Numerical Heat Transfer
（数值传热学）
Chapter 12 How to Use ANSYS FLUENT

Instructor： Ji Wen-Tao, Tao Wen-Quan

School of Energy and Power Engineering
Xi’an Jiaotong University

Xi’an, 2024-Dec. 16

CFD-NHT-EHT CENTER

数值传热学

第12章 ANSYS FLUENT软件学习和应用

主讲：冀文涛 陶文铨

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

2024年12月16日，西安

CFD-NHT-EHT
CENTER

Chapter 12 How to Use ANSYS FLUENT

12. 1. Introduction to Numerical Heat Transfer Software

FLUENT, CFX, COMSOL, STAR-CD, ABAQUS, PHOENICS, ADINA, NASTRAN…..

Market share: Fluent>CFX> others

Accuracy: case-dependent

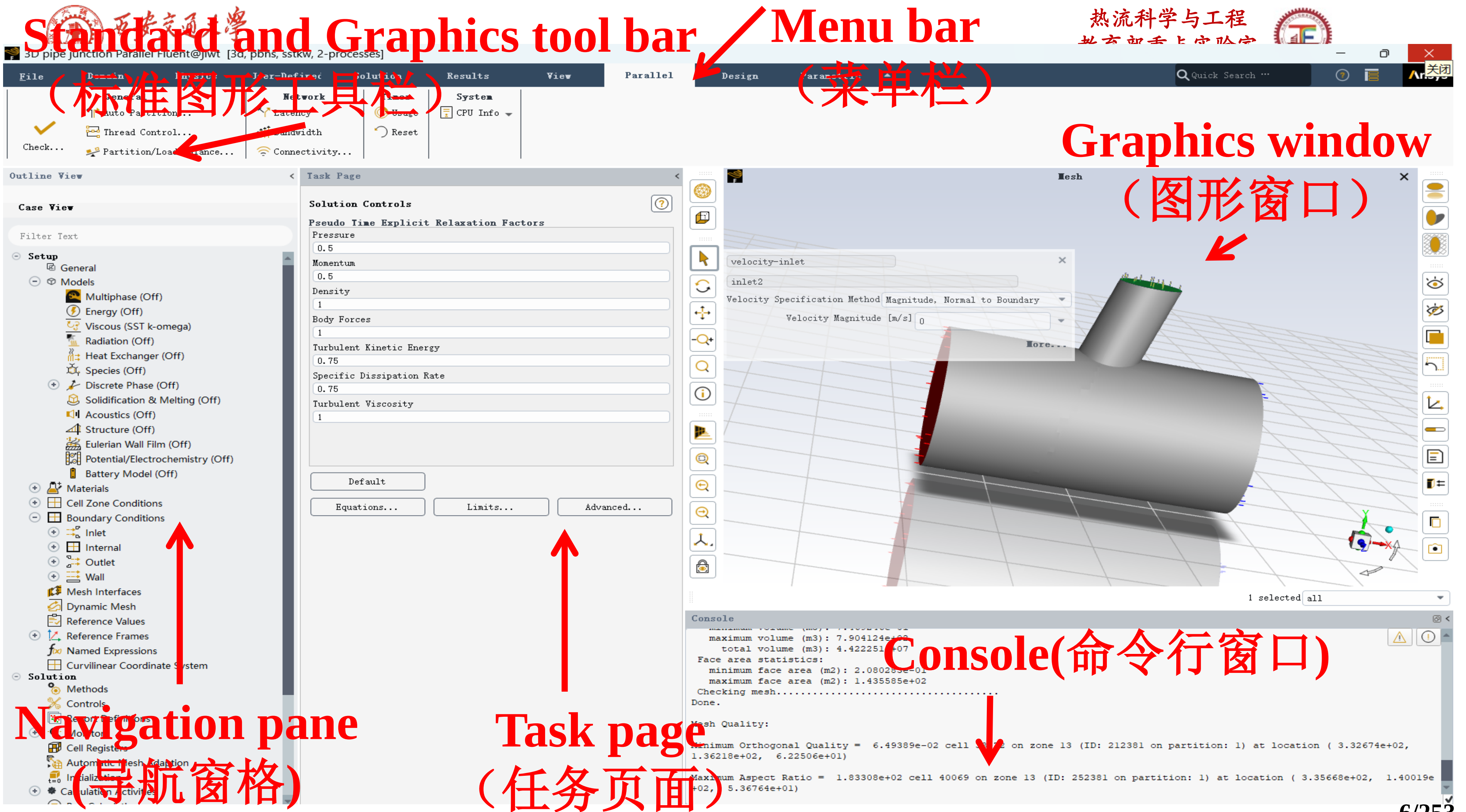
Technical documentation available:

Fluent>CFX> others

Fluent is a general-purpose computational fluid dynamics (CFD) program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries.

It offers a user-friendly interface that streamlines the CFD process from pre- to post-processing within a single-window workflow（流程化操作、操作简便）.

It provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes(重要特点) that can be generated about complex geometries.

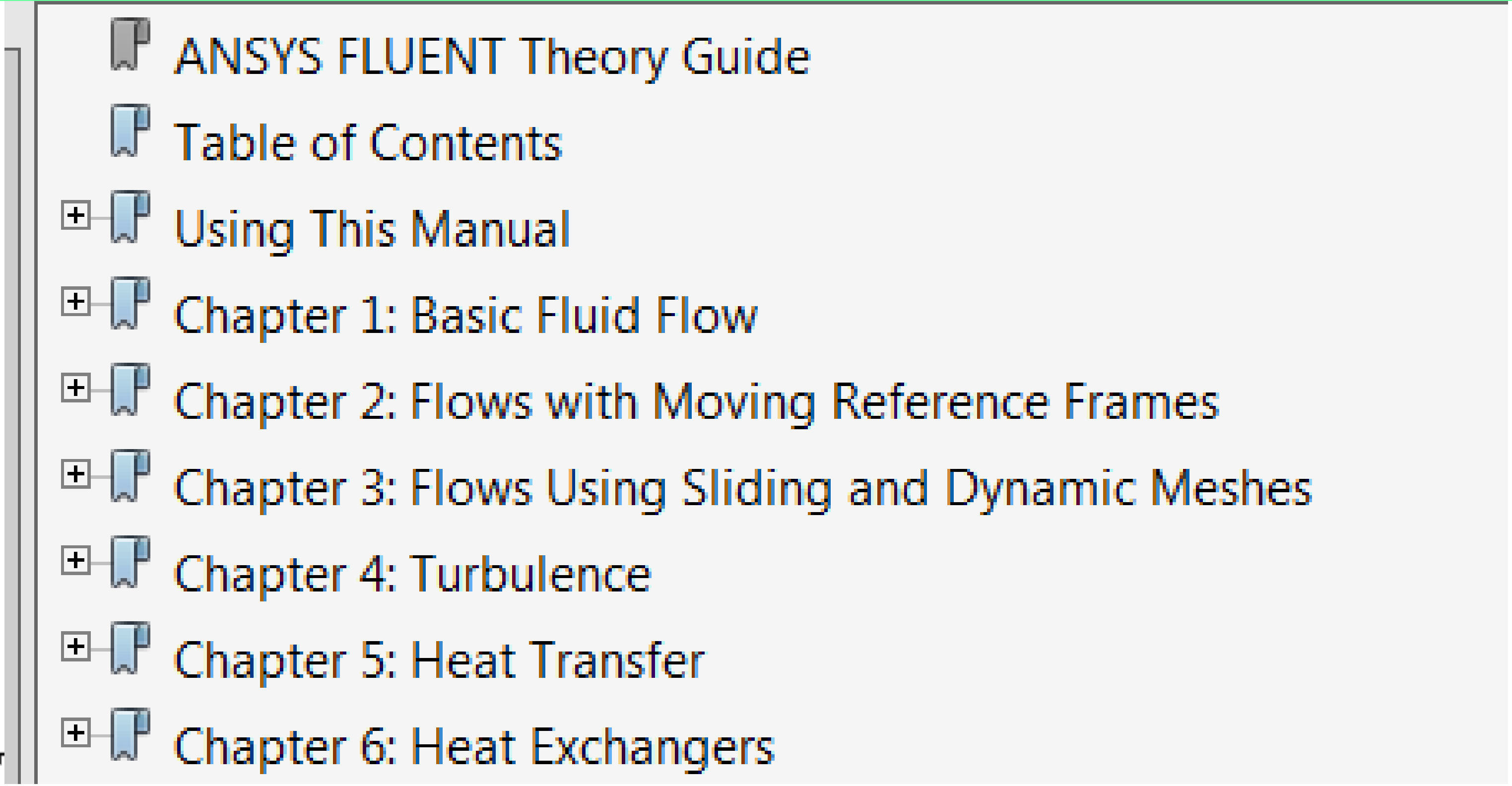

热流科学与工程
Standard and Graphics tool bar
（标准图形工具栏）
Menu bar
（菜单栏）
Graphics window
（图形窗口）
Console(命令行窗口)
Navigation pane
（导航窗格）
Task page
（任务页面）
3D pipe junction Parallel Fluent@jiwt [3d, pbns, sstkw, 2-processes]
File
Results
View
Parallel
Design
Quick Search
System
CPU Info
Check...
Thread Control...
Connectivity...
Reset
Outline View
Case View
Filter Text
Setup
General
Models
Multiphase (Off)
Energy (Off)
Viscous (SST k-omega)
Radiation (Off)
Heat Exchanger (Off)
Species (Off)
Discrete Phase (Off)
Solidification & Melting (Off)
Acoustics (Off)
Structure (Off)
Eulerian Wall Film (Off)
Potential/Electrochemistry (Off)
Battery Model (Off)
Materials
Cell Zone Conditions
Boundary Conditions
Inlet
Internal
Outlet
Wall
Mesh Interfaces
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Cell Registers
Task Page
Solution Controls
Pseudo Time Explicit Relaxation Factors
Pressure
0.5
Momentum
0.5
Density
1
Body Forces
1
Turbulent Kinetic Energy
0.75
Specific Dissipation Rate
0.75
Turbulent Viscosity
1
Default
Equations...
Limits...
Advanced...
Mesh
velocity-inlet
inlet2
Velocity Specification Method Magnitude, Normal to Boundary
Velocity Magnitude [m/s] 0
1 selected all
Console
maximum volume (m3): 7.904124e+02
total volume (m3): 4.422251e+07
Face area statistics:
maximum face area (m2): 1.435585e+02
Checking mesh...
Done.
Maximum Aspect Ratio = 1.83308e+02 cell 40069 on zone 13 (ID: 252381 on partition: 1) at location (3.35668e+02, 1.40019e+02, 5.36764e+01)

The Contents of the FLUENT Manuals

1). *Getting Started Guide* contains general information about getting started with using FLUENT.(56 Pages)

2). *User's Guide* contains detailed information about using FLUENT, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution and analyzing results.(2498 Pages)

3). *Theory Guide* contains reference information for how the physical models are implemented in FLUENT.



