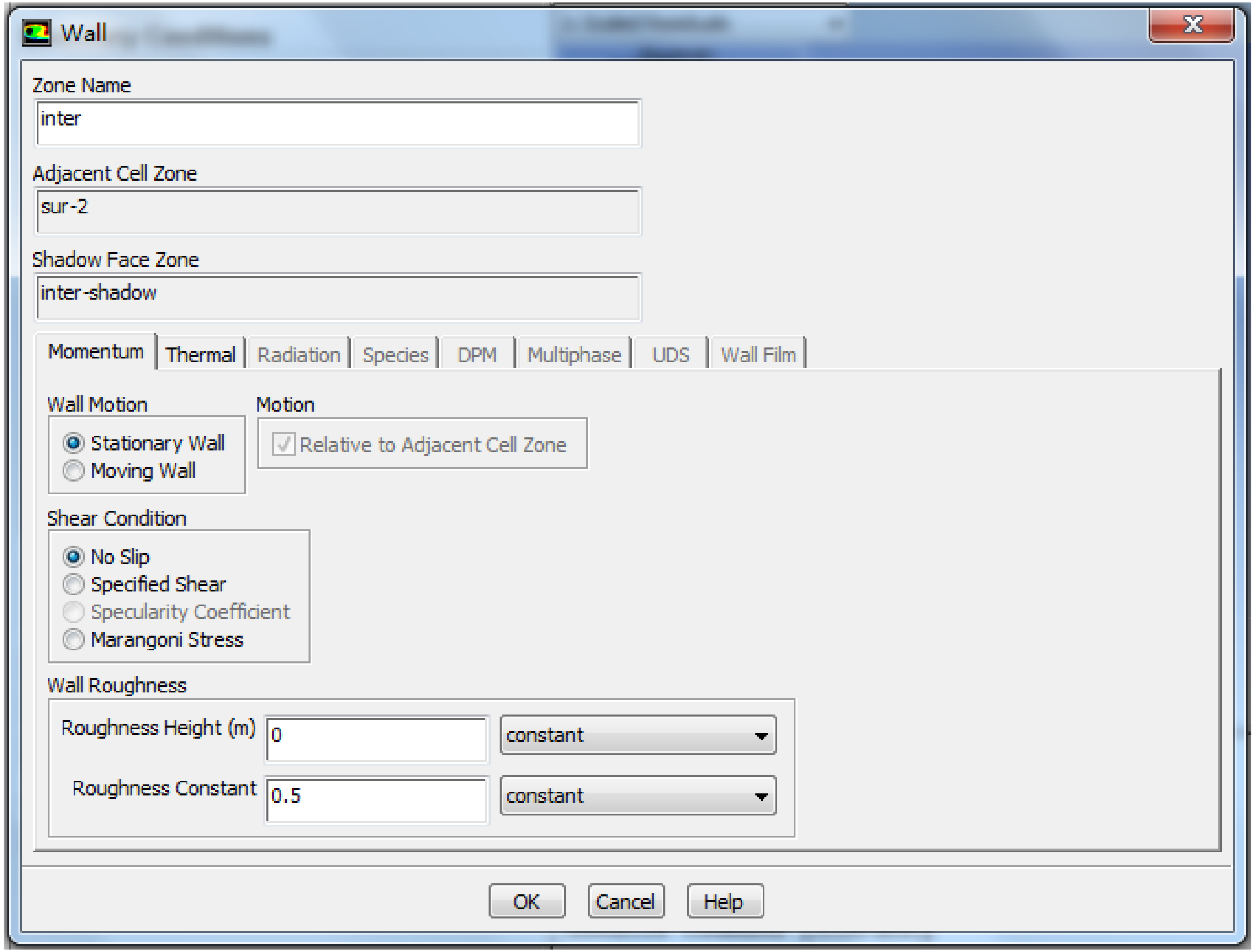
③ Convection–simulates an external convection environment which is not modeled (user-prescribed heat transfer coefficient).

④ Radiation – simulates an external radiation environment which is not modeled (user-prescribed external emissivity and radiation temperature).

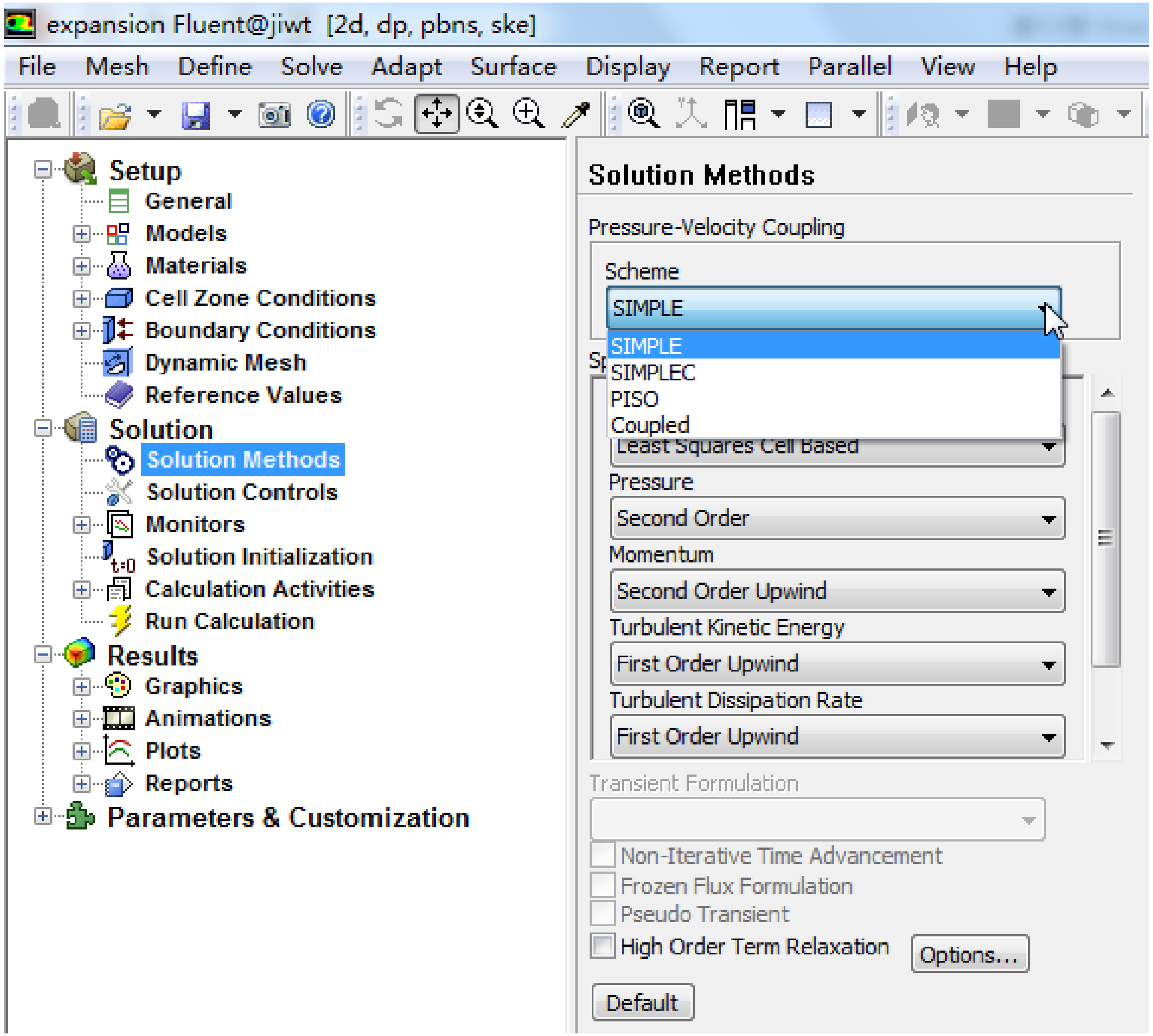
⑤ Mixed– Combination of Convection and Radiation boundary conditions.


Wall
Zone Name
wall
Adjacent Cell Zone
fluid
Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Thermal Conditions
Heat Flux
Temperature
Convection
Radiation
Mixed
Heat Flux (w/m2)
0
constant
Wall Thickness (in)
0
Heat Generation Rate (w/m3)
0
constant
Shell Conduction
Material Name
aluminum
Edit...
OK
Cancel
Help

Momentum conditions



6. Solution Methods



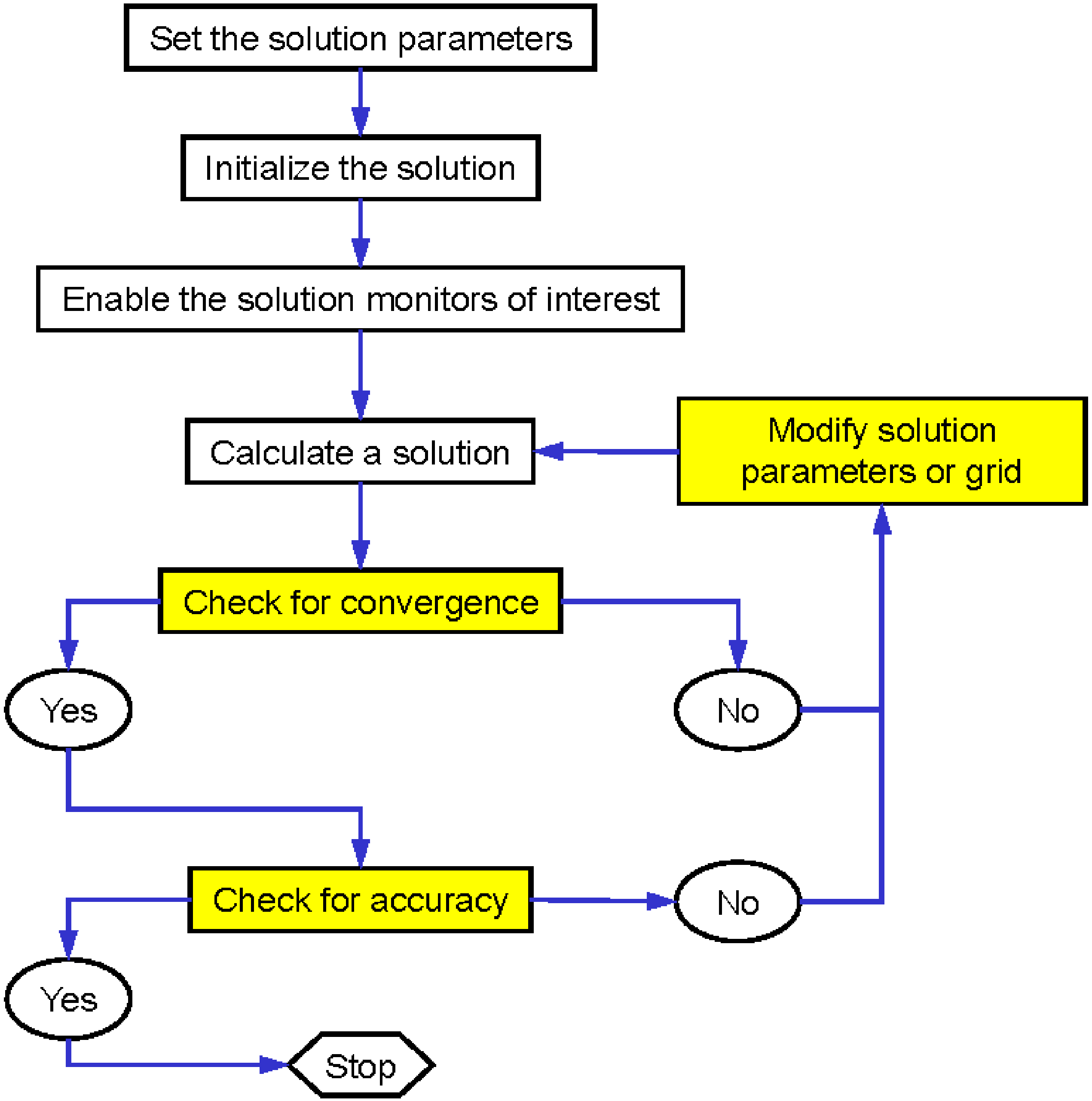
Solver Settings

• By modifying the solver settings you can improve both:

– The rate of convergence of the simulation. (Chapter 6 求解椭圆形流动)
– The accuracy of the computed result.(Chapter 5 对流-扩散方程的离散格式)

To Consider:

– the choice of solver
– discretisation schemes
– checking convergence
– assessing accuracy


Set the solution parameters
Initialize the solution
Enable the solution monitors of interest
Calculate a solution
Modify solution parameters or grid
Check for convergence
Yes
No
Check for accuracy
No
Yes
Stop

◆ Available Solvers

- There are two kinds of solvers available in FLUENT:
  - – Pressure based
  - – Density based

The **pressure-based** solvers take momentum and pressure (or pressure correction) as the primary variables（such as SIMPLE Algorithm）

- Two algorithms are available with the pressure-based solvers:

– Segregated solver – Solves for pressure correction and momentum sequentially.(SIMPLE, SIMPLC,PISO)
– Coupled Solver (PBCS) – Solves pressure and momentum simultaneously(COUPLED).

◆ Available Solvers

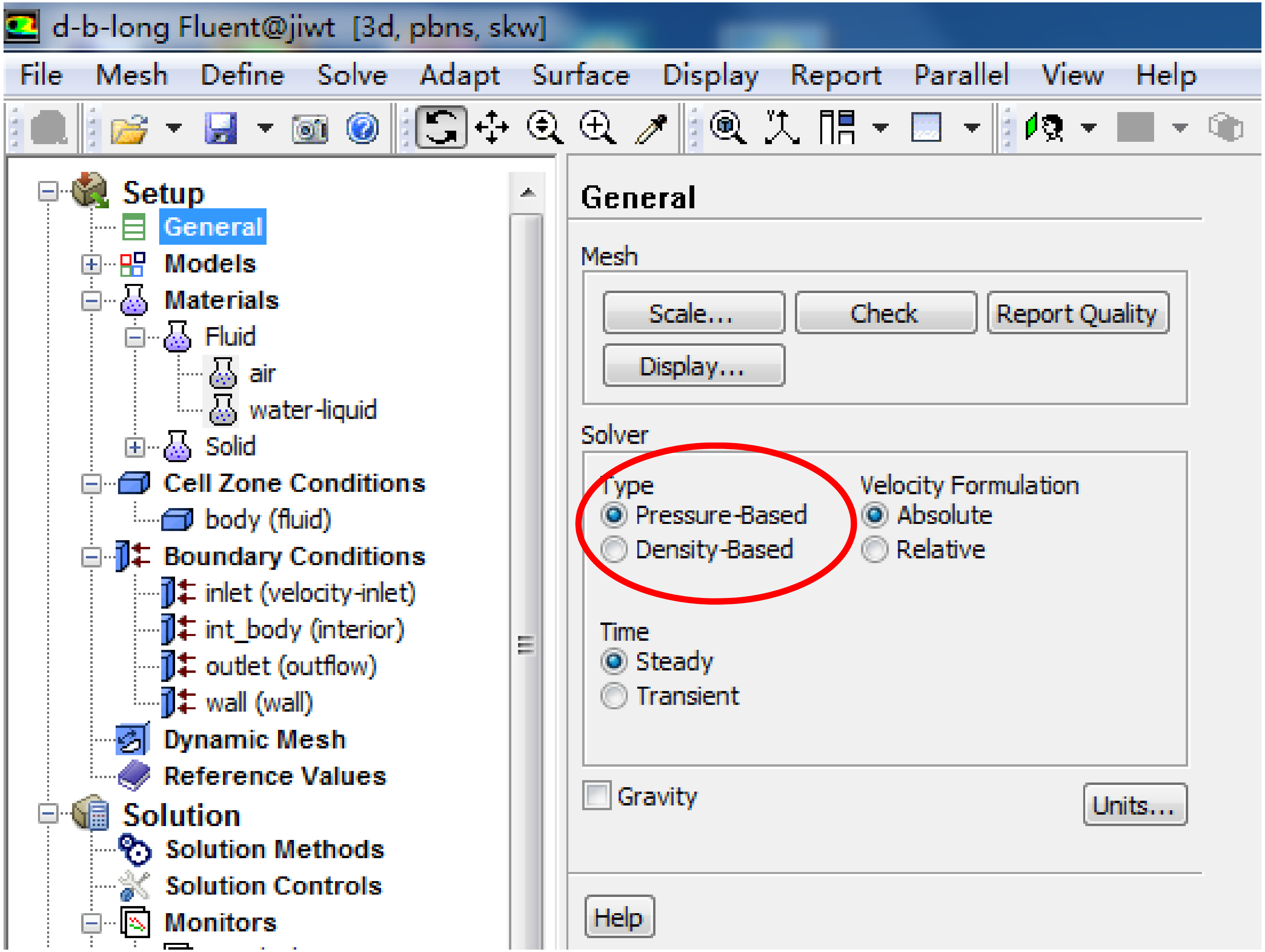
- Density-Based Coupled Solver

  – Equations for continuity, momentum, energy and species (if required) are solved in vector form.