3). *Workbench User's Guide* contains information about getting started with and using FLUENT within the Workbench environment.(110 Pages)

4). *UDF Manual* contains information about writing and using user-defined functions (UDFs).(566 Pages)

Hooking:挂载

5).*Tutorial Guide* contains a number of example problems with detailed instructions, commentary, and post-processing of results.(1146 Pages)

6).*Text Command List* contains a brief description of each of the commands in FLUENT's text interface.(128 Pages)

7). ***Fuel Cell Modules Manual*** **contains information about the background and the usage of two separate add-on fuel cell models for FLUENT.(119 Pages)**

Advantage of commercial NHT Software:

Easy to use!

However, it can not solve all the problems! In our research, for individual calculations, particularly complicated models, there are still a few computation issues for Fluent.

Advantage of Self-programming for NHT:

It is rather important for research!

We can understand the basic procedures and mechanisms in NHT.

Almost all the widely used CFD softwares are controlled by the United States and European Union. It remind us that U.S.A still leads in industrial software and many other industries.

CFD analysis has replaced most of experiments to design industrial products. If the United States and European Union implement the same restrictive measures, we have to go back to do a lot of experiments.

After learning the basic steps of other software, we should develop our domestic CFD software.

12.1.2 ANSYS Fluent software

Fluid flow: 2D planar, 2D axisymmetric, 2D axisymmetric with swirl (rotationally symmetric), and 3D flows.

Incompressible or compressible flows, including all speed regimes (low subsonic, transonic, supersonic, and hypersonic flows)

Heat Transfer: Conduction/Convection/Radiation Heat Transfer

Multiphase Flow, Fluid-Structure Interaction, Turbulence Modeling

Combustions/Pollution Distribution

12.1.3 How Does NHT Software Work?

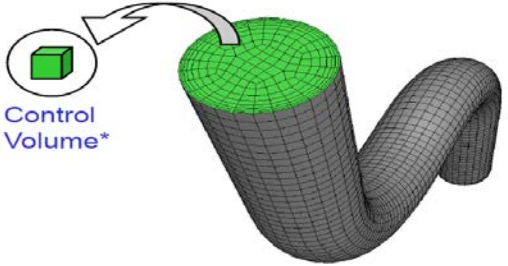
Fluent solvers are based on the finite volume method.

1) Domain is discretized into a finite set of control volumes.(Chapter 2.Discretization of Computational Domain计算区域与控制方程的离散化)

2) General conservation equations for mass, momentum, energy, etc. are solved on this set of volumes. (Chapters 5. Discretized Schemes of Diffusion and Convection Equation；第5章：对流扩散方程的离散格式)

3) Partial differential equations are discretized into a system of algebraic equations.(Chapter 5: Discretized Schemes of Diffusion and Convection Equation)

4) All algebraic equations are then solved numerically to obtain the solution field.(Chapter 7 Solution Methods for Algebraic Equations第7章：代数方程的求解方法)



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

Solver Settings

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**

Post-Processing

12.2.1 NHT Analysis: Basic Steps

◆ Problem Identification and Pre-Processing

1. Define our modeling goals.（控制方程及边界条件）
2. Identify the domain we will model.
3. Design and create the grid. (计算区域与控制方程的离散化，第1和2章)

◆ Solver Execution

4. Set up the numerical model.(算法和格式选择，第5，6章)
5. Compute and monitor the solution.(方程求解，第7章)

◆ Post-Processing

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

1. Define Our Modeling Goals

1) What results are we looking for, and how will they be used?

- What are our modeling options?
- What physical models will need to be included in our analysis?
- What simplifying assumptions do we have to make?
- What simplifying assumptions can we make?
- Could we use user-defined functions (written in C)?

2) What degree of accuracy is required?

3) How quickly do we need the results?

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

5) Where will the computational domain begin and end?

- Do we have boundary condition information?
- Can the boundary condition types accommodate that information?
- Can we extend the domain to a point where reasonable data exists?

6) Can it be simplified or approximated as a 2D or axi-symmetric problem?

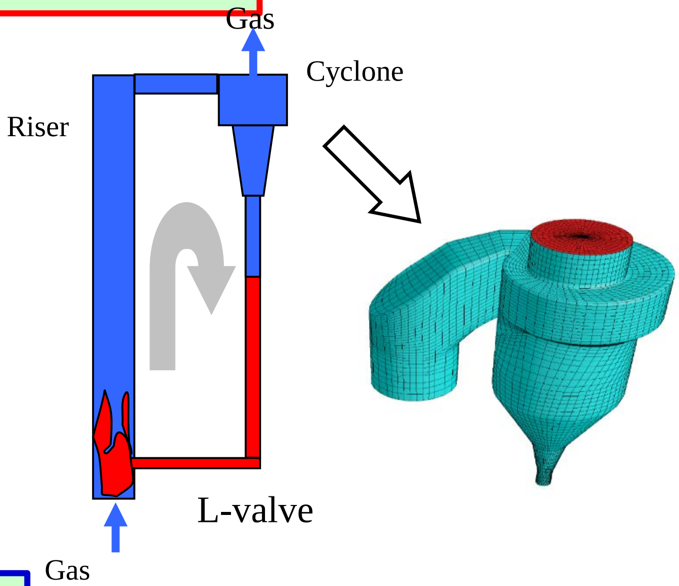
2.Identify the Domain We Will Model

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

2) Where will the computational domain begin and end?

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

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

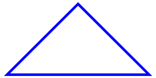


Example: Cyclone Separator

3.Design and Create the Grid

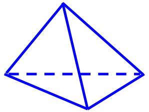
1) Can we use a quad/hex (四边形的/六面体的) grid or should we use a tri/tet (三角形/四面体) grid or hybrid grid?

- **How complex is the geometry and flow?**

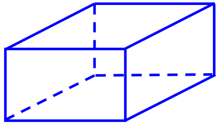


Triangle
三角形

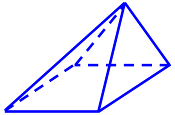
Quadrilateral
四边形



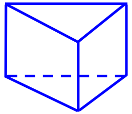
Tetrahedron
四面体



Hexahedron
六面体



Pyramid
金字塔



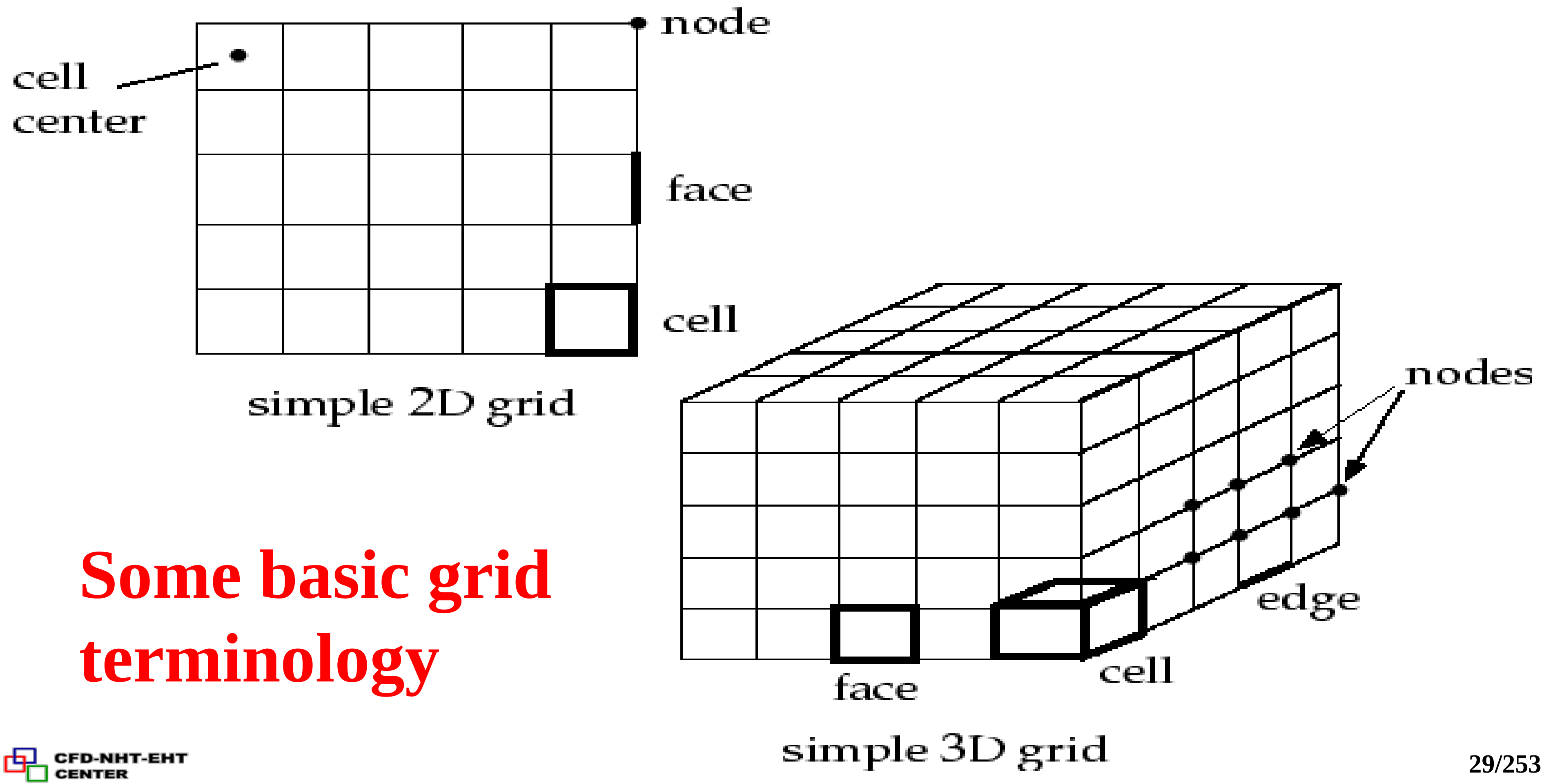
Prism/wedge
五面体

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

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

3) Do we have sufficient computer memory?

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



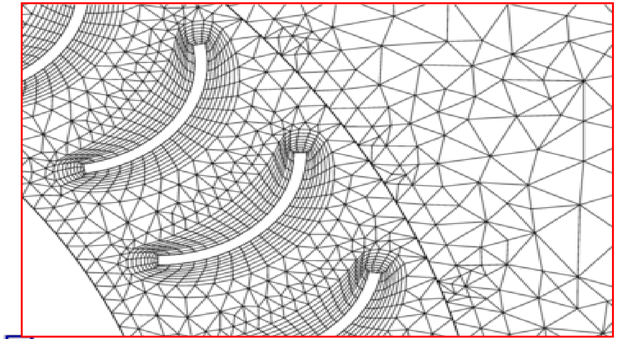
Some basic grid terminology

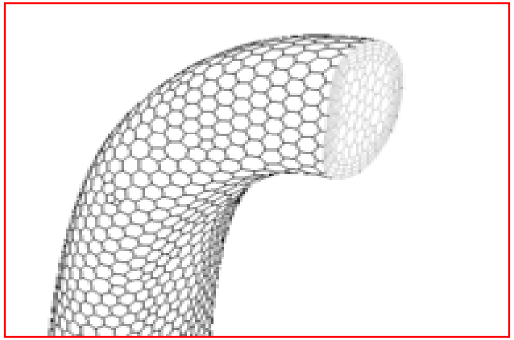
Mesh Terminology

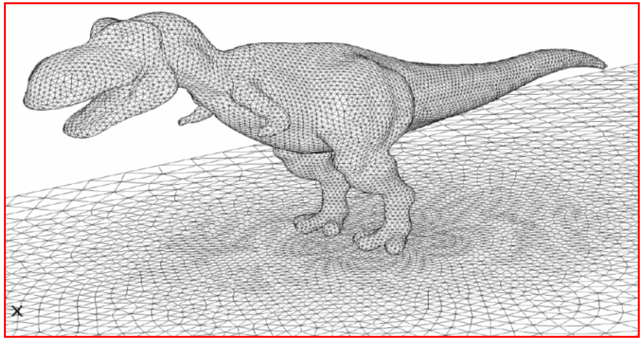
Cell	Control volume
Cell center	Location where cell data is stored
Face	Boundary of a cell (2D or 3D)
Edge	Boundary of a face (3D)
Node	Grid point
Cell thread	Grouping of cells
Face thread	Grouping of faces
Node thread	Grouping of nodes
Domain	A grouping of node, face, and cell threads

Fluent is an unstructured solver. It uses internal data structures to assign an order to the cells, faces, and grid points in a mesh and to maintain contact between adjacent cells.

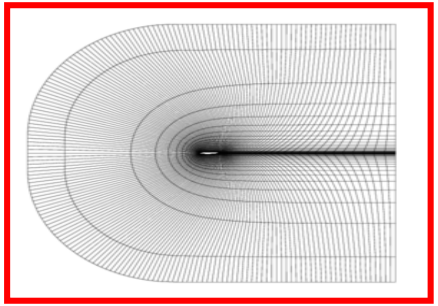
Therefore, it does not require *i, j, k* indexing to locate neighboring cells. This gives us the flexibility to use the best mesh topology for our problem, as the solver does not force an overall structure or topology on the mesh.



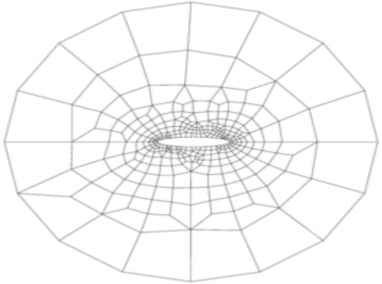




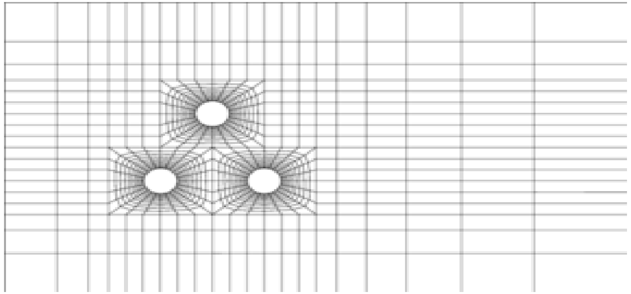
◆ Examples of Acceptable Mesh Topologies



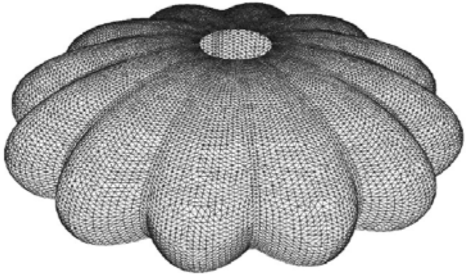
Structured Quad Mesh for an Airfoil



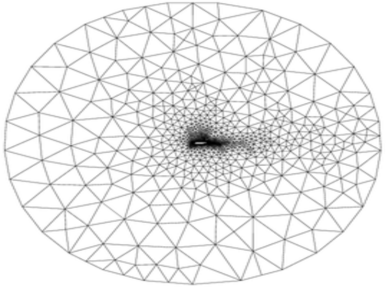
Unstructured Quad Mesh



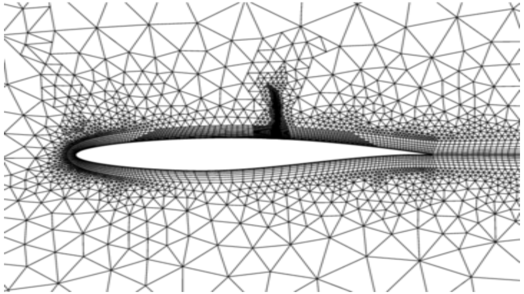
Multiblock Structured Quad Mesh



Parachute Modeled With Zero-Thickness Wall



Unstructured Triangular Mesh for an Airfoil



Hybrid Tri/Quad Mesh with Hanging Nodes

◆ Choosing the Appropriate Mesh Type

Fluent can use meshes comprised of triangular or quadrilateral cells (or a combination of the two) in 2D domain, and tetrahedral, hexahedral, polyhedral, pyramid, or wedge cells (or a combination of these) in 3D domain.

The choice of which mesh type to use will depend on the actual application. When choosing mesh type, consider the following issues:

① Setup time

② Computational expense

③ Numerical diffusion(false diffusion)(Chapter 5.5, P.152)

(1) Setup Time

Many flow problems solved in engineering practice involve complex geometries. The creation of structured or block-structured meshes (consisting of quadrilateral or hexahedral elements) for such problems can be extremely time-consuming.

Therefore, setup time for complex geometries is the major motivation for using unstructured meshes employing triangular or tetrahedral cells. However, if the geometry is relatively simple, there may be no saving in setup time with either approach.

Other risks of using structured or block-structured meshes with complicated geometries include the oversimplification of the geometry, mesh quality issues, and a less efficient mesh distribution(for example, fine resolution in areas of less importance) that results in a high cell count.

(2) Computational Expense

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements.

Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

A characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells.

A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence.

Therefore, if it is a relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, use a mesh of high-aspect-ratio quadrilateral/ hexahedral cells. The mesh is likely to have far fewer cells than if we use triangular/tetrahedral cells.

The following practices are generally recommended:

① For simple geometries, use quadrilateral/hexahedral(四边形/六面体) meshes.

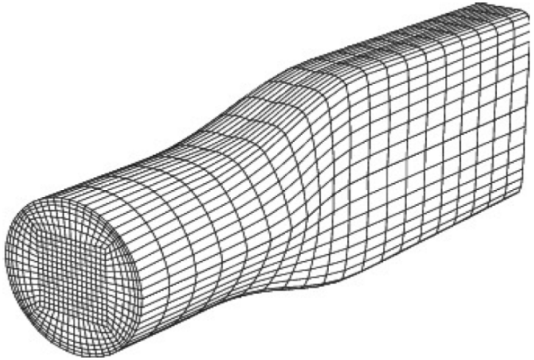
② For moderately complex geometries, use unstructured quadrilateral/hexahedral meshes.

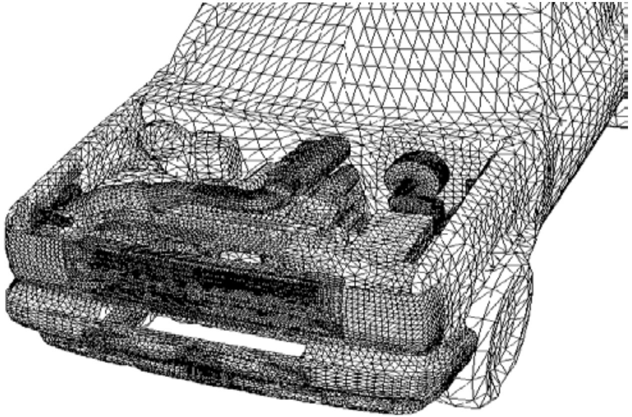
③ For relatively complex geometries, use triangular/tetrahedral(三角形和四面体) meshes with wedge elements in the boundary layers.

④ For extremely complex geometries, use pure triangular/ tetrahedral meshes.

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.





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

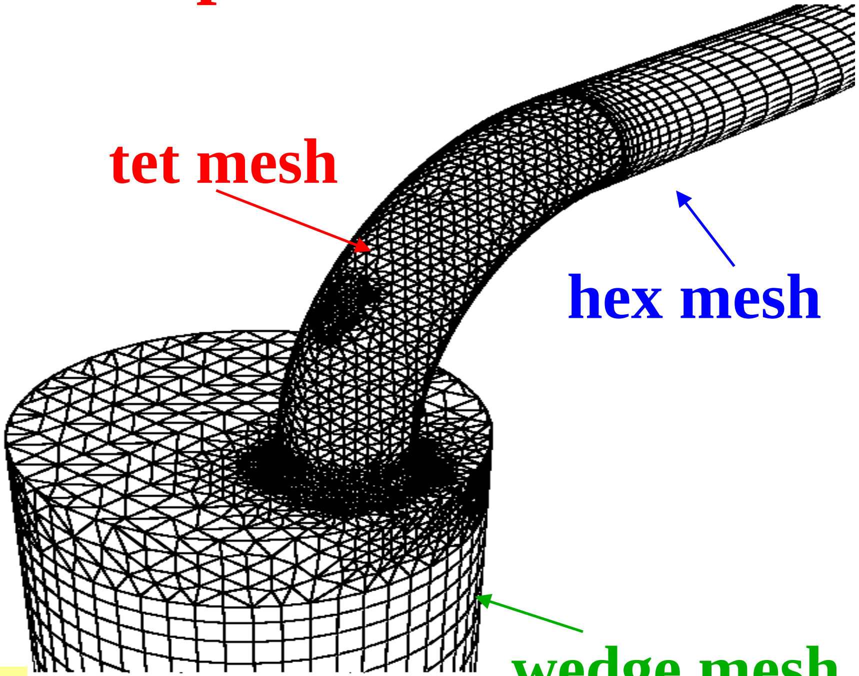
Hybrid Mesh Example

- **Valve port grid**

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

2) Both efficiency and accuracy are enhanced relative to a hex or tet mesh alone.