  – Pressure is obtained through an equation of state.

  – Additional scalar equations are solved in a segregated fashion.

- The Density-Based Coupled Solver can be run either explicit or implicit.


d-b-long Fluent@jiwt [3d, pbns, skw]
File Mesh Define Solve Adapt Surface Display Report Parallel View Help
Setup
General
Models
Materials
Fluid
air
water-liquid
Solid
Cell Zone Conditions
body (fluid)
Boundary Conditions
inlet (velocity-inlet)
int_body (interior)
outlet (outflow)
wall (wall)
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
General
Mesh
Scale...
Check
Report Quality
Display...
Solver
Type
Pressure-Based
Density-Based
Velocity Formulation
Absolute
Relative
Time
Steady
Transient
Gravity
Units...
Help

◆ Choosing a Solver

• The pressure-based solver is applicable for a wide range of flow regimes from low speed incompressible flow to high-speed compressible flow.

– Requires less memory (storage).

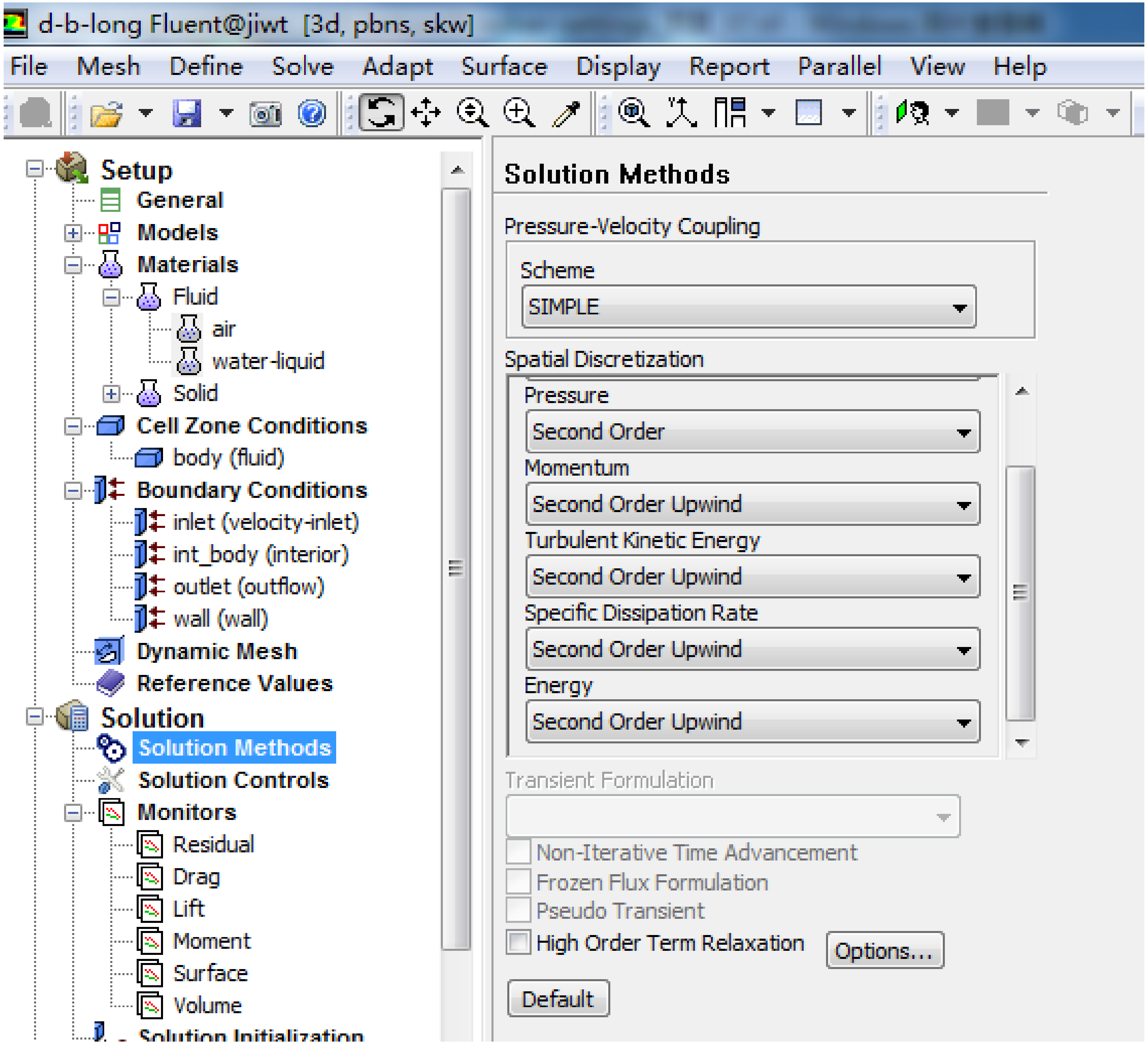
– Allows flexibility in the solution procedure.

• The pressure-based coupled solver (PBCS) is applicable for most single phase flows, and yields superior performance to the standard pressure-based solver.

– Requires 1.5–2 times more memory than the segregated solver.

- **The density-based coupled solver (DBCS)** is applicable when there is a strong coupling, or interdependence, between density, energy, momentum, and/or species.

– Examples: High speed compressible flow with combustion, hypersonic flows, shock interactions.
– The implicit option is generally preferred over explicit since it has a very strict limit on time step size
– The explicit approach is used for cases where the characteristic time scale of the flow is on the same order as the acoustic time scale. (e.g. propagation of high-Ma shock waves).


d-b-long Fluent@jiwt [3d, pbns, skw]
File Mesh Define Solve Adapt Surface Display Report Parallel View Help
Setup
General
Models
Materials
Fluid
air
water-liquid
Solid
Cell Zone Conditions
body (fluid)
Boundary Conditions
inlet (velocity-inlet)
int_body (interior)
outlet (outflow)
wall (wall)
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Residual
Drag
Lift
Moment
Surface
Volume
Solution Initialization
Solution Methods
Pressure-Velocity Coupling
Scheme
SIMPLE
Spatial Discretization
Pressure
Second Order
Momentum
Second Order Upwind
Turbulent Kinetic Energy
Second Order Upwind
Specific Dissipation Rate
Second Order Upwind
Energy
Second Order Upwind
Transient Formulation
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
High Order Term Relaxation
Options...
Default

◆ Interpolation schemes for the convection term(Chapter 5):

1)**First-Order Upwind**–Easiest to converge, only first-order accurate.

2)**Power Law** – More accurate than first-order for flows when Re_{cell} <5 (typ. low Re flows)

3)**Second-Order Upwind** – Uses larger stencils for 2nd order accuracy, essential with tri/tet mesh or when flow is not aligned with grid; convergence may be slower.

4)**Monotone Upstream-Centered Schemes for Conservation Laws (MUSCL)** – Locally 3rd order convection discretisation scheme for unstructured meshes; more accurate in predicting secondary flows, vortices, forces, etc.

5)**Quadratic Upwind Interpolation (QUICK)**–Applies to quad/hex and hybrid meshes, useful for rotating/swirling flows, 3rd-order accurate on uniform mesh.

Interpolation Methods (Gradients)

• Gradients of solution variables are required in order to evaluate diffusive fluxes, velocity derivatives, and for higher-order discretisation schemes.

• The gradients of solution variables at cell centers can be determined using three approaches:

–**Green-Gauss Cell-Based**– Least computationally intensive. Solution may have false diffusion.

–**Green-Gauss Node-Based**–More accurate/computation-ally intensive; minimizes false diffusion; recommended for unstructured meshes.

–**Least-Squares Cell-Based**–Default method; has the same accuracy and properties as Node-based Gradients and is less computationally intensive.

Interpolation Methods for Pressure

• Interpolation schemes for calculating cell-face pressures when using the pressure-based solver in FLUENT are available as follows:

1)Standard–The default scheme; reduced accuracy for flows exhibiting large surface-normal pressure gradients near boundaries

2) PRESTO! – Use for highly swirling flows, flows involving steep pressure gradients, or in strongly curved domains

3)Linear – Use when other options result in convergence difficulties or unphysical behavior

3)Second-Order – Use for compressible flows; not to be used with porous media, jump, fans, etc. or VOF/Mixture multiphase models

4) Body Force Weighted – Use when body forces are large, e.g., high Ra natural convection or highly swirling flows

◆ Pressure-Velocity Coupling

• Pressure-velocity coupling refers to the numerical algorithm which uses a combination of continuity and momentum equations to derive an equation for pressure (or pressure correction) when using the pressure-based solver.

• Five algorithms are available in FLUENT.

Semi-Implicit Method for Pressure-Linked Equations (SIMPLE)

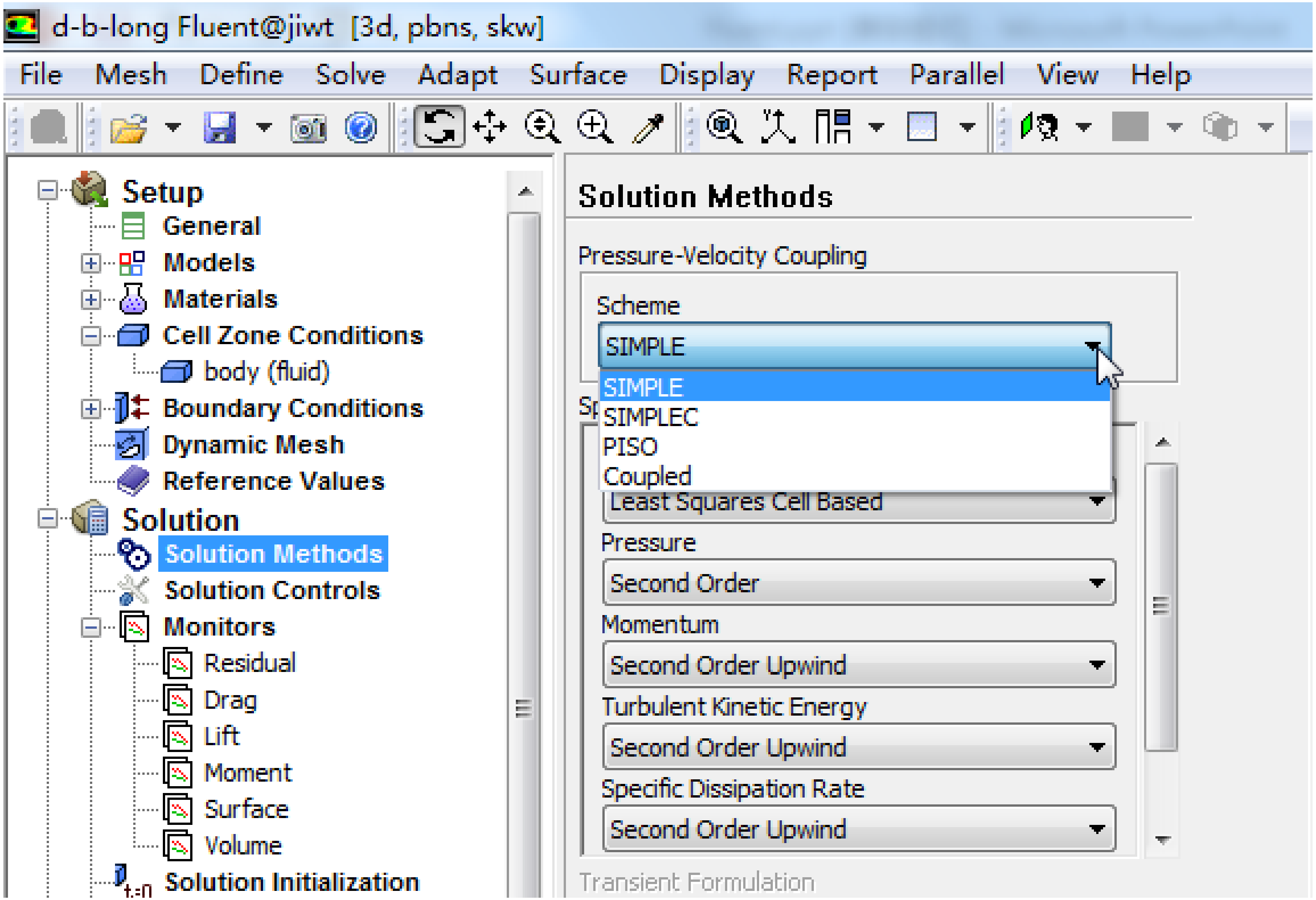
• The default scheme, robust

SIMPLE-Consistent (SIMPLEC)

• Allows faster convergence for simple problems (e.g., laminar flows with no physical models employed).

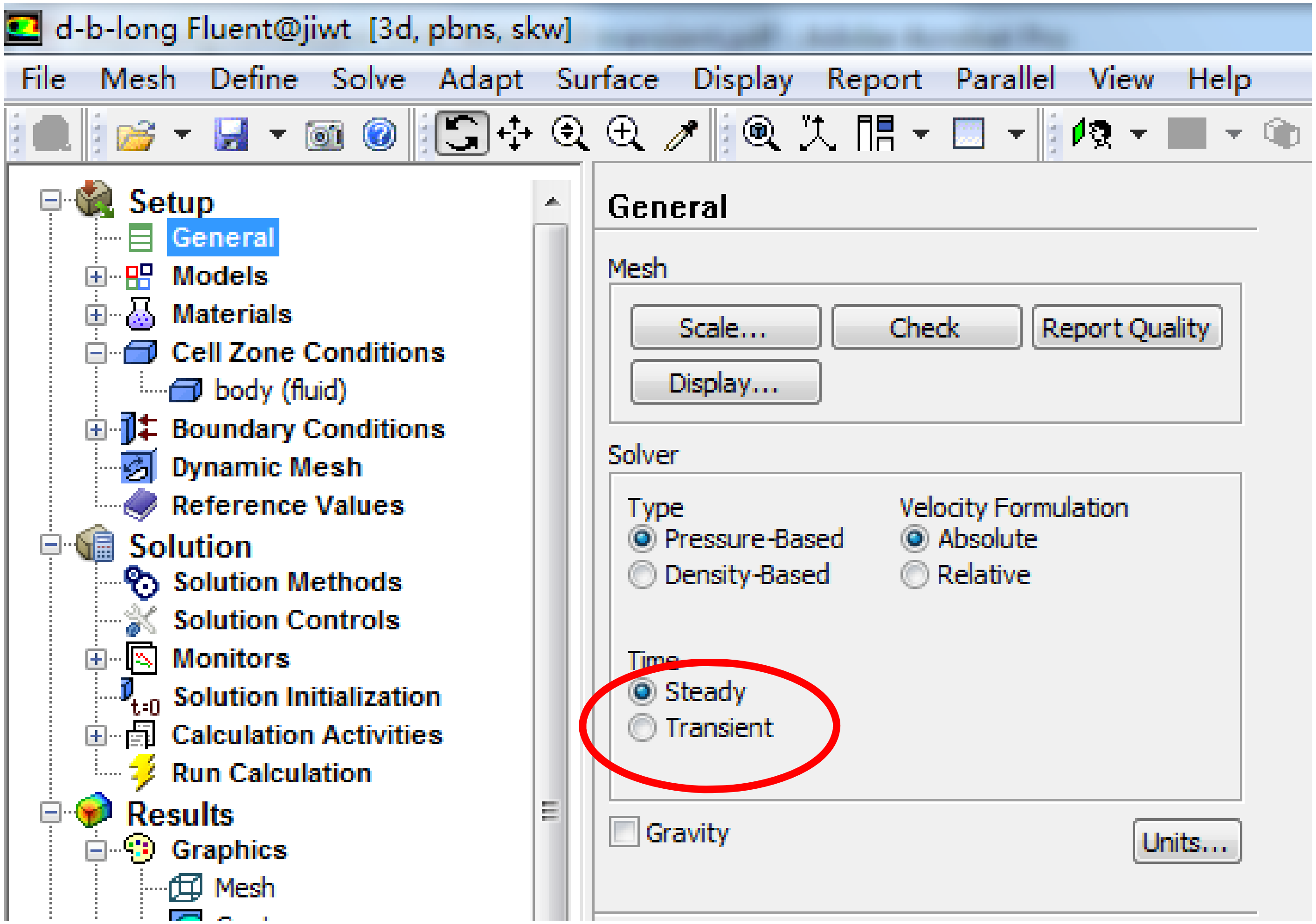
Pressure-Implicit with Splitting of Operators (PISO)