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



Hybrid mesh for an engine valve port

(3). Numerical diffusion(False Diffusion) (Chapter 5.5, P.152)

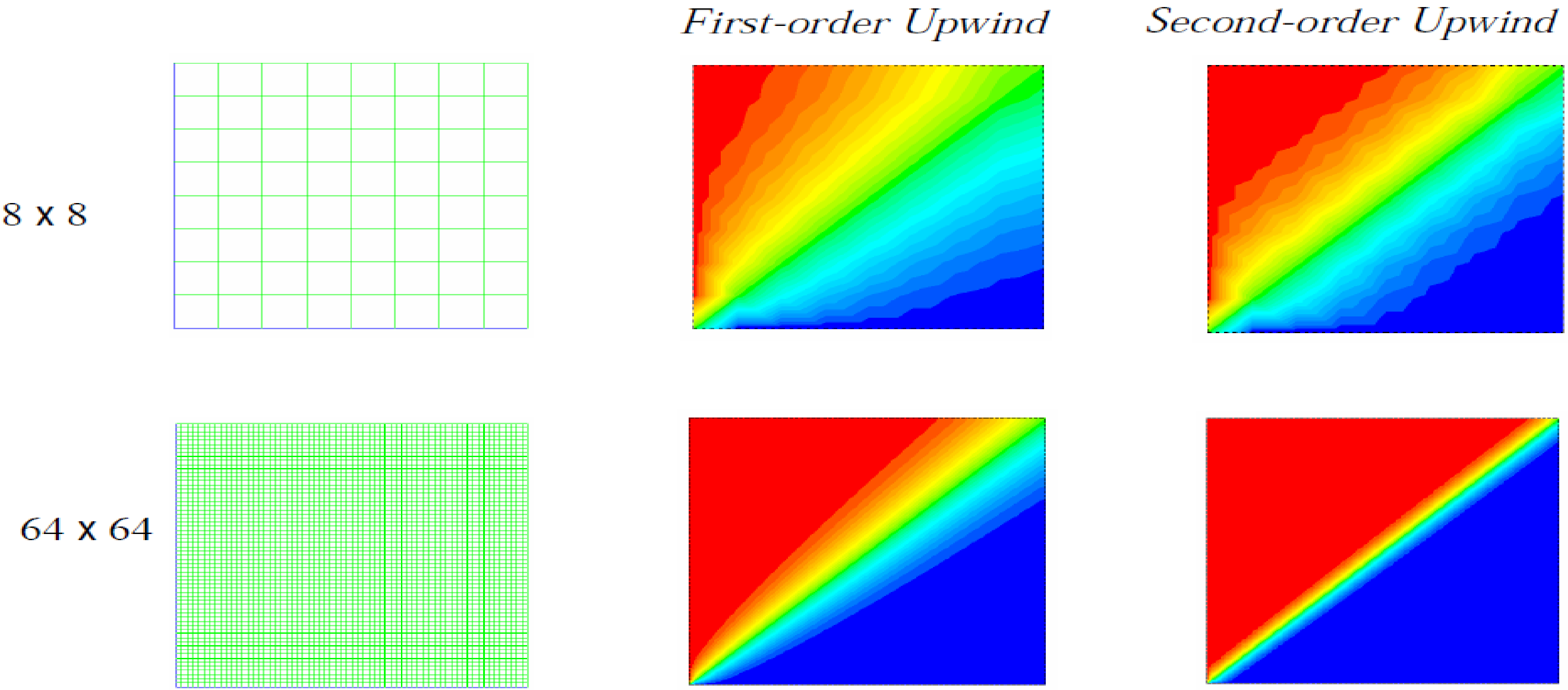
A dominant source of error in multidimensional situations is false diffusion. Its effect on a flow calculation is similar as that of increasing the real diffusion coefficient.

The following comments can be made about false diffusion:

① **All practical numerical schemes for solving fluid flow contain a finite amount of false diffusion. This is because false diffusion arises from truncation errors (截断误差) that are a consequence of representing the fluid flow equations in discrete form.**

② The second-order upwind, QUICK and the MUSCL discretization scheme used in Fluent can help reduce the effects of false diffusion on the solution.（Chapter 5.6, P162）

③ The amount of false diffusion is inversely related to the resolution of the mesh. i.e. a coarser mesh will have more false diffusion than a more refined mesh. Therefore, one way of dealing with false diffusion is to refine the mesh.



False diffusion

•**False diffusion is minimized when the flow is aligned with the mesh.**

If we use a quadrilateral/hexahedral mesh, this situation might occur, but not for complex flows. It is only in a simple flow, such as the flow through a long duct, in which we can rely on a quadrilateral/hexahedral mesh to minimize false diffusion.

In such situations, it is advantageous to use a quadrilateral/hexahedral mesh, since we will be able to get a better solution with fewer cells than if we were using a triangular/tetrahedral mesh.

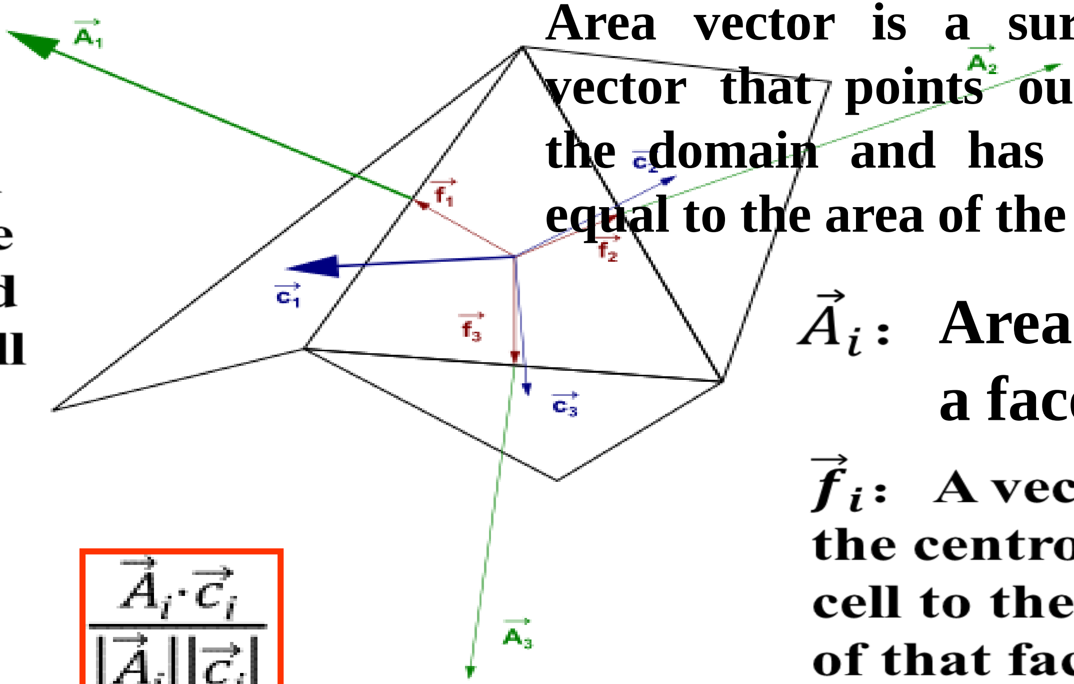
Mesh Requirements and Considerations

Mesh Quality- Orthogonal quality (正交质量)

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Regardless of the type of mesh used in our domain, checking the quality of our mesh is essential. One important indicator of mesh quality that Fluent allows to check is a quantity referred to as the orthogonal quality.

The worst cells will have an orthogonal quality closer to 0 and the best cells will have an orthogonal quality closer to 1.

Area vector is a surface normal vector that points outwards from the domain and has a magnitude equal to the area of the wall cell face



$\vec{c}_i$: A vector from the centroid of the cell to the centroid of the adjacent cell that shares that face

$\vec{A}_i$: Area vector of a face

$\vec{f}_i$: A vector from the centroid of the cell to the centroid of that face

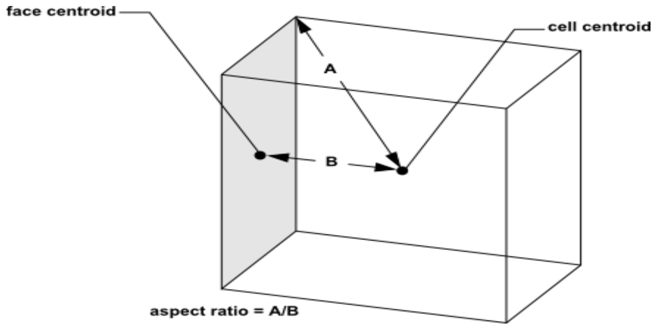
$$\boxed{\frac{\vec{A}_i \cdot \vec{f}_i}{|\vec{A}_i||\vec{f}_i|}} \qquad \boxed{\frac{\vec{A}_i \cdot \vec{c}_i}{|\vec{A}_i||\vec{c}_i|}}$$

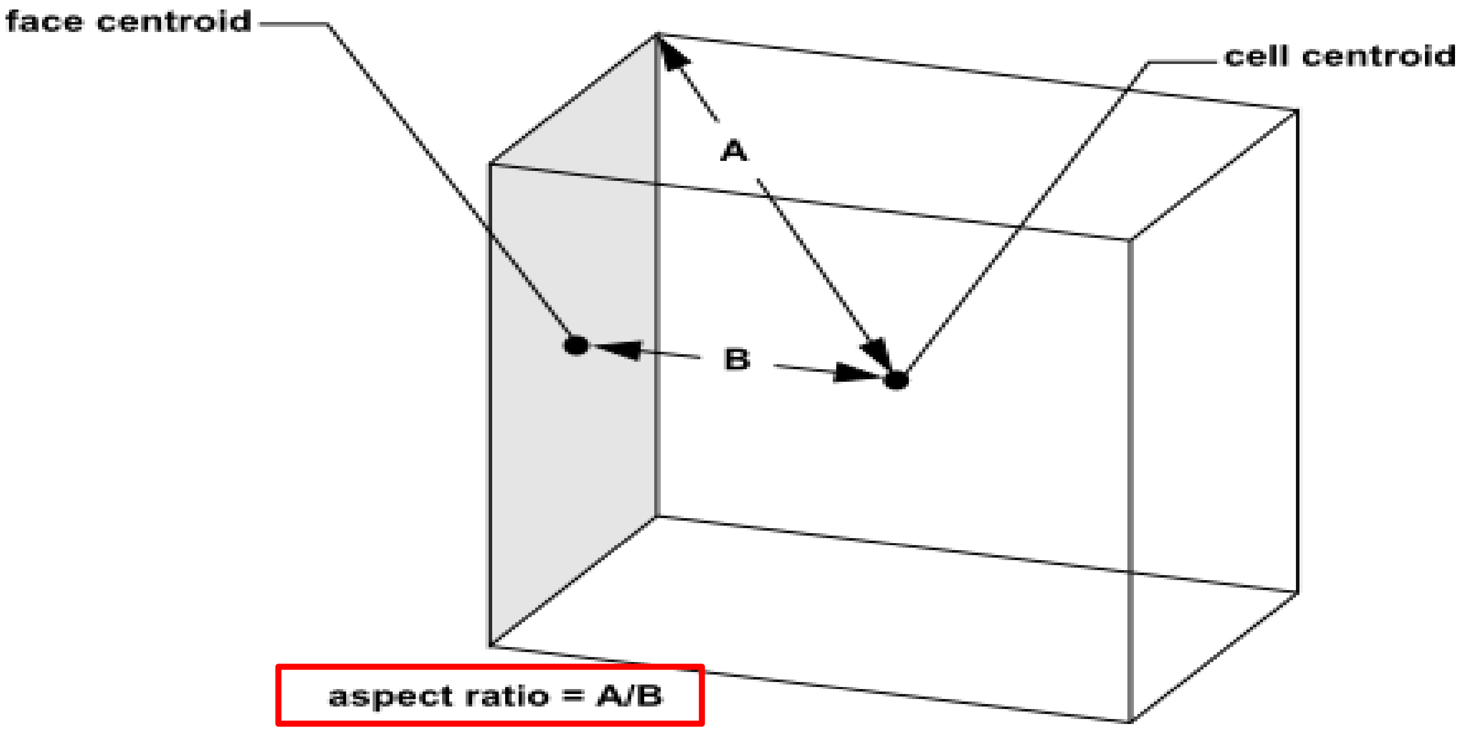
The minimum value of the two above equations for all of the faces is defined as the orthogonal quality for the cell.

Mesh Quality- Aspect ratio

Another important indicator of mesh quality is aspect ratio(长宽比). The aspect ratio is a measure of the stretching of a cell.

It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids (computed as a dot product of the distance vector and the face normal), and the distances between the cell centroid and nodes.





The higher of this ratio, the stretching of the cell is obvious.

◆ Mesh Element Distribution

Since it is discretely defining a continuous domain, the degree to which the salient features of the flow (such as shear layers, separated regions, shock waves, boundary layers, and mixing zones) are resolved depends on the density and distribution of mesh elements.

In many cases, poor resolution in critical regions can dramatically affect results. For example, the prediction of separation due to an adverse pressure gradient depends heavily on the resolution of the boundary layer upstream of the point of separation.

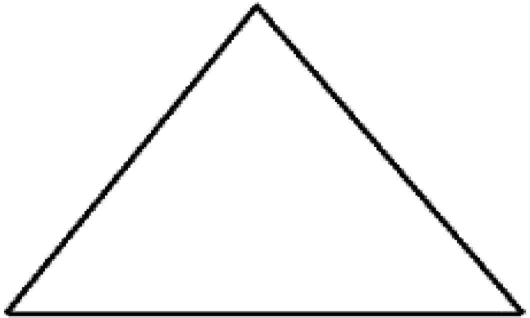
Resolution of the boundary layer (i.e., mesh spacing near walls) also plays a significant role in the accuracy of the computed wall shear stress and heat transfer coefficient. In the near-wall region, different mesh resolutions are required depending on the near-wall model being used.

In general, no flow passage should be represented by fewer than 5 cells. Most cases will require many more cells to adequately resolve the flow passage. In regions of large gradients, as in shear layers or mixing zones, the mesh should be fine enough to minimize the change in the flow variables from cell to cell.

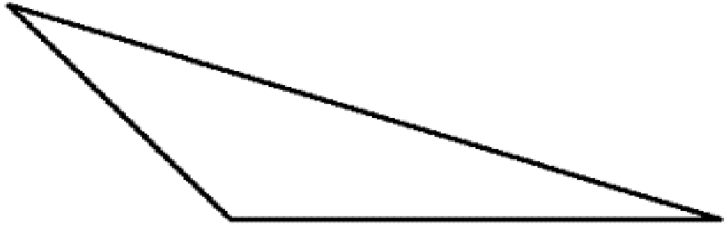
Cell Quality

The quality of the cell (including its orthogonal quality(正交质量), aspect ratio(最大长宽比), and skewness(偏斜度)) also has a significant impact on the accuracy of the numerical solution.

Skewness is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. A general rule is that the maximum skewness for a triangular/tetrahedral mesh in most flows should be kept below 0.95, with an average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties and may require changing the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.

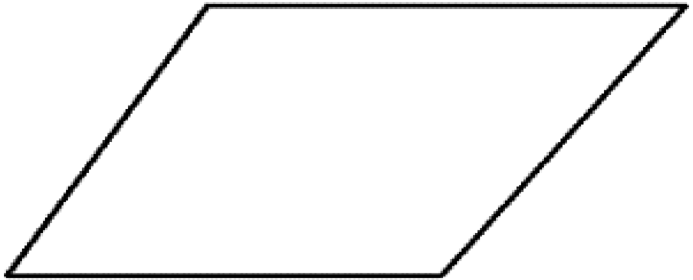


Equilateral Triangle



Highly Skewed Triangle

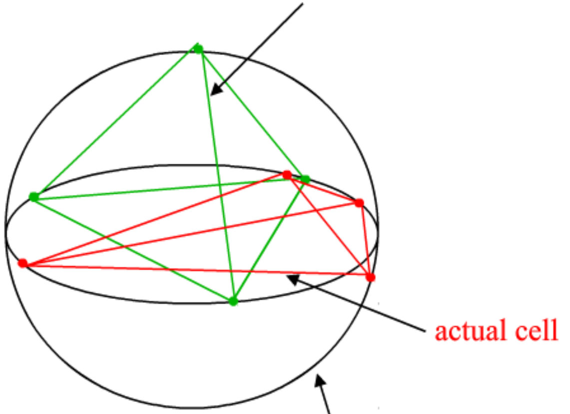
Equiangular Quad



Highly Skewed Quad

Value of Skewness	Cell Quality
1	degenerate
0.9 — <1	bad (sliver)
0.75 — 0.9	poor
0.5 — 0.75	fair
0.25 — 0.5	good
>0 — 0.25	excellent
0	equilateral

The difference of cell and equilateral cell should be as small as possible!



Equilateral-Volume-Based Skewness

$$\text{Skewness} = \frac{\text{Optimal Cell Size - Cell Size}}{\text{Optimal Cell Size}}$$

Mesh Quality（2D）:

Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.

Minimum Orthogonal Quality = 6.07960e-01

Maximum Aspect Ratio = 5.42664e+00

Mesh Quality（3D）:

Minimum Orthogonal Quality = 5.09565e-01

(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)

Maximum Ortho Skew = 4.90435e-01

(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)

Maximum Aspect Ratio = 5.51406e+01

◆ Smoothness

Rapid changes in cell volume between adjacent cells translate into larger truncation errors.

*Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations.(Chapter 2.2, P32,截断误差)

Fluent provides the capability to improve the smoothness by refining the mesh based on the change in cell volume or the gradient of cell volume.

◆ Flow-Field Dependency

The effect of resolution, smoothness, and cell shape on the accuracy and stability of the solution process is dependent on the flow field being simulated. For example, very skewed cells can be tolerated in benign(平缓的) flow regions, but can be very damaging in regions with strong flow gradients.

Since the locations of strong flow gradients generally cannot be determined before the simulation, we should strive to achieve a high-quality mesh over the entire flow domain.

4. Set Up the Numerical Model

- **For a given problem, we will need to:**

① **Select appropriate physical models.**

- **Turbulence, combustion, multiphase, etc.**

② **Define material properties.**

- **Fluid/Solid/Mixture**

③ **Prescribe operating conditions.**

④ **Prescribe boundary conditions at all boundary zones.**

⑤ **Provide an initial solution.**

⑥ **Set up solver controls.**

⑦ **Set up convergence monitors.**

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

5. Compute the Solution

①The discretized conservation equations are solved iteratively.

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

② 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.

③ 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?