• Useful for unsteady flow problems or for meshes containing cells with higher than average skewness


d-b-long Fluent@jiwt [3d, pbns, skw]
File
Mesh
Define
Solve
Adapt
Surface
Display
Report
Parallel
View
Help
Setup
General
Models
Materials
Cell Zone Conditions
body (fluid)
Boundary Conditions
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Residual
Drag
Lift
Moment
Surface
Volume
Solution Initialization
Solution Methods
Pressure-Velocity Coupling
Scheme
SIMPLE
SIMPLE
SIMPLEC
PISO
Coupled
Least Squares Cell Based
Pressure
Second Order
Momentum
Second Order Upwind
Turbulent Kinetic Energy
Second Order Upwind
Specific Dissipation Rate
Second Order Upwind
Transient Formulation

◆ Enabling the Transient Solver

- To enable the transient solver, select the Transient button on the General problem setup form:



• Before performing iterations, you will need to set some additional controls.
  – Solver settings
  – Animations
  – Data export/Autosave options

Selecting the Transient Time Step Size

• Time step size, *Δt*, is set in the Run Calculation form.
  – Δt must be small enough to resolve time-dependent features; make sure convergence is reached within the number of Max Iterations per Time Step
  – The order or magnitude of an appropriate time step size can be estimated as:

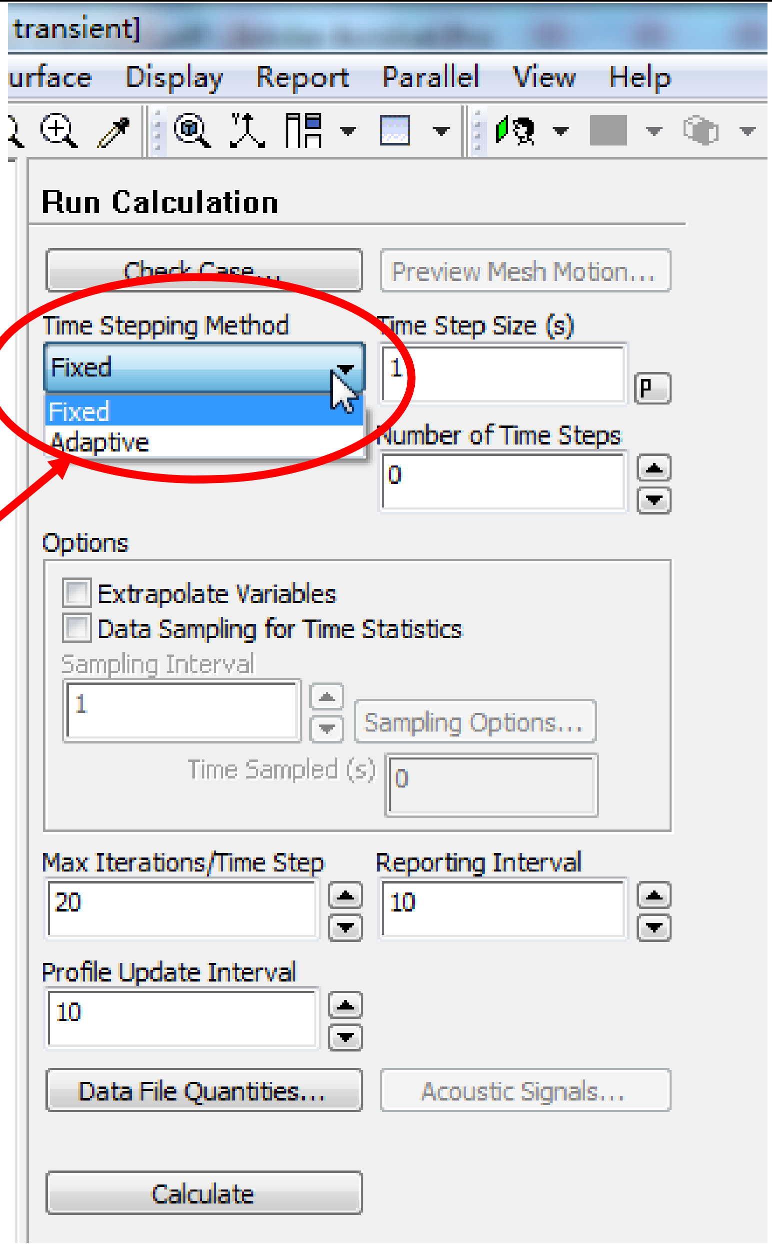
$$\boxed{\Delta t \approx \frac{\text{Typical cell size}}{\text{Characteristic flow velocity}}}$$

Time step size estimate can also be chosen so that the transient characteristics of the flow can be resolved (e.g. flow within a known period of fluctuations)

1) To iterate without advancing in time, specify zero time steps. This will instruct the solver to converge the current time step only.
2) The PISO scheme may aid in accelerating convergence for many transient flows form).

Transient Flow Modeling Options

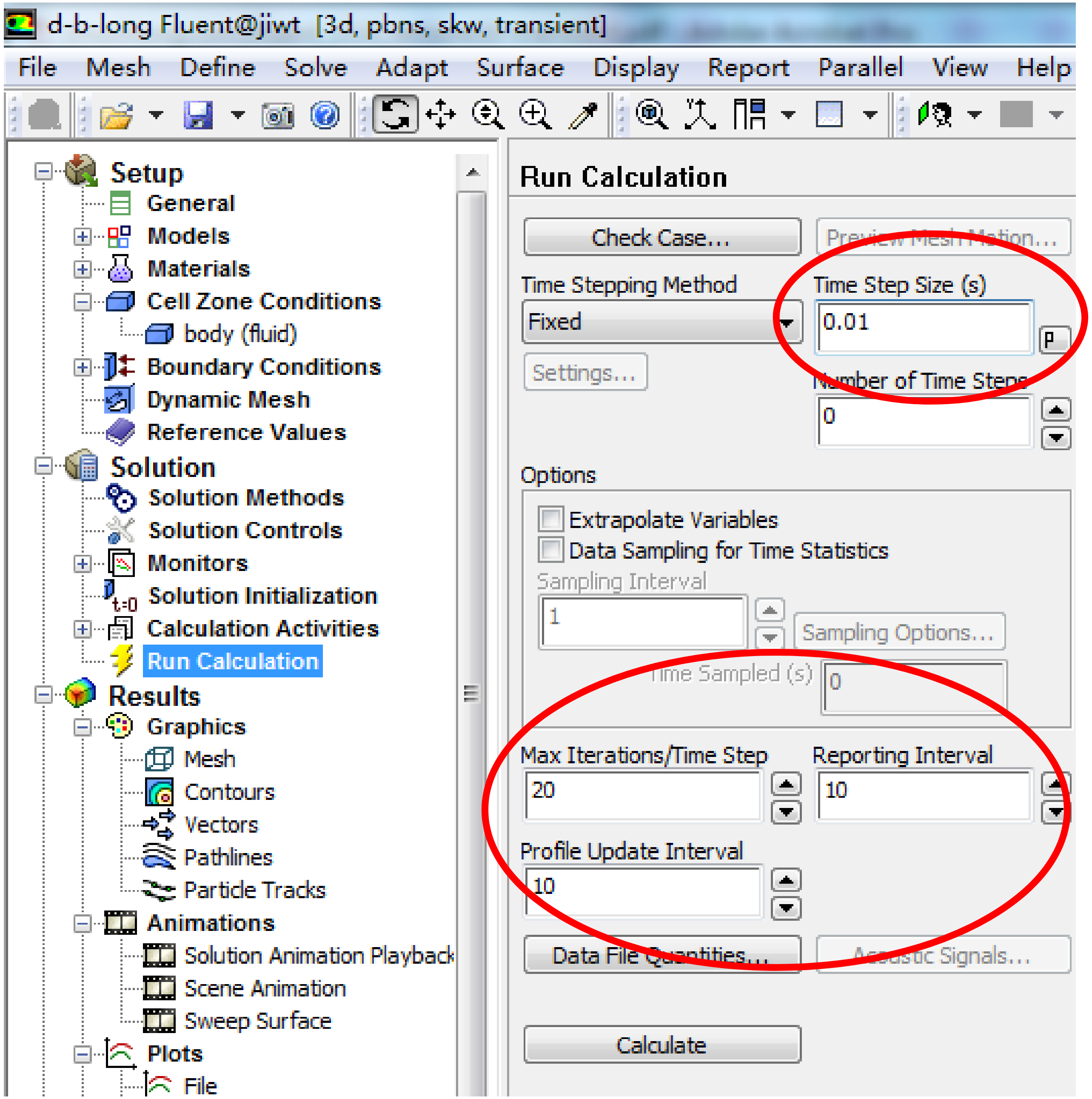
- Adaptive Time Stepping
  - Automatically adjusts time-step size based on local truncation error analysis



◆ Performing Iterations

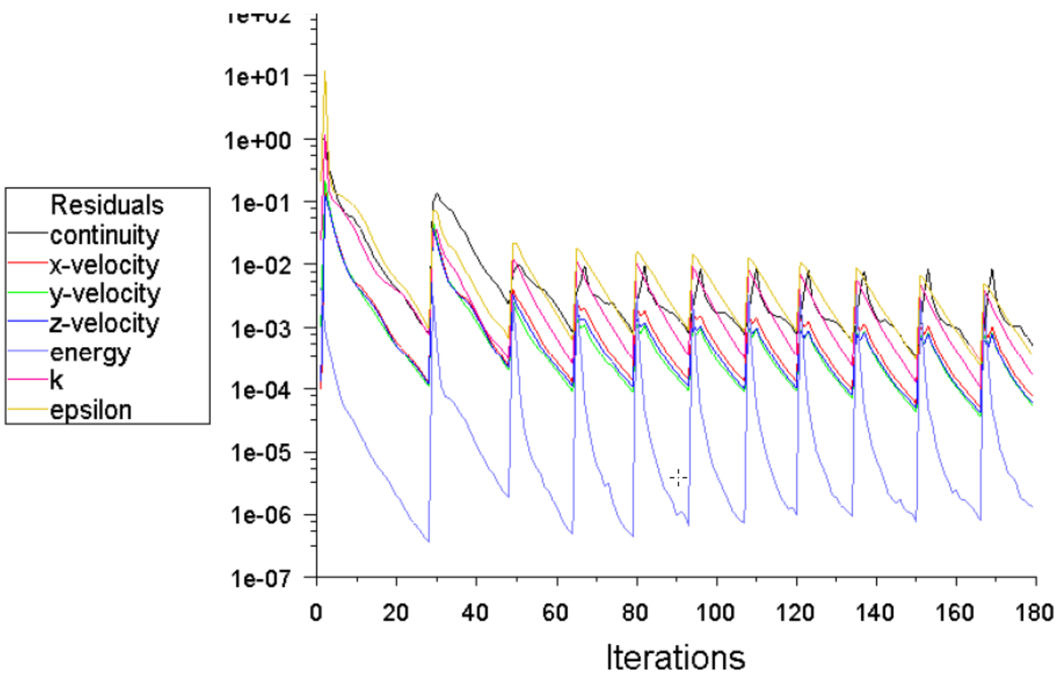
- The most common time advancement scheme is the iterative scheme.
  - The solver converges the current time step and then advances time.
  - Time is advanced when Max Iterations/Time Step is reached or convergence criteria are satisfied.
  - Time steps are converged sequentially until the Number of Time Steps is reached.

- Solution initialization defines the initial condition and it must be realistic.
  - Sets both the initial mass of fluid in the domain and the initial state of the flow field.


d-b-long Fluent@jiwt [3d, pbns, skw, transient]
File Mesh Define Solve Adapt Surface Display Report Parallel View Help
Setup
General
Models
Materials
Cell Zone Conditions
body (fluid)
Boundary Conditions
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation
Results
Graphics
Mesh
Contours
Vectors
Pathlines
Particle Tracks
Animations
Solution Animation Playback
Scene Animation
Sweep Surface
Plots
File
Run Calculation
Check Case...
Time Stepping Method
Fixed
Settings...
Time Step Size (s)
0.01
Number of Time Steps
0
Options
Extrapolate Variables
Data Sampling for Time Statistics
Sampling Interval
1
Sampling Options...
Time Sampled (s)
0
Max Iterations/Time Step
20
Reporting Interval
10
Profile Update Interval
10
Data File Quantities...
Acoustic Signals...
Calculate

◆ Convergence Behavior

• Residual plots for transient simulations are not always indicative of a converged solution.
• You should select the time step size such that the residuals reduce by around three orders of magnitude within one time step.
– This will ensure accurate resolution of transient behavior.



◆ Tips for Success in Transient Flow Modeling

• Use PISO scheme for Pressure-Velocity Coupling – this scheme provides faster convergence for transient flows than the standard SIMPLE approach.

• Select the time step size so that the solution converges three orders of magnitude for each time step (of course, convergence behavior is problem-specific).

• Select the number of iterations per time step to be around 20 – it is better to reduce the time step size than to do too many iterations per time step.

• Remember that accurate initial conditions are just as important as boundary conditions for transient problems – initial condition should always be physically realistic!

• Configure any animations you wish to see before running the calculations.

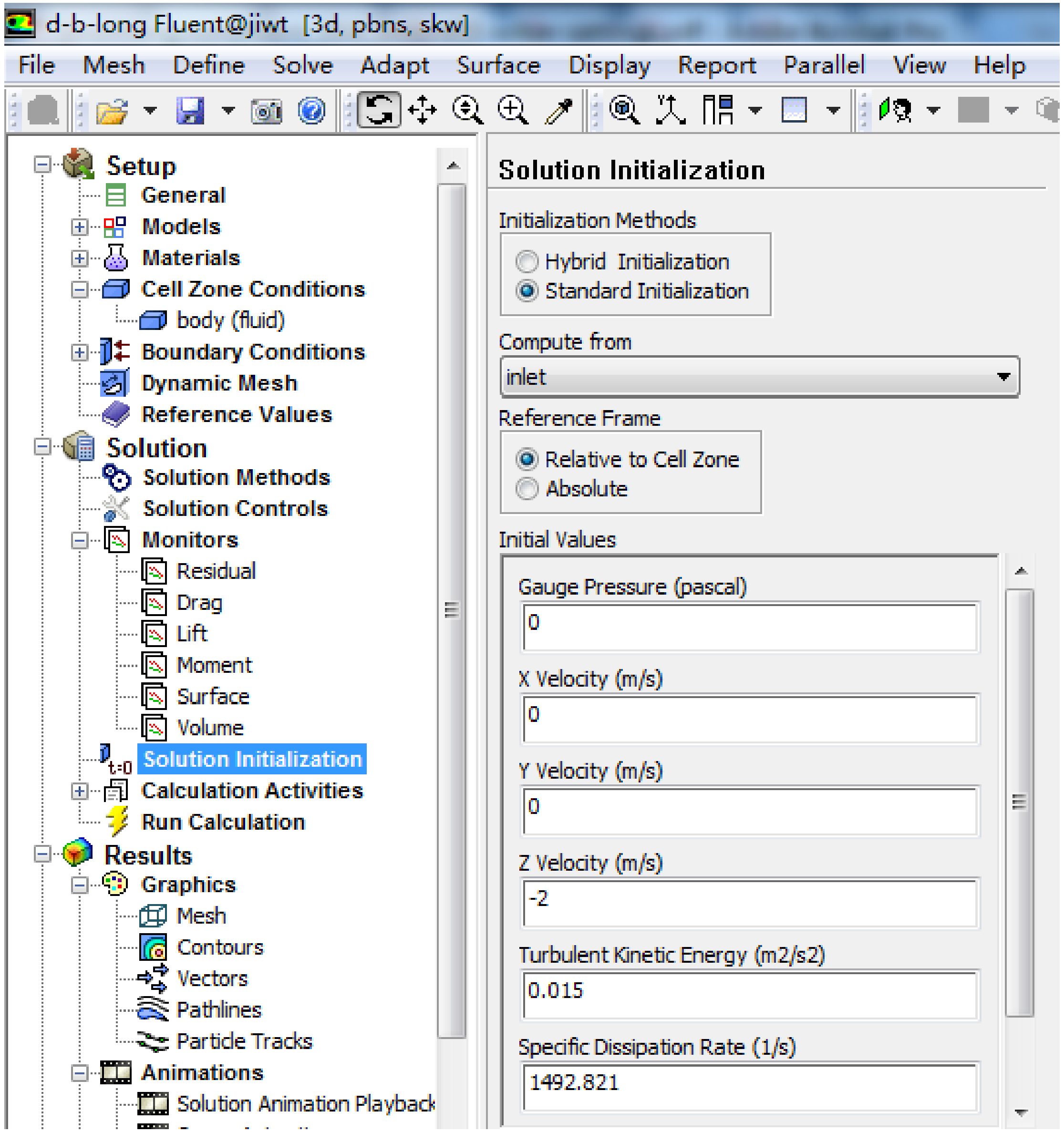
7.Solution Initialization

1)The solver works in an iterative manner.