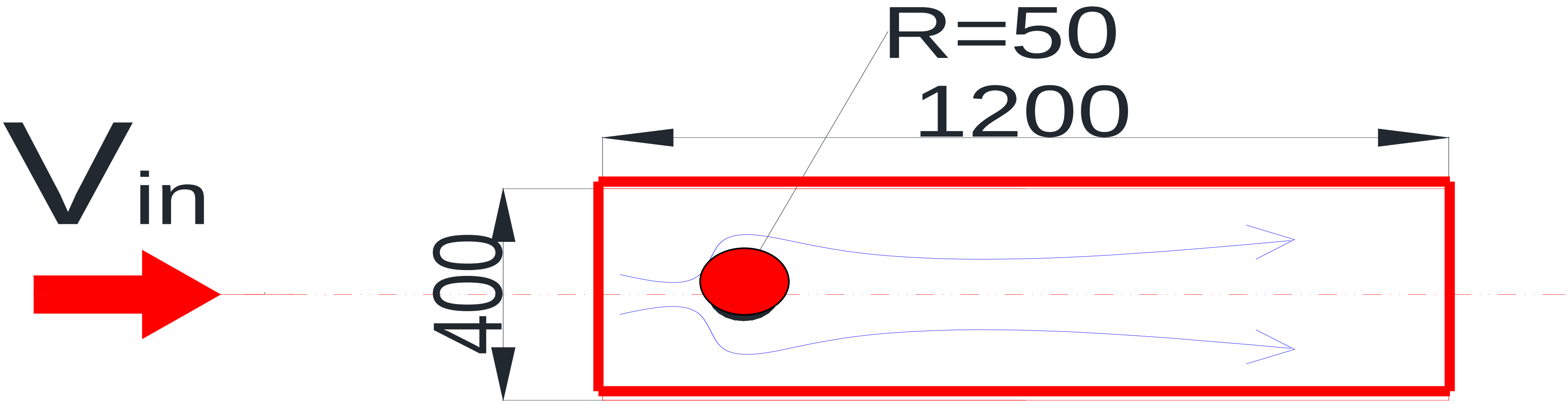
- Is the computational domain large enough?
- Are boundary conditions appropriate?
- Are boundary values reasonable?

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?

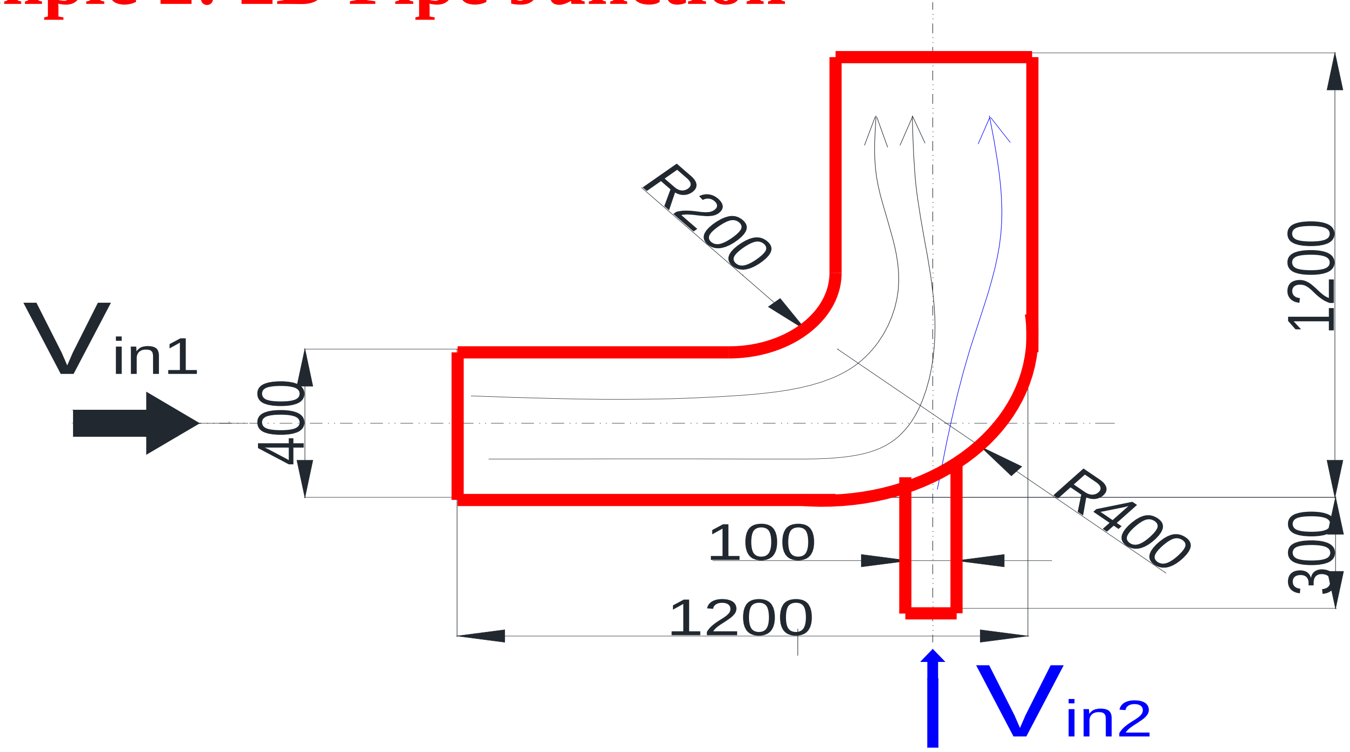
12.3.Simple Examples to Using FLUENT

Example 1: Flow over a cylinder(Kármán vortex street)



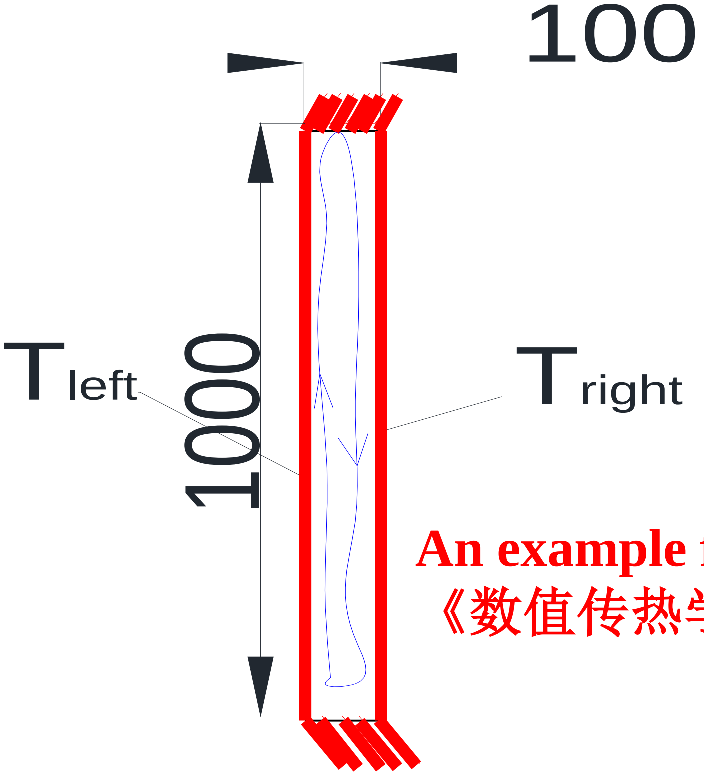
V_{in1}=0.01m/s

Example 2: 2D Pipe Junction



T_{in1}=300K V_{in1}=1m/s T_{in2}=360K V_{in1}=5m/s

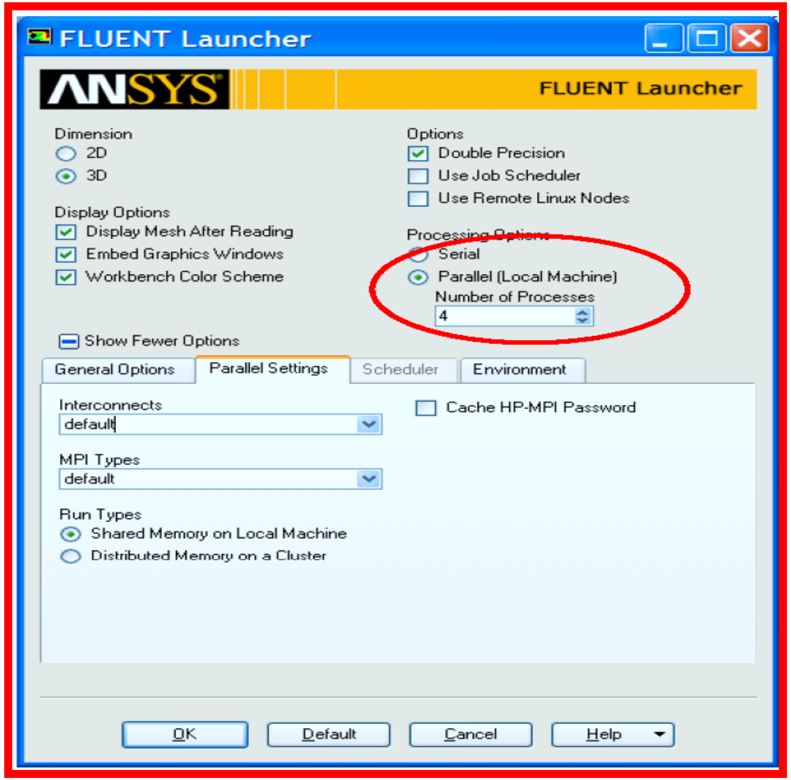
Example 3: Natural convection in a slot

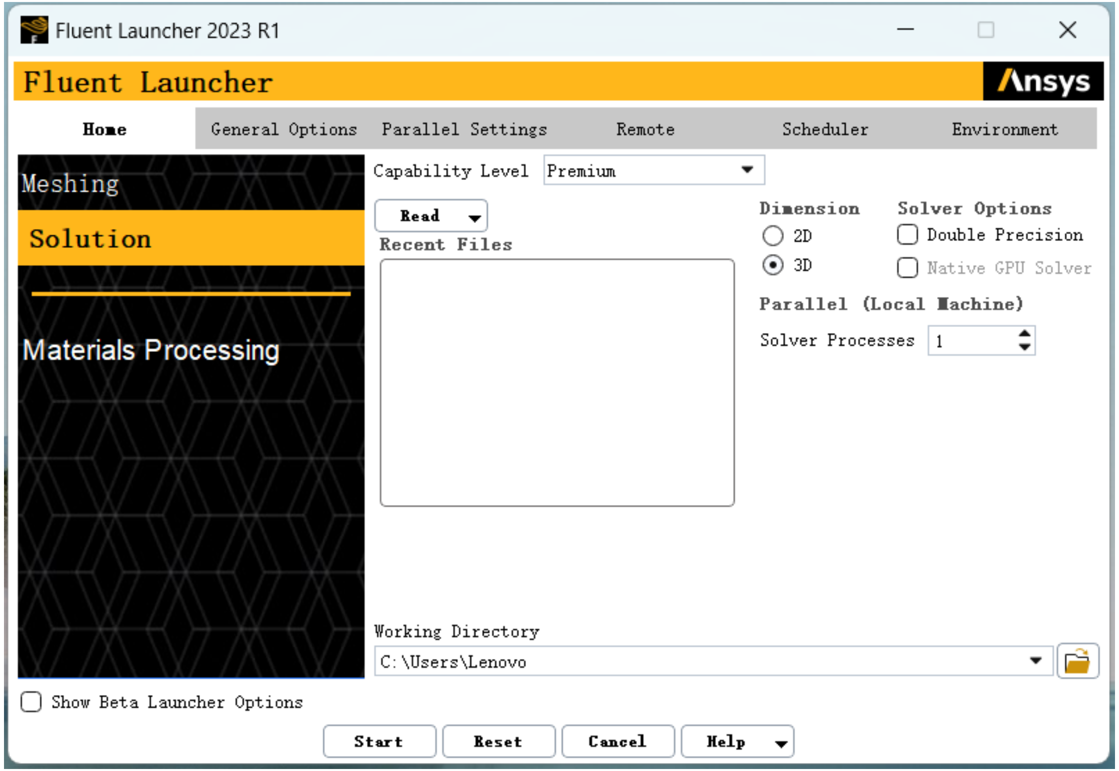


An example from text book(P161)
《数值传热学》第161页例题

T_{left}=320K, T_{right}=300K

12.4.Procedures of Using FLUENT





Fluent launcher interface(2018vs.2023)