2)Therefore before the very first iteration, a value must exist for every quantity in every grid cell.

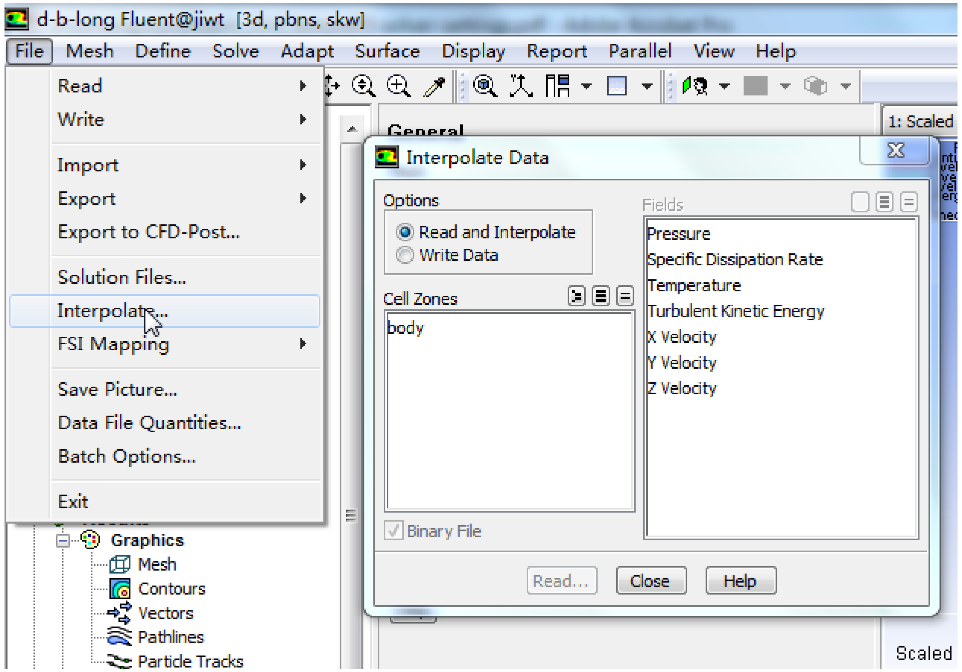
3)Setting this value is called 'Initialization'

4) The more realistic the value, the better (quicker)convergence will be.

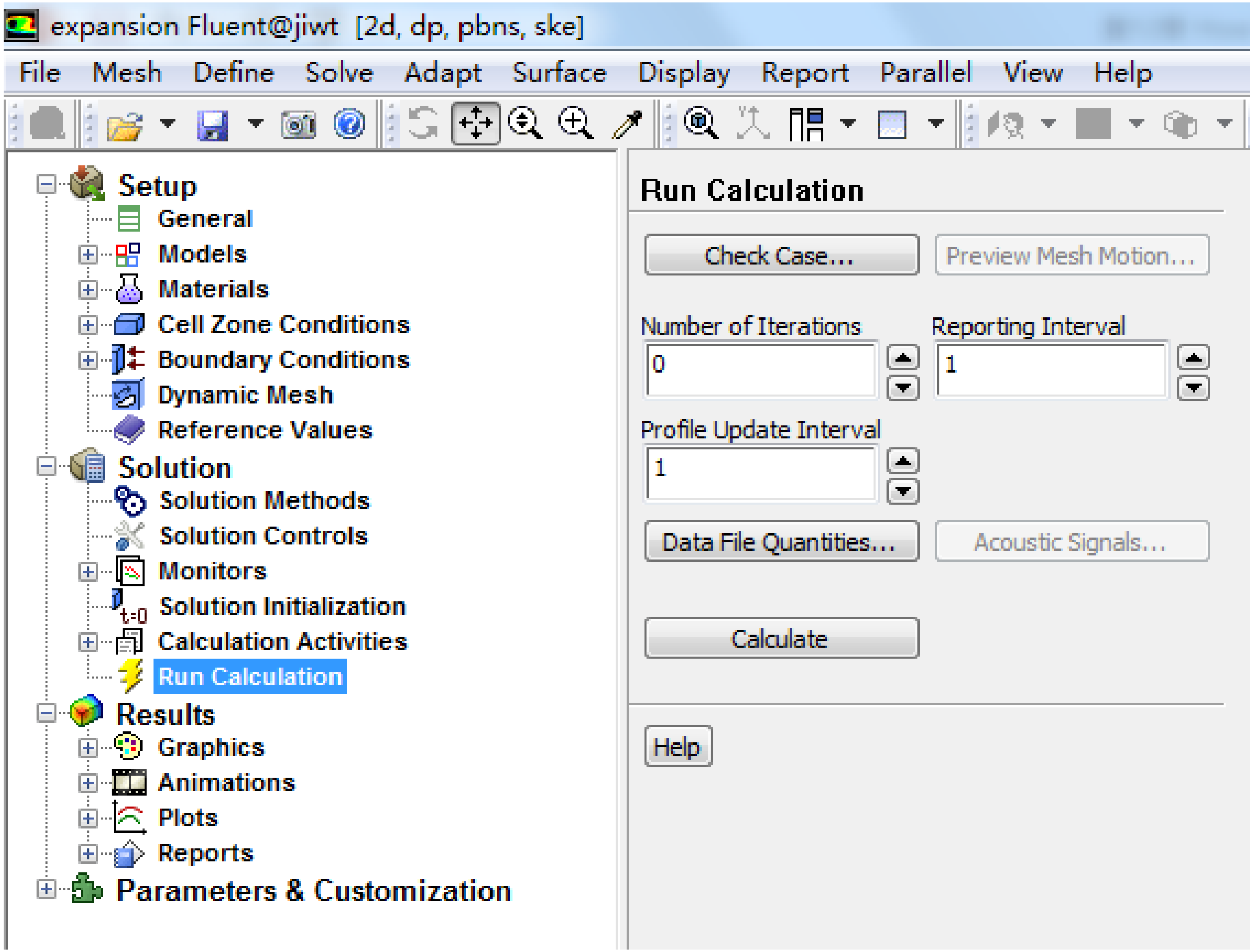


◆ Starting from a Previous Solution

- Convergence rates are dependent on how good the starting point is.
- Therefore if you already have a similar result from another simulation, you can save time by interpolating that result into the new simulation.
- Then use the 'Read and Interpolate' option on the new model.

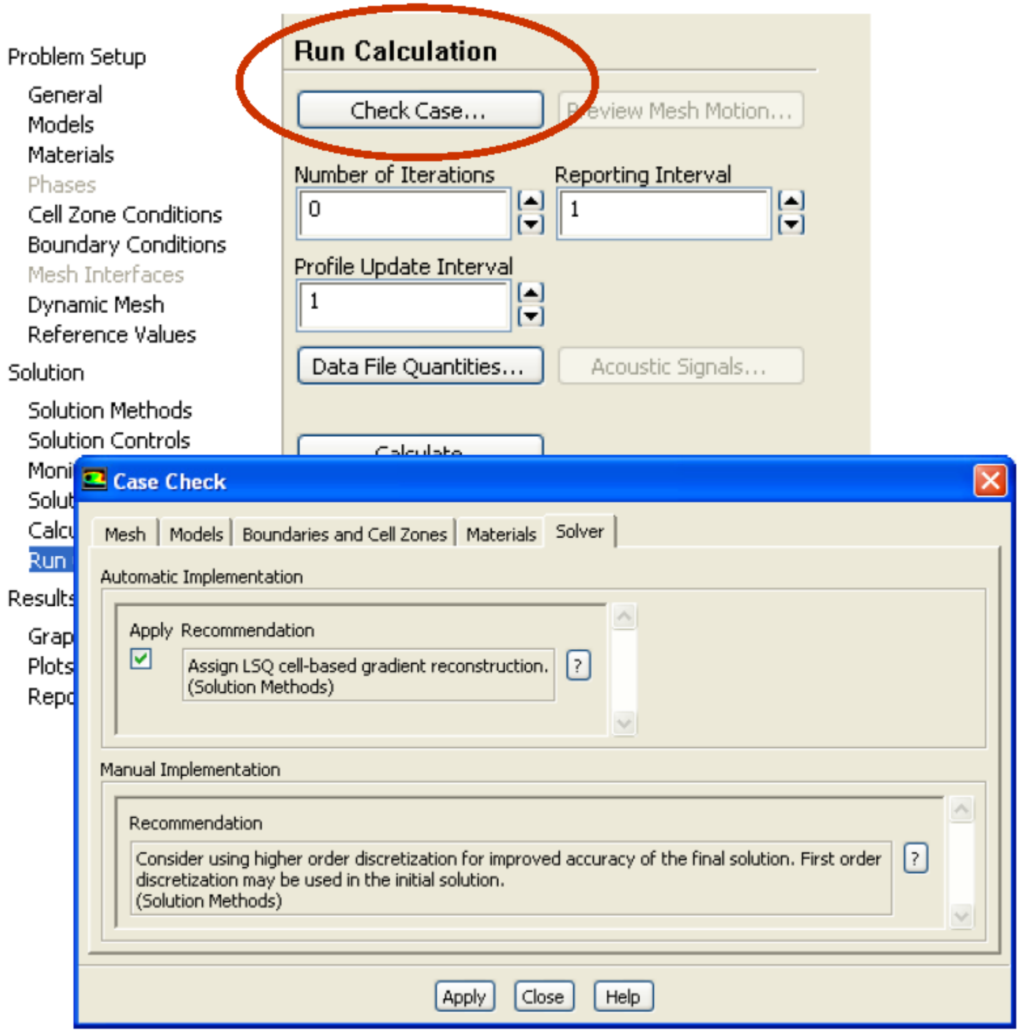


8.Run Calculation



◆ Case Check

• Case Check is a utility in FLUENT which searches for common setup errors and inconsistencies.
– Provides guidance in selecting case parameters and models.



• Contain recommendations which the user can optionally apply or ignore.

◆ Convergence

- The solver should be given sufficient iterations so that the problem is **converged.**

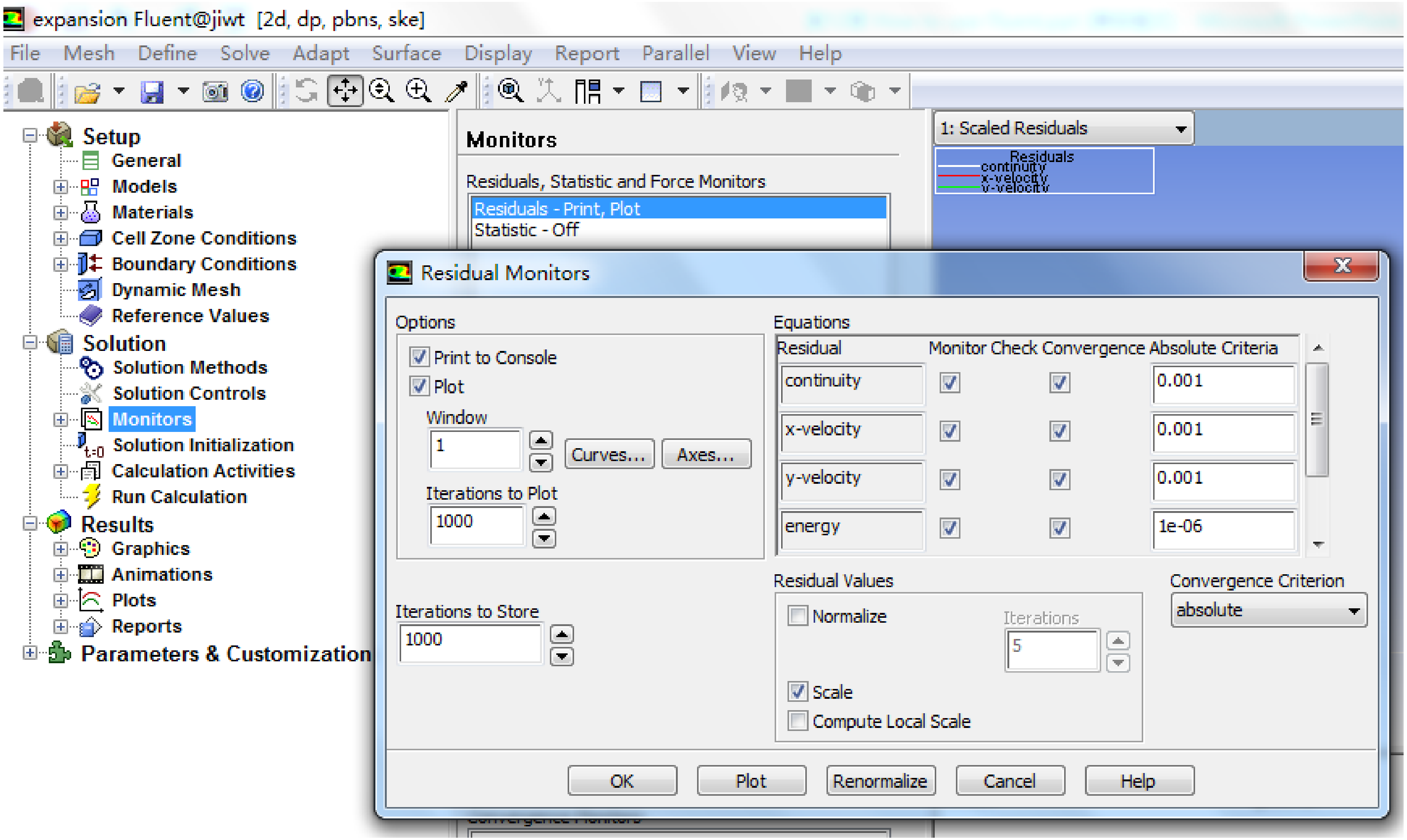
- At convergence, the following should be satisfied:

  – The solution no longer changes with subsequent iterations.

  – Overall mass, momentum, energy, and scalar balances are achieved.

  – All equations (momentum, energy, etc.) are obeyed in all cells **to a specified tolerance**

9. Monitoring



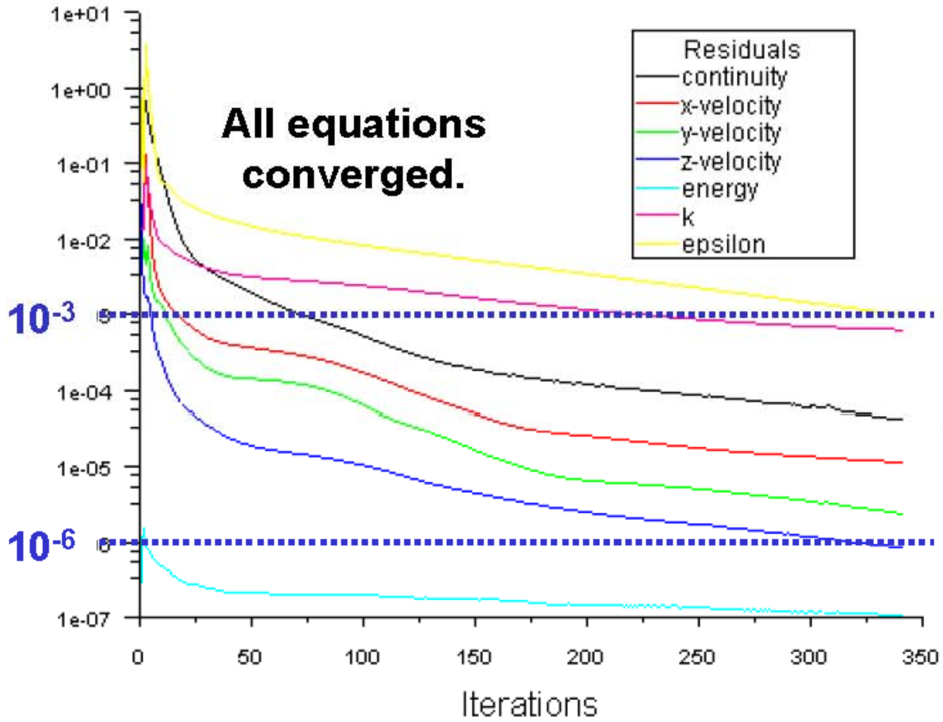
- **Monitoring convergence using residual history:**

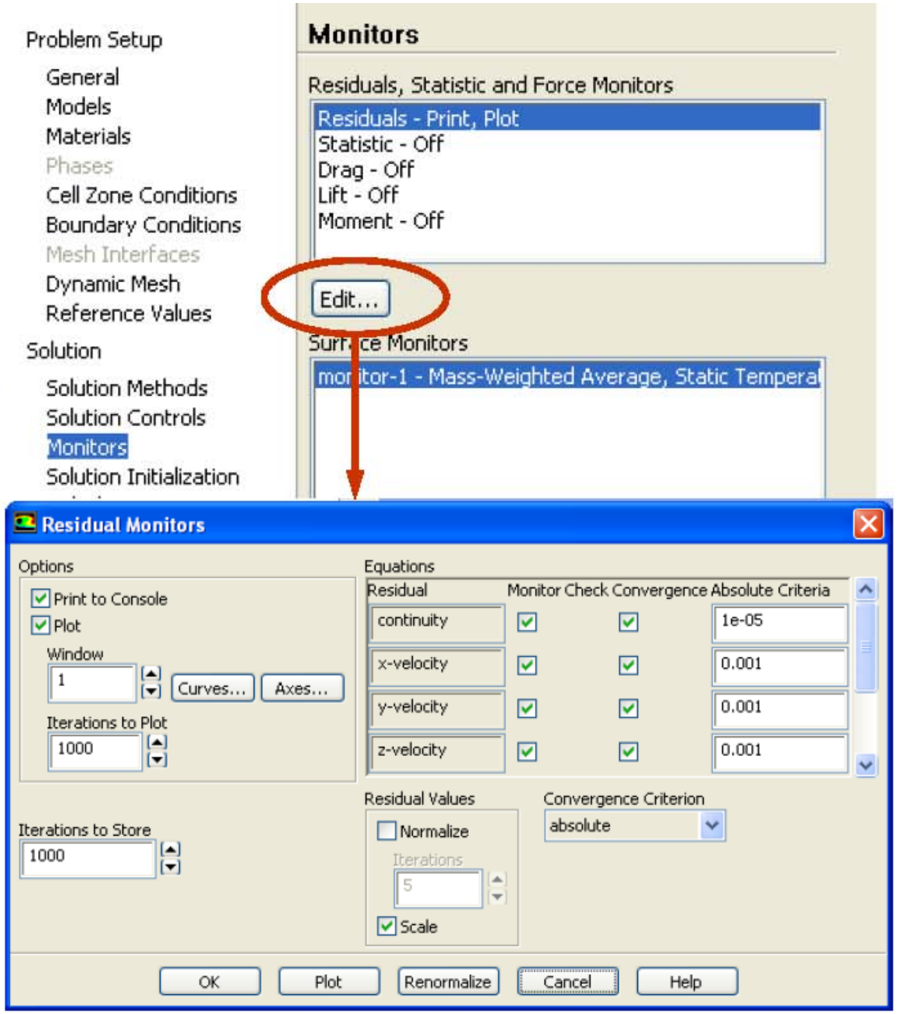
– Generally, a decrease in residuals by three orders of magnitude indicates at least qualitative convergence. At this point, the major flow features should be established.

– Scaled energy residual should decrease to 10-6 (for the Pressure-based solver).

– Scaled species residual may need to decrease to 10-5 to achieve species balance.

◆ Convergence Monitors – Residuals

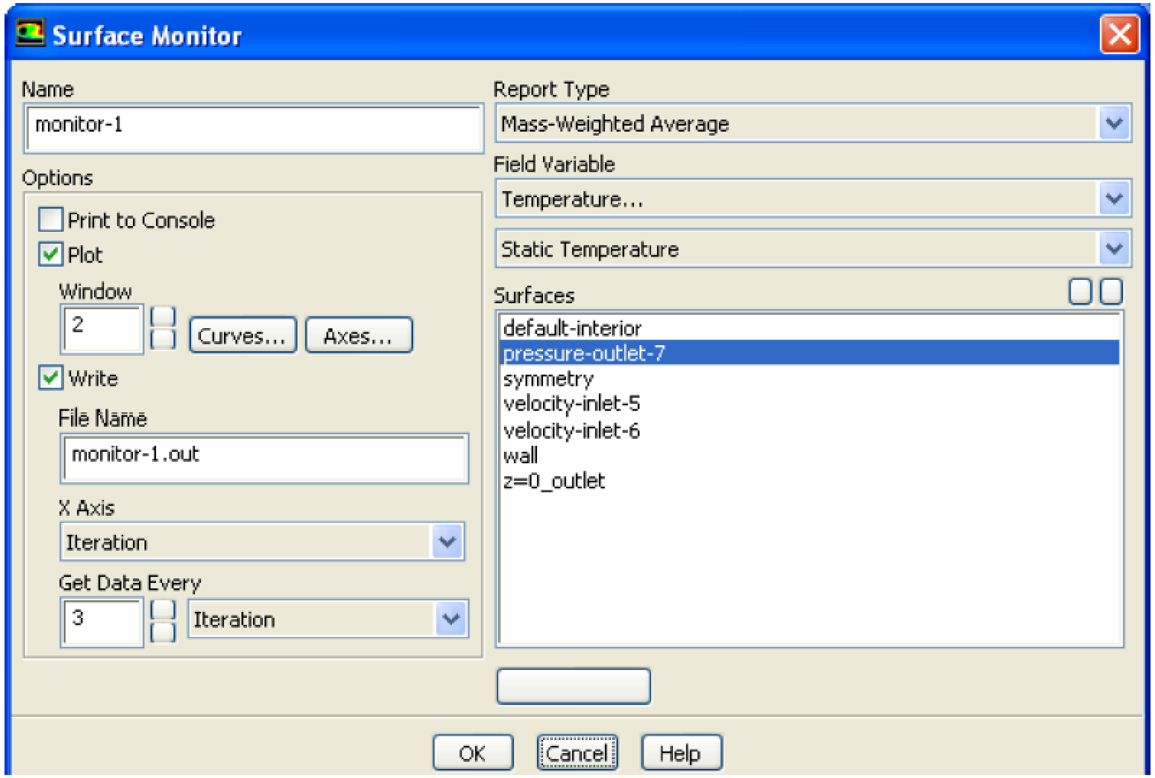




• Residual plots show when the residual values have reached the specified tolerance.

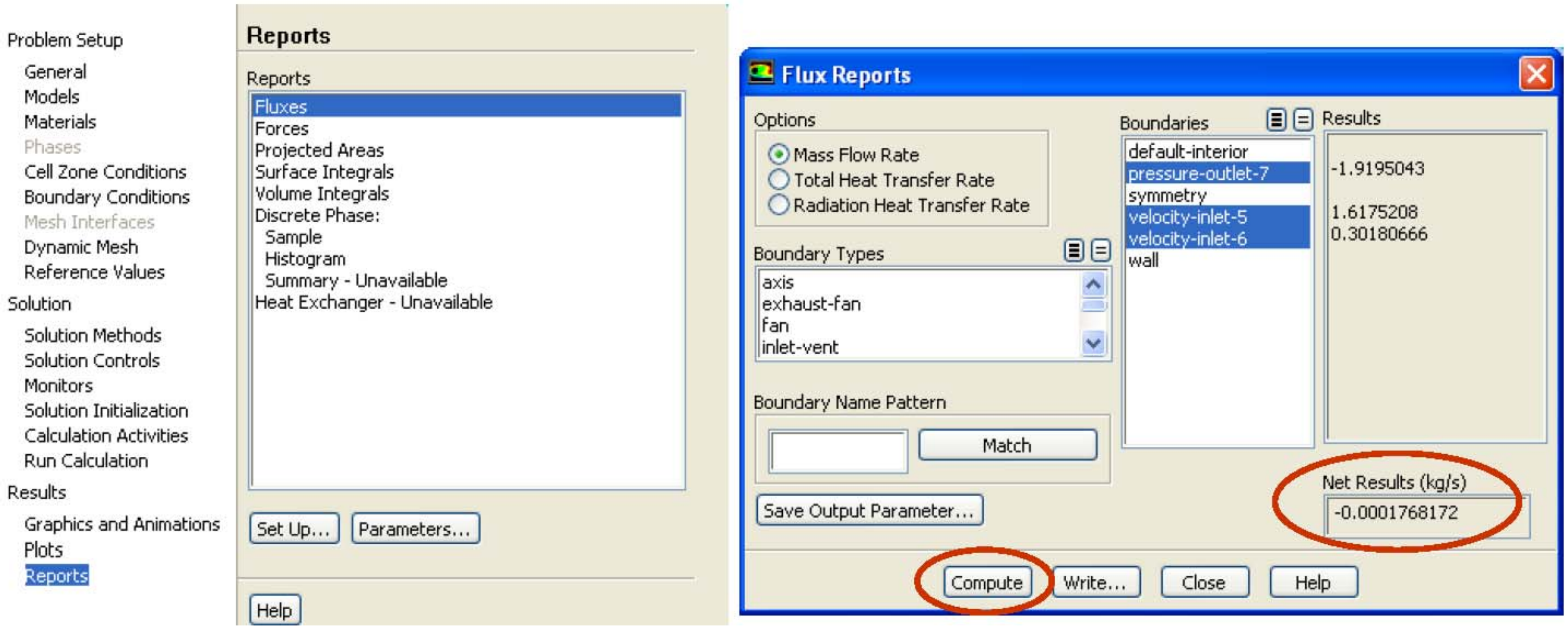
◆ Convergence Monitors – Forces and Surfaces

If there is a particular value you are interested in (lift coefficient, average surface temperature etc), it is useful to plot how that value is converging.



Checking Overall Flux Conservation

- Another important metric to assess whether the model is converged is to check the overall heat and mass balance.
- The net flux imbalance (shown in the GUI as Net Results) should be less than 1% of the smallest flux through the domain boundary



◆ Tightening the Convergence Tolerance

• If solution monitors indicate that the solution is converged, but the solution is still changing or has a large mass/heat imbalance, this clearly indicates the solution is not yet converged.

• In this case, you need to:

– Reduce values of Convergence Criterion or disable Check Convergence in the Residual Monitors panel.

– Continue iterations until the solution converges.

◆ Convergence Difficulties

- Sometimes running for further iterations is not the answer:
  – Either the solution is diverging
  – Or the residuals are 'stuck（卡住）' with a large imbalance still remaining.

- Troubleshooting

  Continuity equation convergence trouble affects convergence of all equations.

  – Compute an initial solution using a first-order discretization scheme.

  – Alter the under-relaxation or Courant numbers

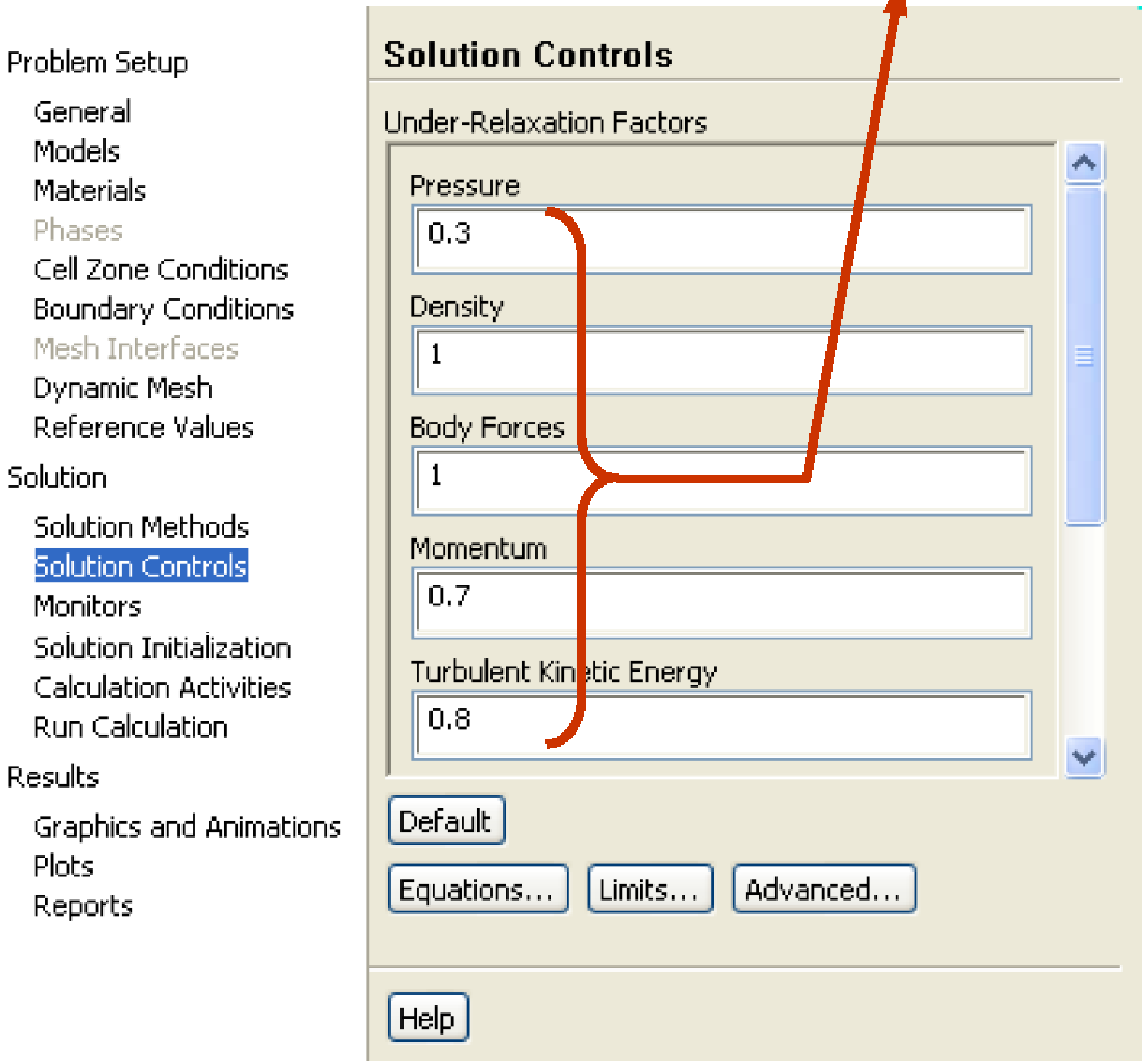
  – Check the mesh quality. It can only take one very skewed grid cell to prevent the entire solution converging

◆ Modifying Under-Relaxation Factors

- Under-relaxation factor, α, is included to stabilize the iterative process for the pressure-based solver.
- Use default under-relaxation factors to start a calculation.

- If value is too high, the model will be unstable, and may fail to converge
- If value is much too low, it will take longer (more iterations) to converge.
  - Default settings are suitable for a wide range of problems, you can reduce the values when necessary.
  - Appropriate settings are best learned from experience!

$$\phi_P = \phi_{P,\mathrm{old}} + \alpha \Delta \phi_P$$



Solution Accuracy

• Remember, a converged solution is not necessarily a correct one!

– Always inspect and evaluate the solution by using available data, physical principles and so on.
– Use the second-order upwind discretization scheme for final results.
– Ensure that solution is grid-independent

• If flow features do not seem reasonable:

– Reconsider physical models and boundary conditions
– Examine mesh quality and possibly re-mesh the problem
– Reconsider the choice of the boundaries' location (or the domain): inadequate choice of domain (especially the outlet boundary) can significantly impact solution accuracy.