◆ Parallel Processing(并行处理)

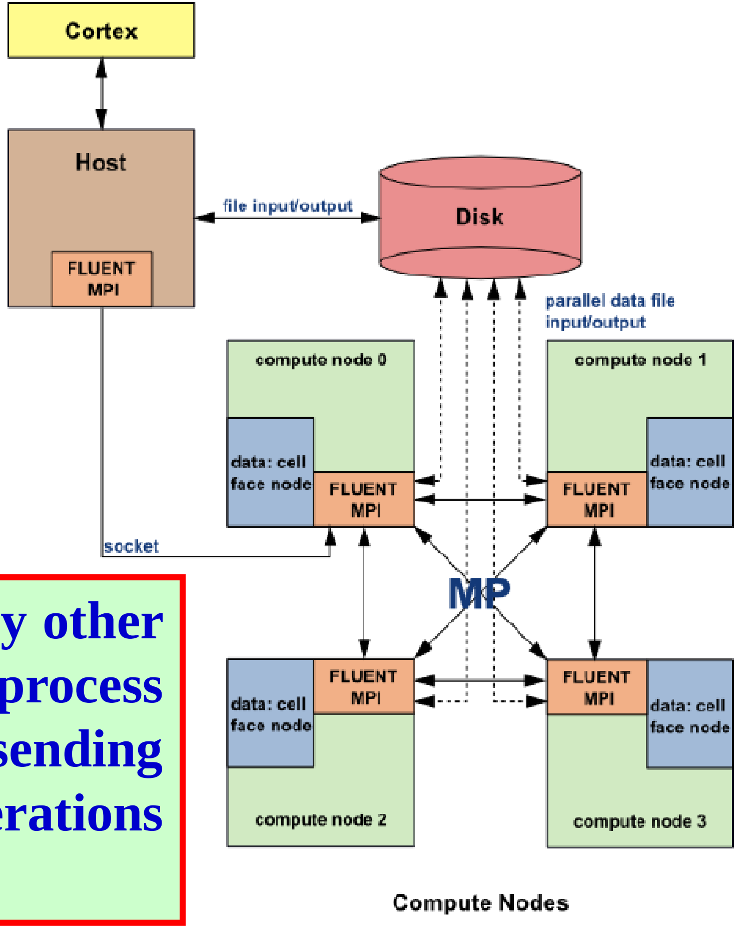
FLUENT can readily be run across many processors in parallel. This will greatly speed up the simulation time.

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. The number of partitions is equal to or less than the number of processors (or cores) available on our computer.

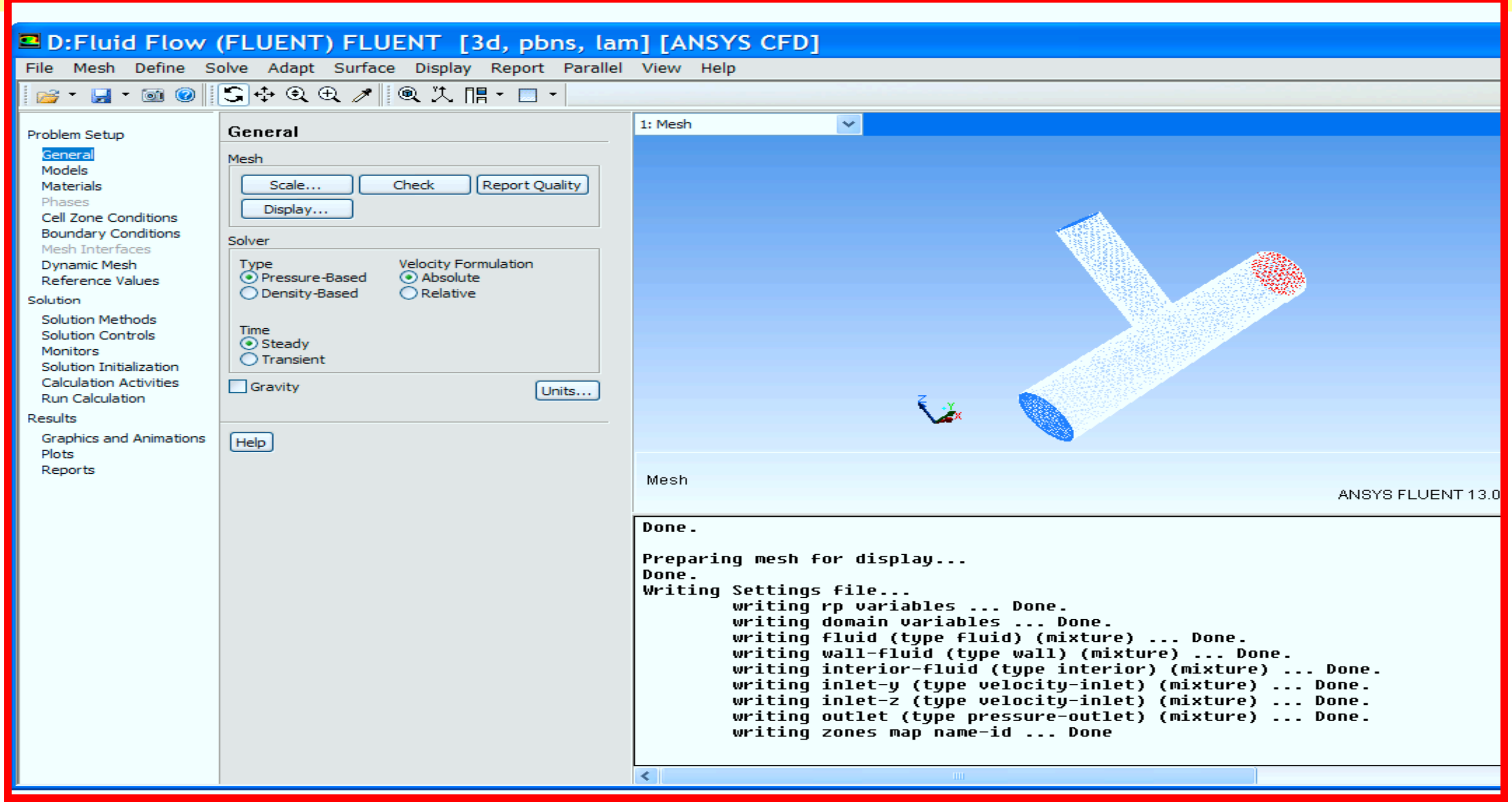
3)Alternatively a distributed parallel cluster(集群) can be set up, and the simulation can run across many machines simultaneously.

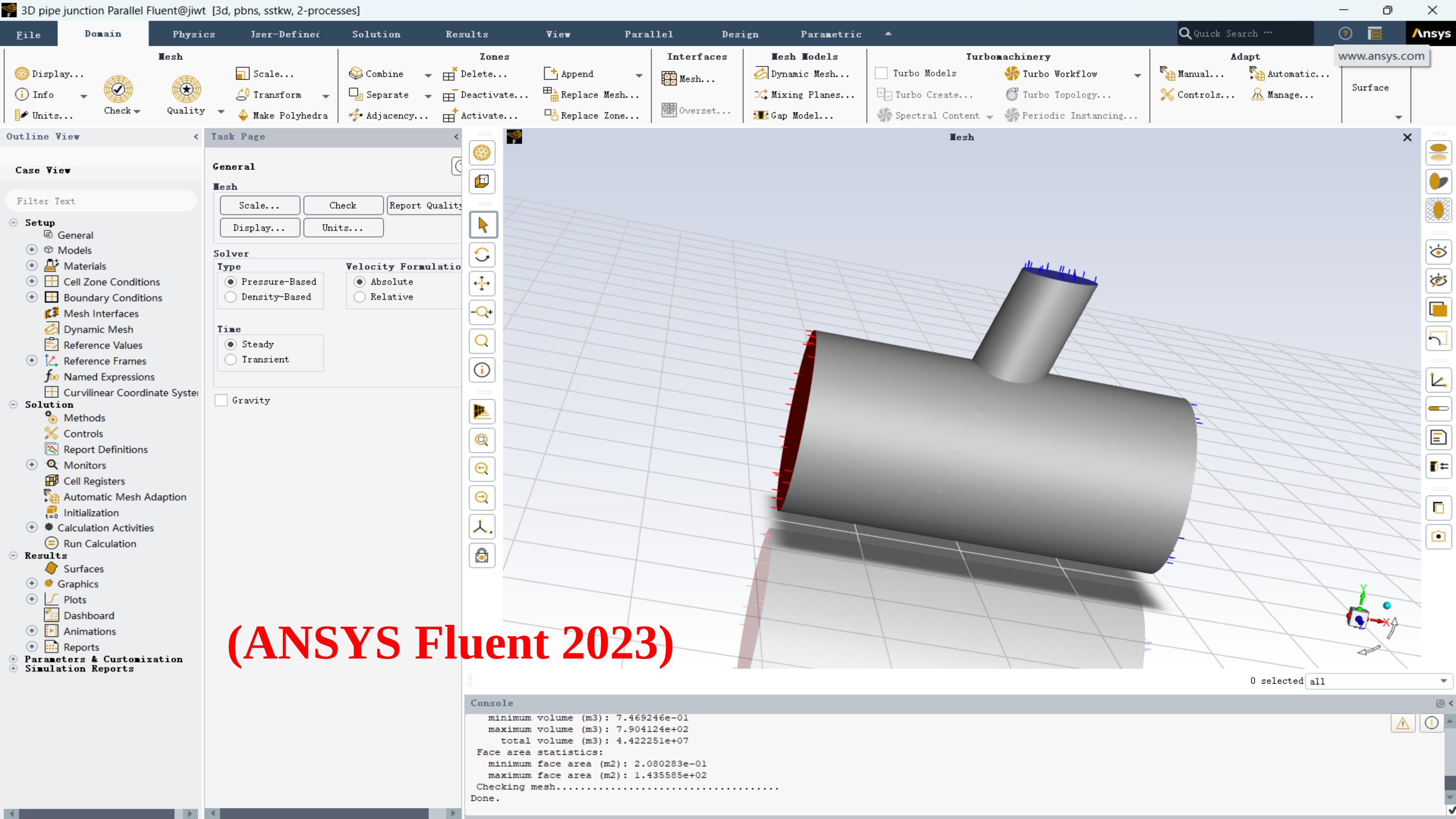
Generally, as the number of compute nodes increases, turnaround time(周转时间) for solutions will decrease. However, beyond a certain point, the ratio of network communication to computation increases, leading to reduced parallel efficiency, so optimal system sizing is important for simulations.