◆ Grid-Independent Solutions

• It is important to verify that the mesh used was fit-for-purpose.
– Even if the grid metrics like skewness are showing the mesh is of a good quality, there may still be too few grid cells to properly resolve the flow.

• To trust a result, it must be grid-independent. In other words, if the mesh is refined further, the solution does not change.

• Typically you should perform this test once for most your of problems.

10. Results and Analysis: Graphics, Animation and Reports

Reporting Heat Flux

- Heat flux report:

– It is recommended that you perform a heat balance check so to ensure that your solution is truly converged.

- Exporting Heat Flux Data:

– It is possible to export heat flux data on wall zones (including radiation).

12.5 Meshing with ICEM for structural grid

- **Mesh**

Volume comprised of elements used to discretize a domain for numerical solution

- Heat Transfer
- Fluid dynamics
- Other

- 2D – Surface/Shell
  - –Quads(四边形)
  - –Tris（三角形）

3D-Volume

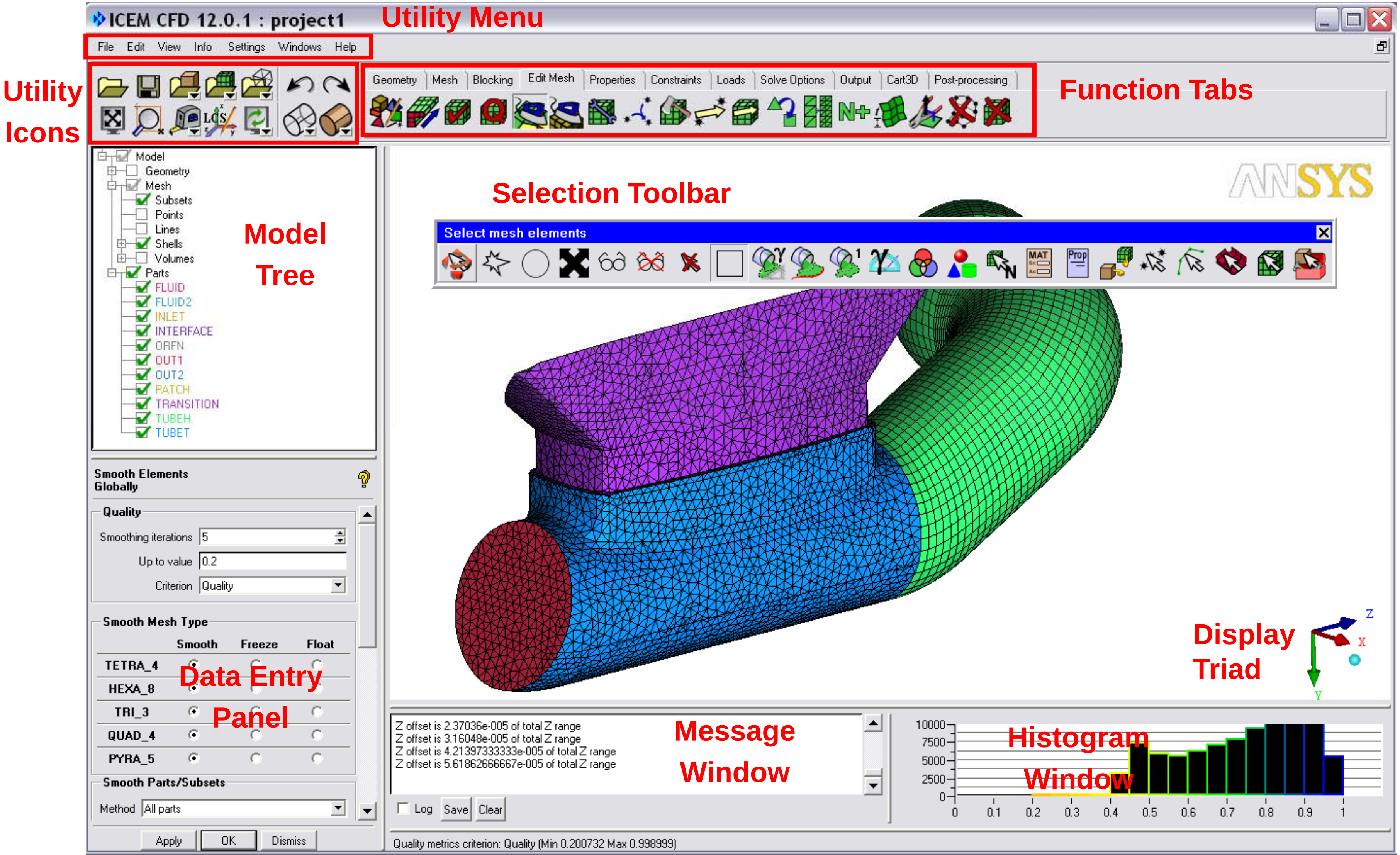
- –Tetra（四面体）
- –Pyramid（棱锥）
- –Prism（棱柱）
- –Hexa（六面体）

- •Formats
  - –Unstructured（非结构化网格）
  - –Block Structured（结构化网格）

–Nodes

- •Point locations of element corners

GUI and Layout

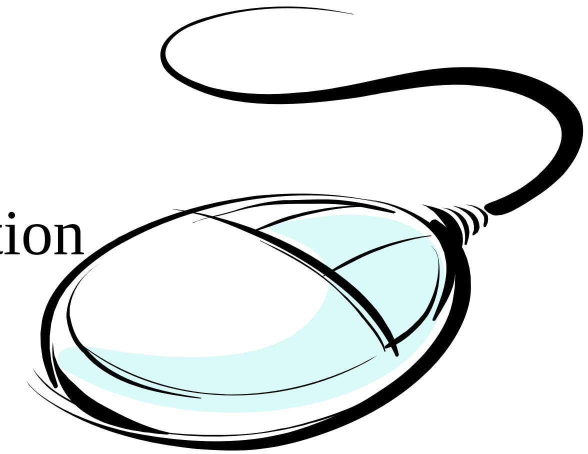


File and Directory Structure

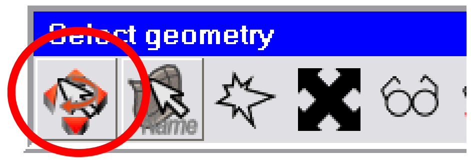
- **Primary file types:**
  - **Tetin (.tin):** Geometry
    - Geometry and material points
    - Part association
    - Global and entity mesh sizes
    - Created in Ansys ICEM CFD or Direct Cad Interface
  - **Domain file (.uns)**
    - Unstructured mesh
  - **Blocking file (.blk)**
    - Blocking topology
  - **Attribute file (.fbc, .atr)**
    - Boundary conditions, local parameters & element types
  - **Parameter file (.par)**
    - solver parameters & element types
  - **Journal and replay file(.jrf, .rpl)**
    - Record of performed operations (echo file)

Mouse Usage

- **'Dynamic' viewing mode (click and drag)**
  - left: rotate (about a point)
  - middle: translate
  - right: zoom(up-down)
    screen Z-axis rotation



  - Wheel zoom
- **Selection mode (click)**
  - left: select (click and drag for box select)
  - middle: apply operation
  - right: unselect



- **F9 toggles the mouse control to Dynamic mode while in Select mode**

- **Spaceball allows for dynamic motion even while in select mode**

Utility Menus

ICEM CFD 16.0 :

File Edit View Info Settings Help

File Menu (file i/o)

- New Project...
- Open Project...
- Save Project...
- Save Project As...
- Close Project...
- Change Working Dir...
- Geometry
- Mesh
- Blocking
- Attributes
- Parameters
- Cartesian
- Results
- Import Geometry
- Import Mesh
- Export Geometry
- Export Mesh
- Workbench Readers
- Replay Scripts
- Exit

Edit Menu

- Undo
- Redo
- Clear Undo
- Shell
- Facets -> Mesh
- Mesh -> Facets
- Struct mesh->CAD Surfaces
- Struct mesh->Unstruct Mesh
- Struct mesh->Super Domain
- Domain file->Cart3D Tri file
- Cart3D Tri file->Domain file
- Cart3D Check point file->Domain file
- Remove header lines in tetin file

View Menu

- Fit
- Box Zoom
- Top
- Bottom
- Left
- Right
- Front
- Back
- Isometric
- View Control
- Save Picture
- Mirrors and Replicates
- Annotation ...
- Add Marker
- Clear Markers
- Mesh Cut Plane

Info Menu

- Geometry Info
- Surface Area
- Frontal Area
- Curve Length
- Curve Direction
- Mesh Info
- Element Info
- Node Info
- Element Type / Property Info
- Toolbox
- Project File
- Domain File
- Mesh Report

Settings Menu (preferences)

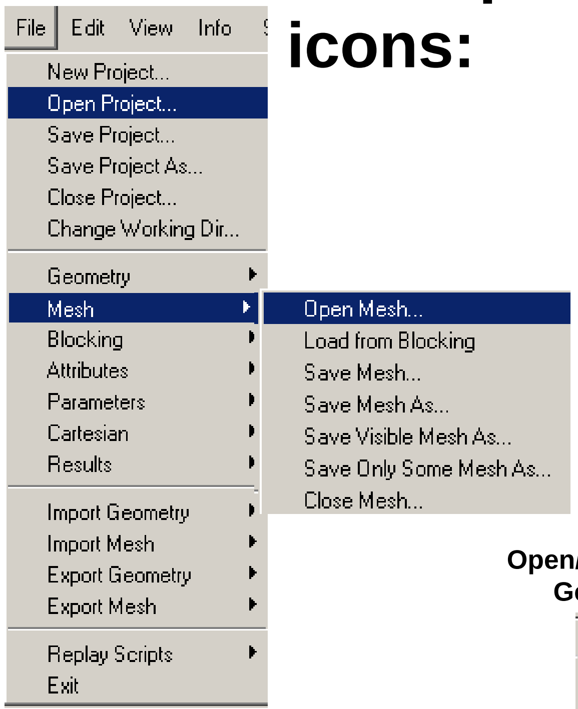
- General
- Product
- Display
- Speed
- Memory
- Lighting
- Background Style
- Mouse Bindings
- Selection
- Remote
- Model
- Geometry Options
- Meshing
- Solver
- Restore
- Reset

Help Menu

- Help Topics
- Tutorial Manual
- User Manual
- Programmer's Guide
- Installation & Licensing Guide
- What's New
- Legal Notices
- Show Customer Number
- About ANSYS ICEM CFD

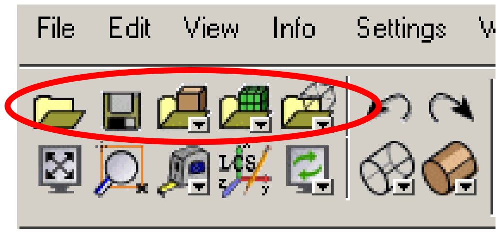
File menu

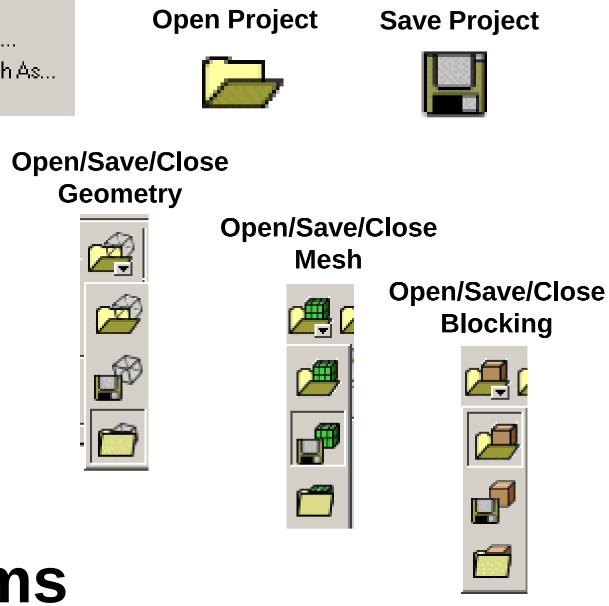
- **To open/save/close**
  - Projects
  - All file types can be opened/saved/closed independently
- **Also to**
  - Read in results data
  - Import/Export Geometry/Mesh
  - Invoke scripting
- **Exit**



***Save* frequently!**

- **Most common functions are duplicated as utility icons:**

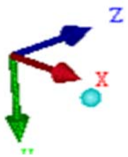




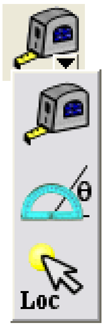
- **Icons**
  - **Many menu items duplicated by utility icons:**

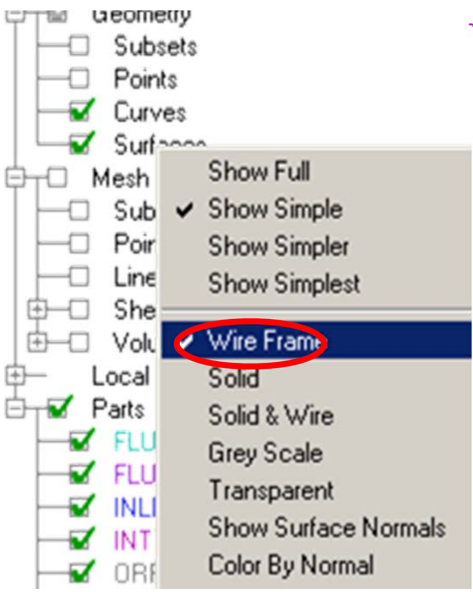
Other Commonly Used Utilities

- Edit > Undo/Redo

- View
  - Fit 
    - Fit active entities into screen
  - Box Zoom 
  - Standard views 

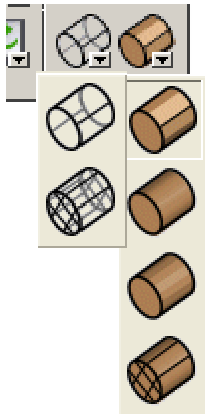
- Measure
  - Distance
  - Angle
  - Location



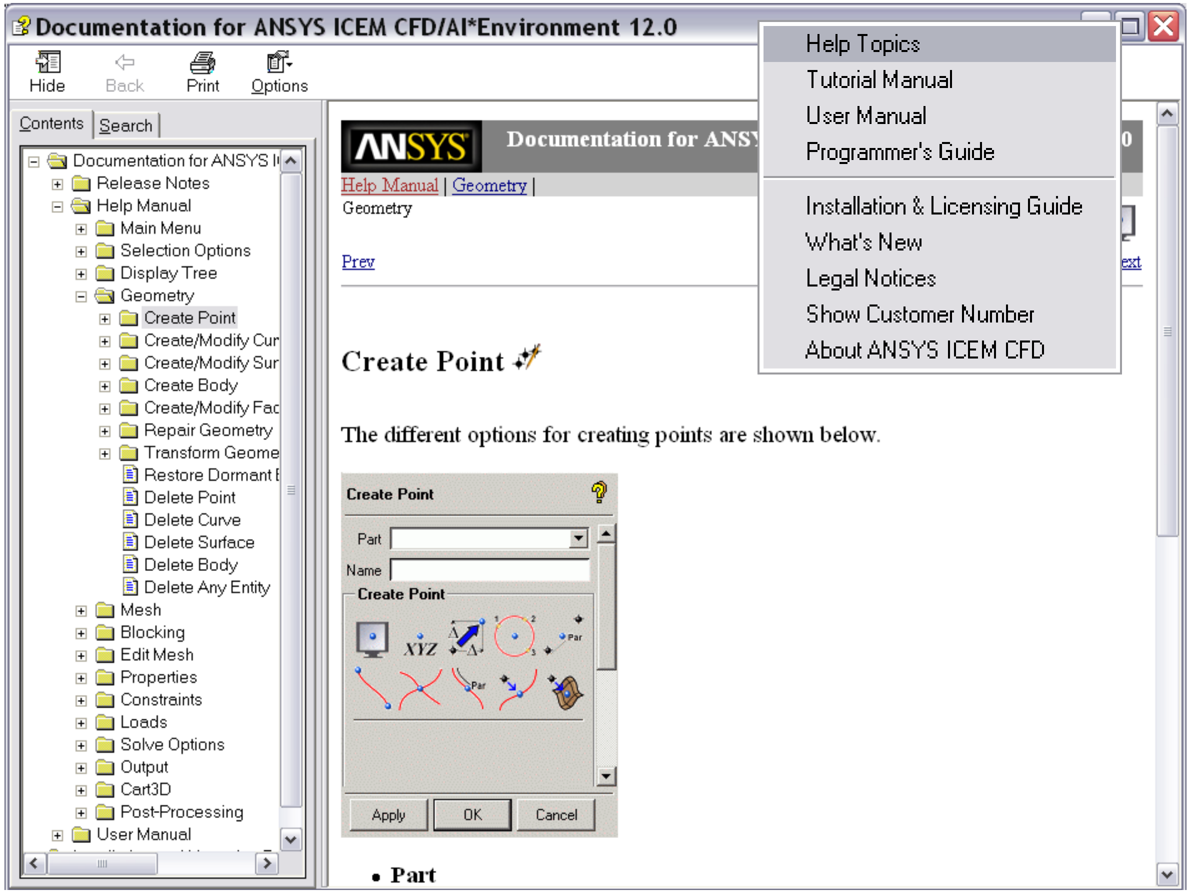


– Local Coordinate System
- •Used by:
- •Select location
- •Measuring
- •Node/point movement/creation
- •Alignment
- •Loads
- •Transformation

– Surface display
- •Wireframe
- •Solid
- •Transparent

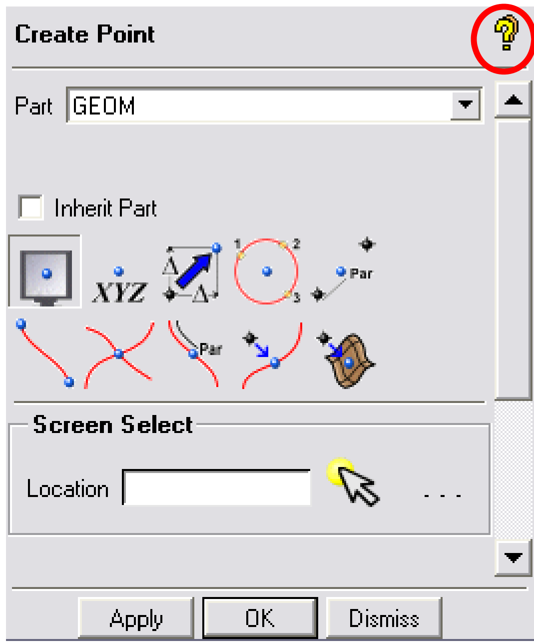


Help



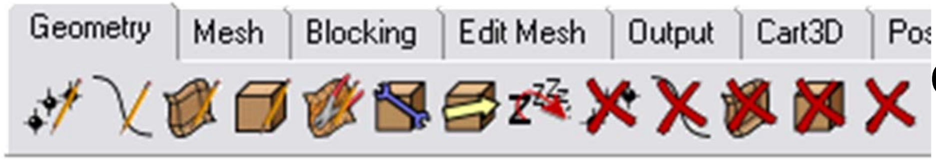
- **Menu Driven**
  - Searchable
  - Includes tutorials
  - Programmers guide (for ICEM procedures)
- **Hyper-link to specific topic**

- **Bubble explanation with cursor positioning**



Function Tabs

Geometry



Create/Modify geometry

Mesh

Set mesh sizes, types and methods
Set options
Auto create Shell, Volume, Prism meshes

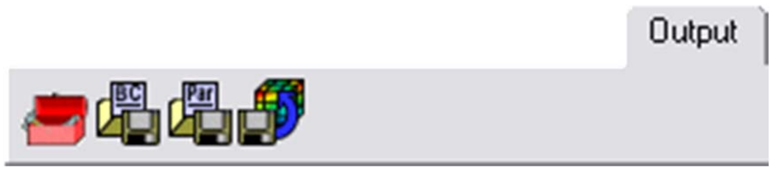
Blocking

Initialize a block
Split/modify blocks
Generate structured hexa mesh

Edit Mesh

Check, Smooth
Refine/Coarsen
Merge, Auto repair,
Manual edit
Transform, etc.

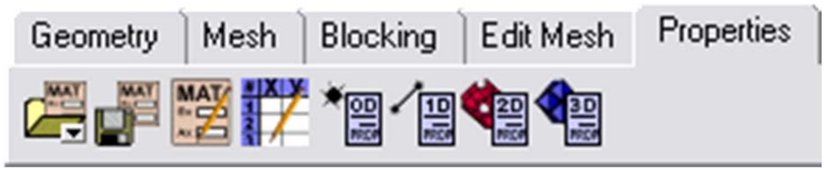
Output



Set Boundary Conditions and Parameters
Write mesh for 100+ solvers.

Primary Function Tabs

Properties



Create, read, write out material properties
Apply to geometry/elements

Constraints

Set constraints, displacements, define contacts, initial velocity, rigid walls

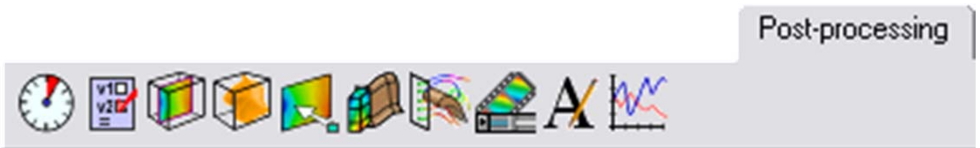
Loads

Set force, pressure and temperature loads

Solve options

Set parameters, attributes, create subcases, write out input file, run solver

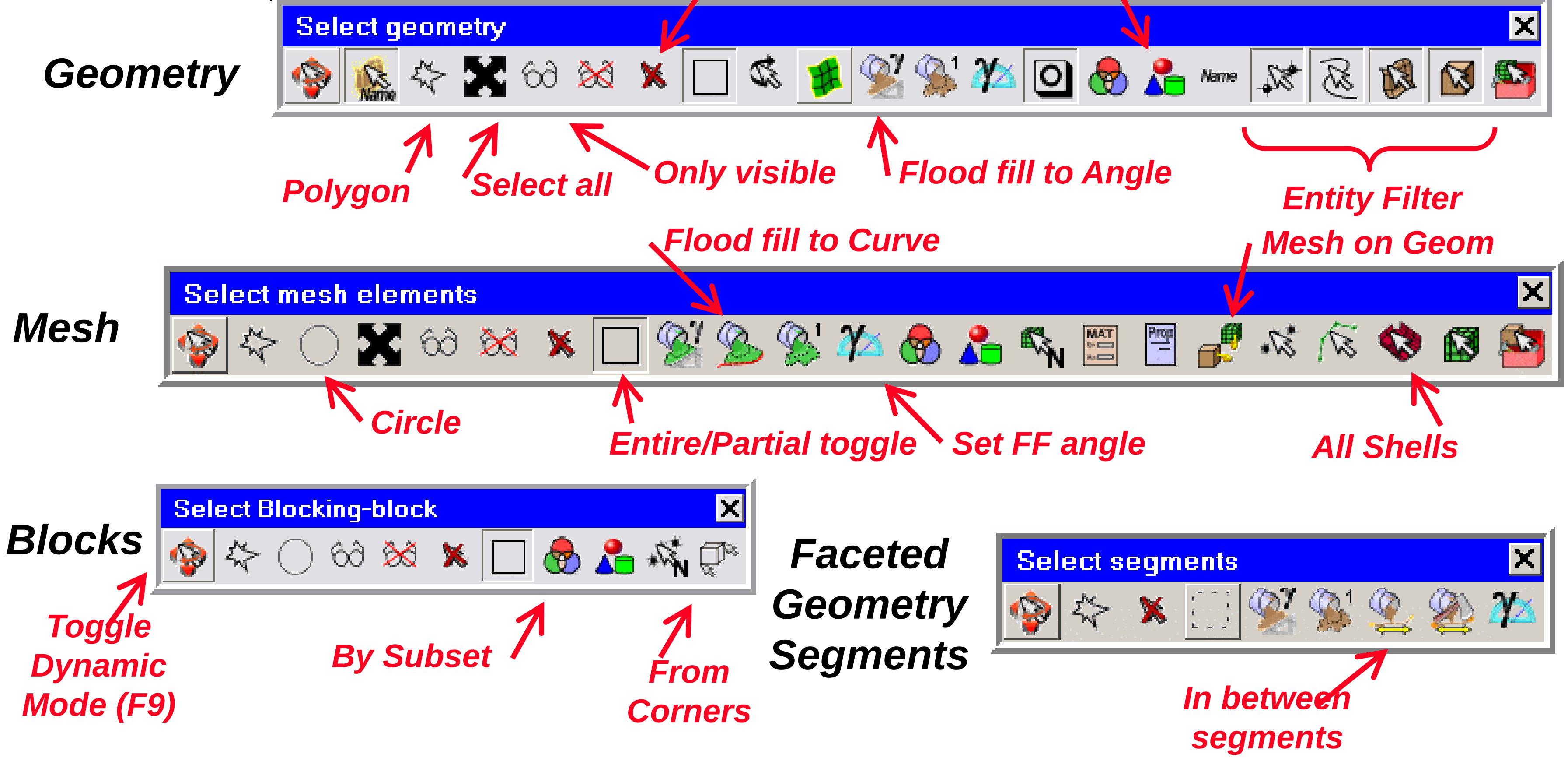
Post Processing



Visualize results: cut plane, streams, animation, calculate integral and more.

Selection Toolbar

- During select mode, selection toolbar appears
  - Some tools are common to all
  - Linked to select mode hotkeys
  - Filtering of entities



Workflow

Typical ICEM Workflow:

1.Create/open new project

2.Import/Create geometry

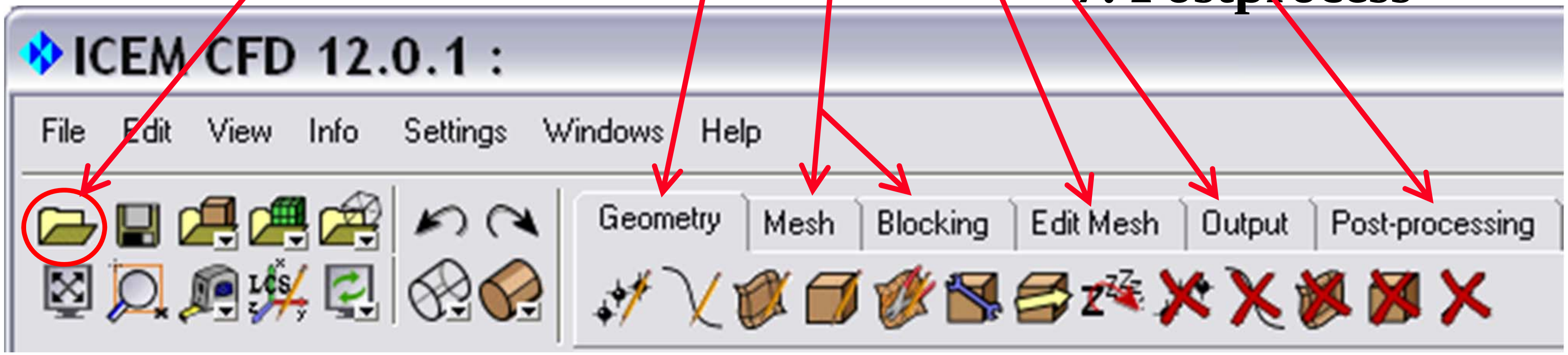
3.Build topology/Clean geometry

4. Mesh model (Possibly Hex Blocking)

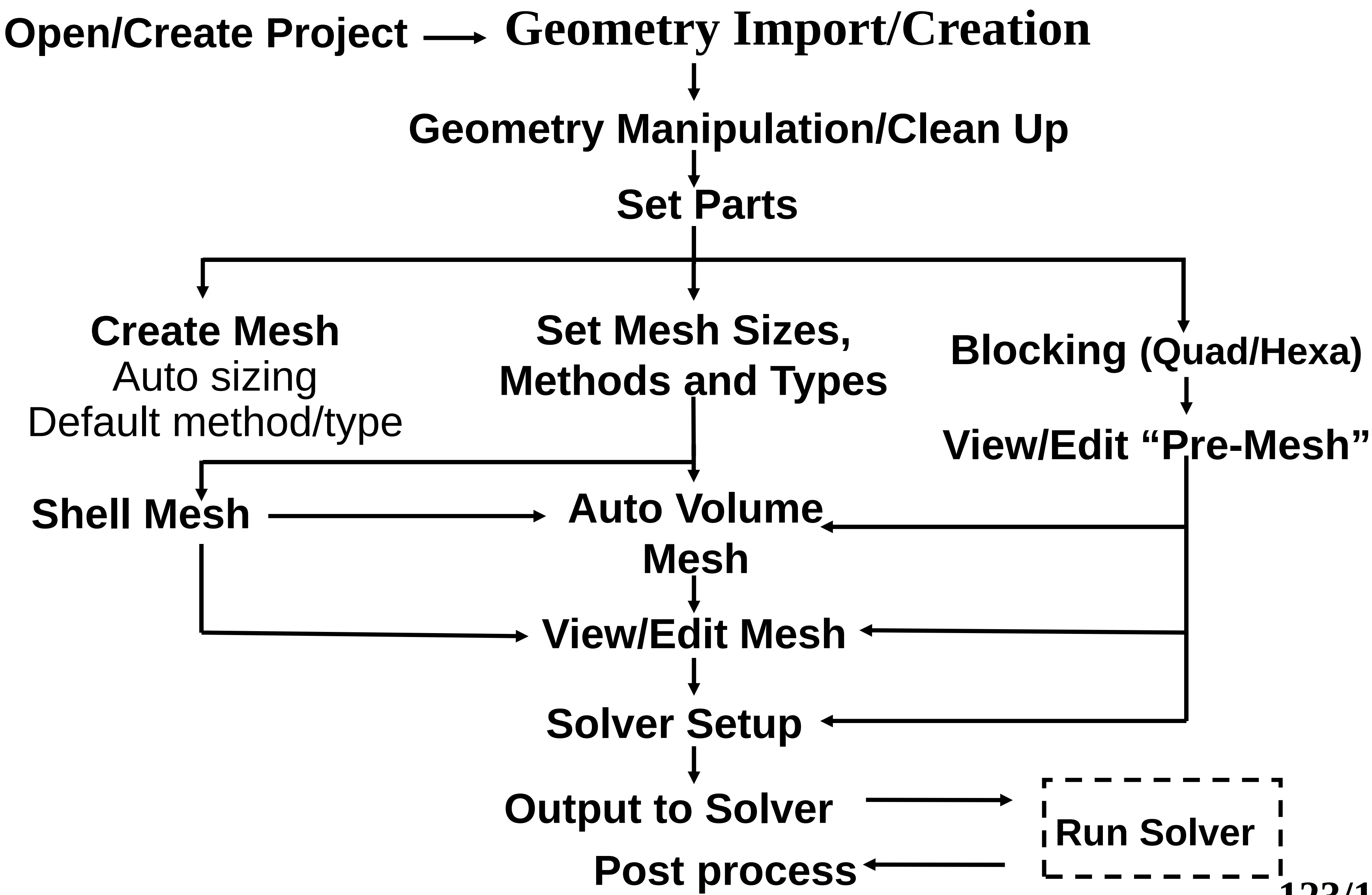
5. Check/edit mesh

6. Output to Solver

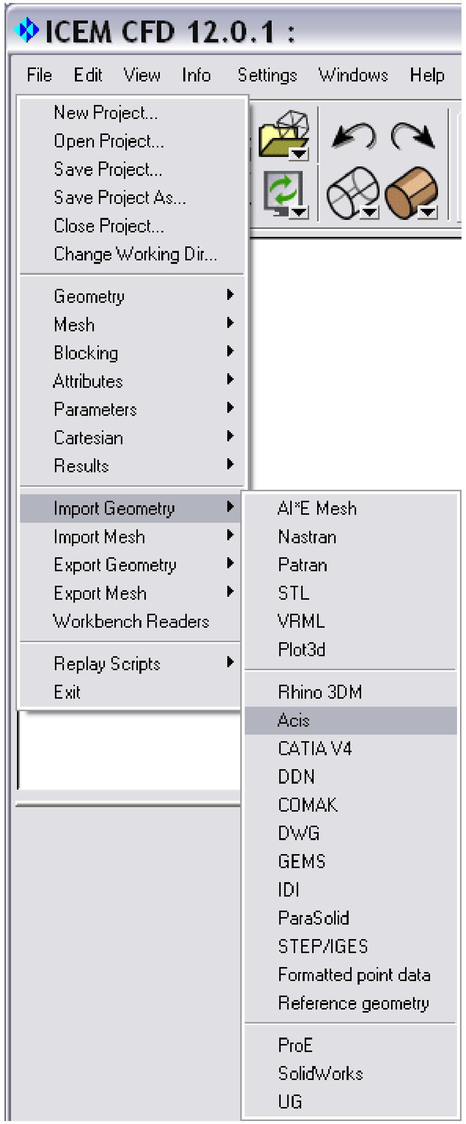
7. Postprocess



Overall Meshing Process:



Geometry Import



CAD from just about any source

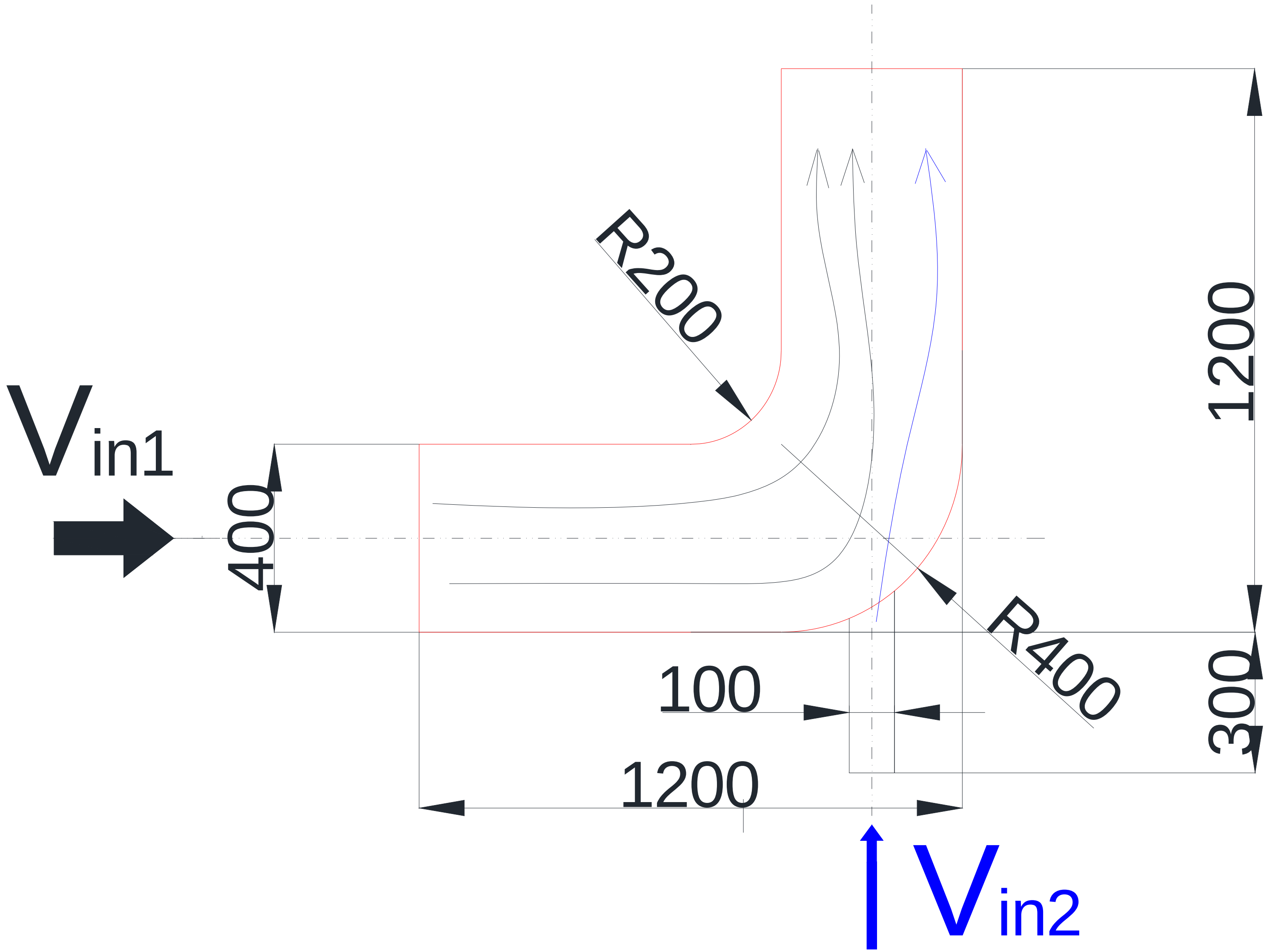
- **Direct CAD Interfaces**
  - Set up ICEM meshing requirements within CAD environment
    - Saved within CAD part
    - Retained for parametric geometry changes
  - Directly write out ICEM formatted geometry (tetin file)
    - No 3rd party exchange (clean!)
  - ProE
  - Unigraphics
  - SolidWorks
- **Direct import**
  - ACIS (.sat)
  - CATIA V4
  - DWG/DXF

–Catia V4
–IDEAS
–GEMS
–IDEAS (IDI)
–Parasolid

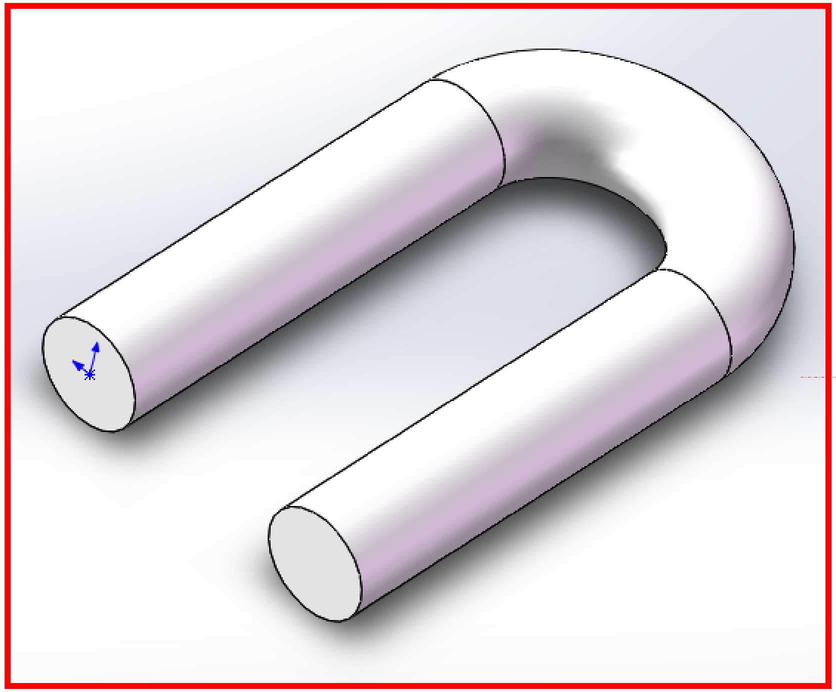
–STEP/IGES
–Pro/E
–Unigraphics

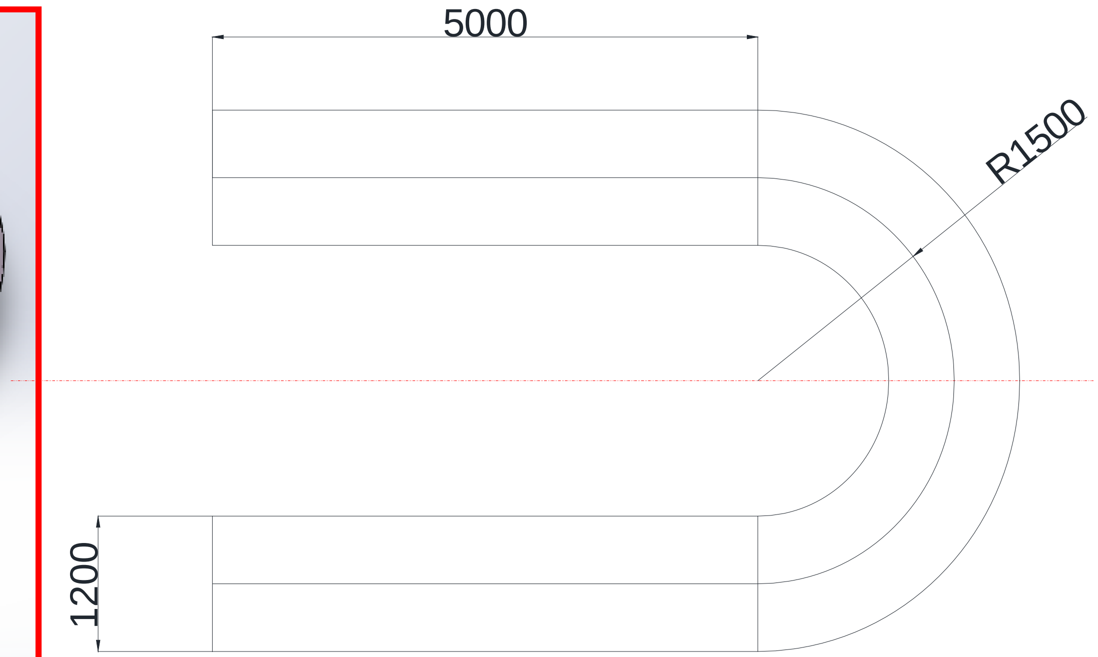
Examples to generate structural grid

Example 1: 2D Pipe Junction

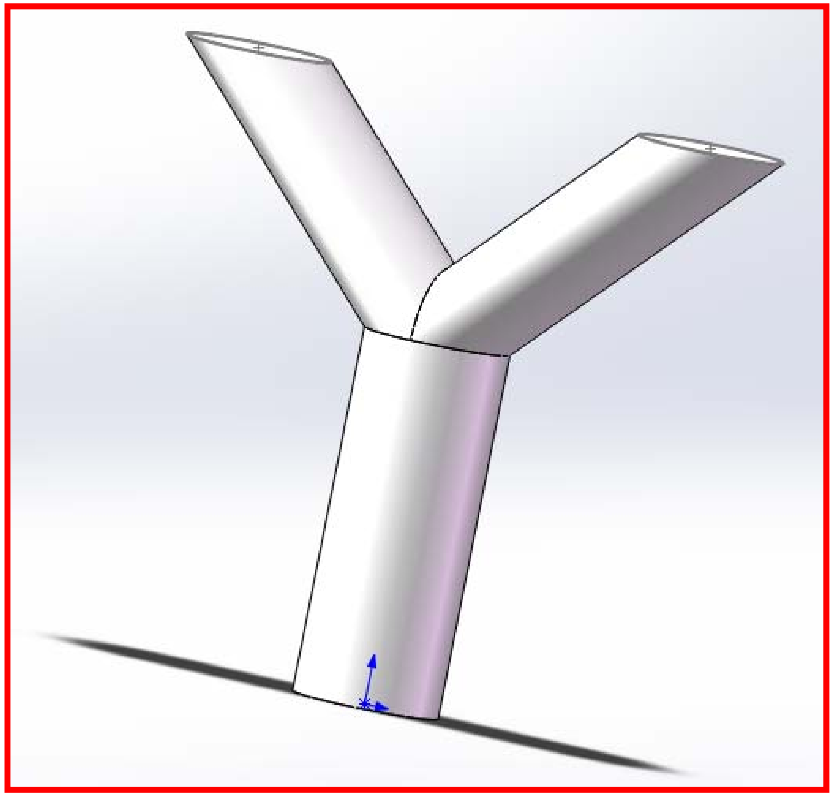


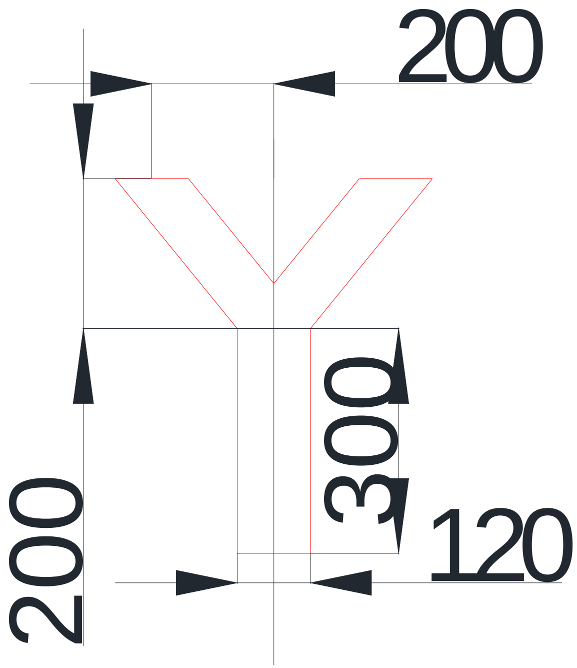
Example 2: Flow in a U turn



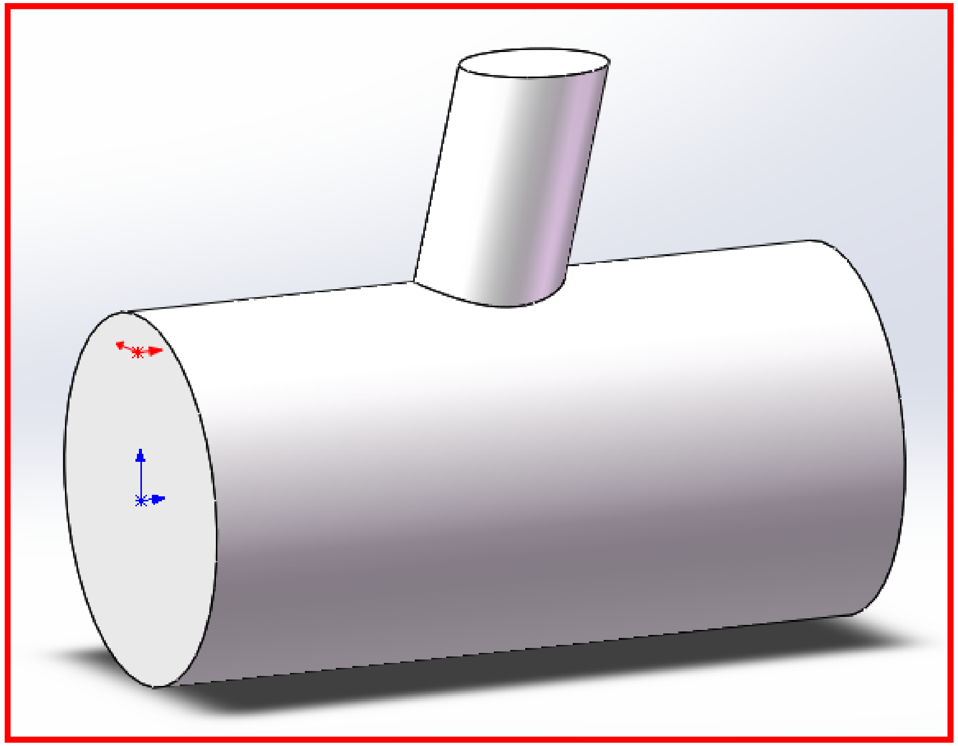


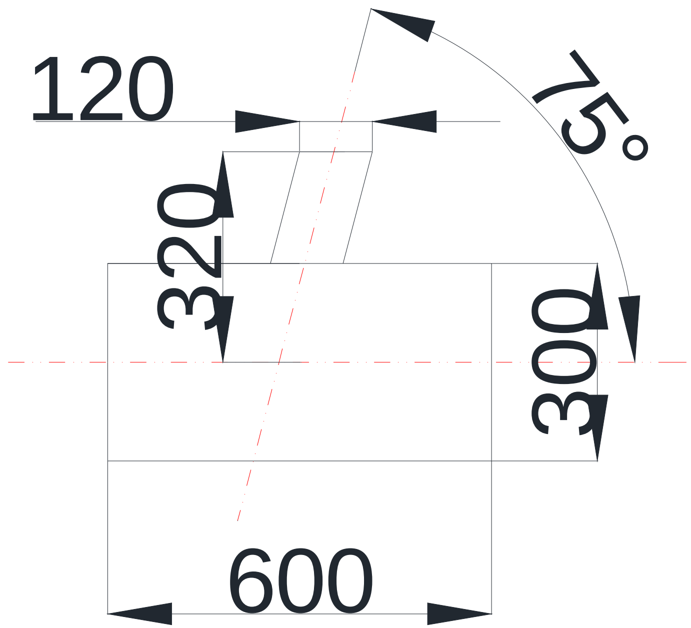
Example 3: Flow in a “Y” tube





Example 4: Three pipe junction





Thanks very much!

谢谢各位!

同舟共济

渡彼岸!