Each compute node is virtually connected to every other compute node, and relies on inter-process communication to perform such functions as sending and receiving arrays, and performing global operations (such as summations over all cells).

- 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.



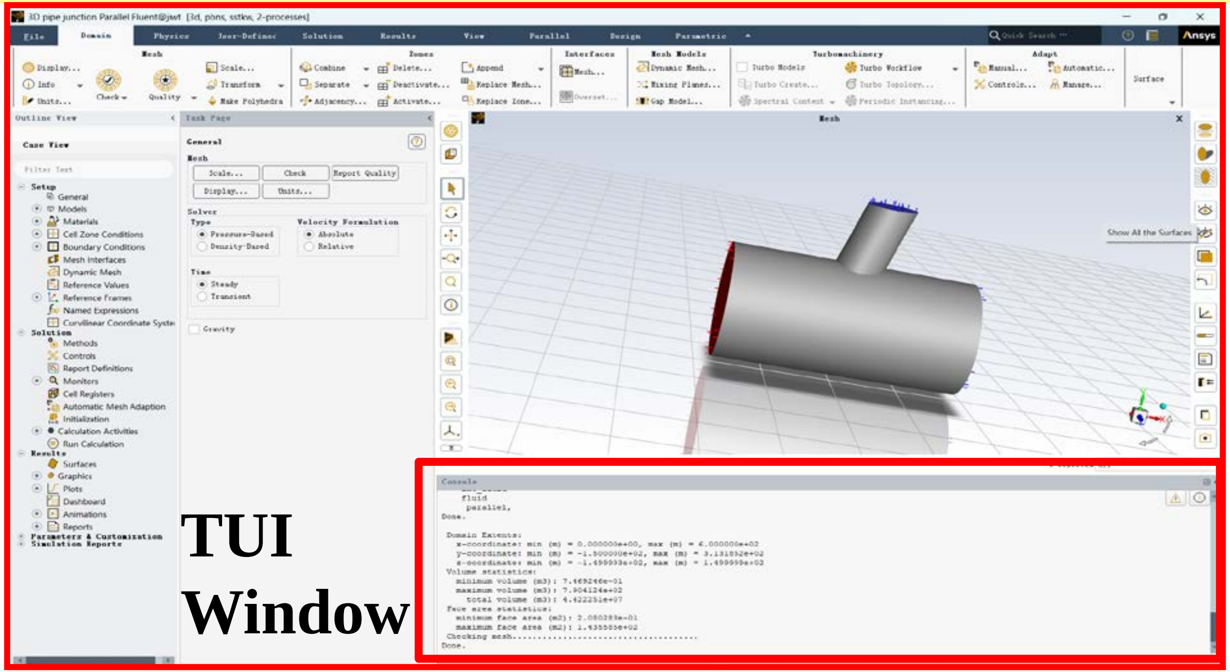

3D pipe junction Parallel Fluent@jiwt [3d, pbns, sstkw, 2-processes]
File
Domain
Physics
User-Defined
Solution
Results
View
Parallel
Design
Parametric
Quick Search
Ansys
Mesh
Display...
Info
Units...
Check
Quality
Scale...
Transform
Make Polyhedra
Zones
Combine
Separate
Adjacency...
Delete...
Deactivate...
Activate...
Append
Replace Mesh...
Replace Zone...
Interfaces
Mesh...
Overset...
Mesh Models
Dynamic Mesh...
Mixing Planes...
Gap Model...
Turbomachinery
Turbo Models
Turbo Create...
Spectral Content
Turbo Workflow
Turbo Topology...
Periodic Instancing...
Adapt
Manual...
Controls...
Automatic...
Manage...
Surface
www.ansys.com
Outline View
Case View
Filter Text
Setup
General
Models
Materials
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Curvilinear Coordinate System
Solution
Methods
Controls
Report Definitions
Monitors
Cell Registers
Automatic Mesh Adaption
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Dashboard
Animations
Reports
Parameters & Customization
Simulation Reports
Task Page
General
Mesh
Scale...
Check
Report Quality
Display...
Units...
Solver
Type
Pressure-Based
Density-Based
Velocity Formulation
Absolute
Relative
Time
Steady
Transient
Gravity
Mesh
(ANSYS Fluent 2023)
0 selected
all
Console
minimum volume (m3): 7.469246e-01
maximum volume (m3): 7.904124e+02
total volume (m3): 4.422251e+07
Face area statistics:
minimum face area (m2): 2.080283e-01
maximum face area (m2): 1.435585e+02
Checking mesh......................................
Done.

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.

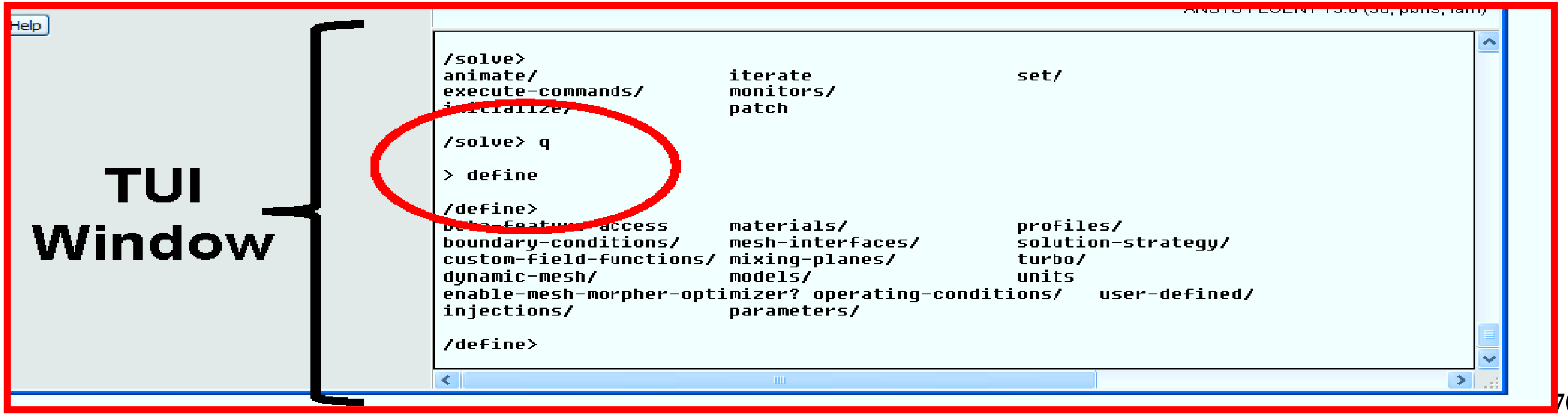
2) Some advanced commands are only available through the TUI.



The TUI offers many valuable benefits:

1) Journal files can be constructed to automate repetitive tasks(自动化重复任务). A journal file is a text file which contains TUI commands which FLUENT will execute sequentially(按顺序执行).

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.



(1).The menu system structure is similar to the directory tree structure of Linux operating systems. When we first start ANSYS FLUENT, we are in the "root" menu.

(2)To generate a listing of the submenus and commands in the current menu, simply press Enter.

><Enter>
adapt/file/report/define/mesh/solve/display/parallel/surface/exit/plot/

(3)By convention, submenu names end with a "/" to differentiate them from menu commands. To execute a command, just type its name (or an abbreviation). Similarly, to move down into a submenu, enter its name or an abbreviation. When we move into the submenu, the prompt will change to reflect the current menu name.

- **Examples of abbreviations of the commands:**
  - **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
```

• FLUENT 2023 GUI Navigation

Selecting the basic items in the tree opens the relevant inputs in the center pane.

① General

② Models

③ Materials

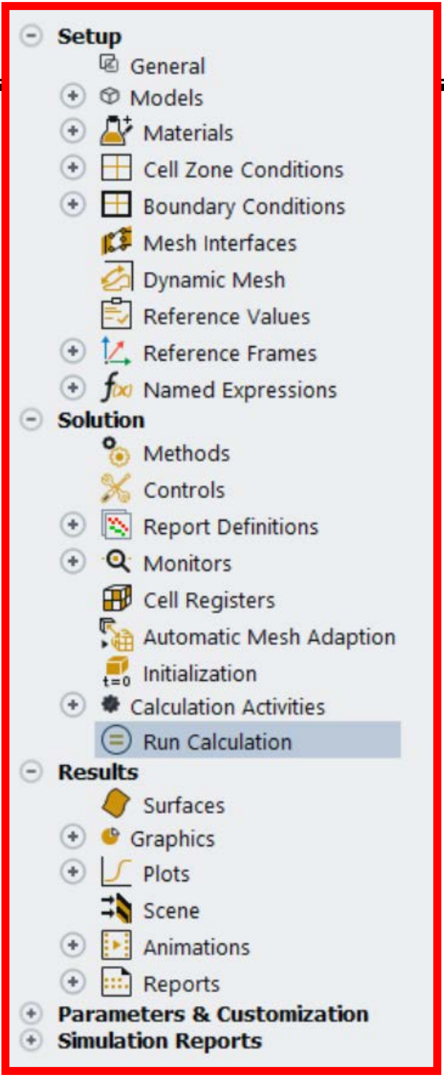
④ Boundary Conditions

⑤ Solution

⑥ Initialization and Calculation

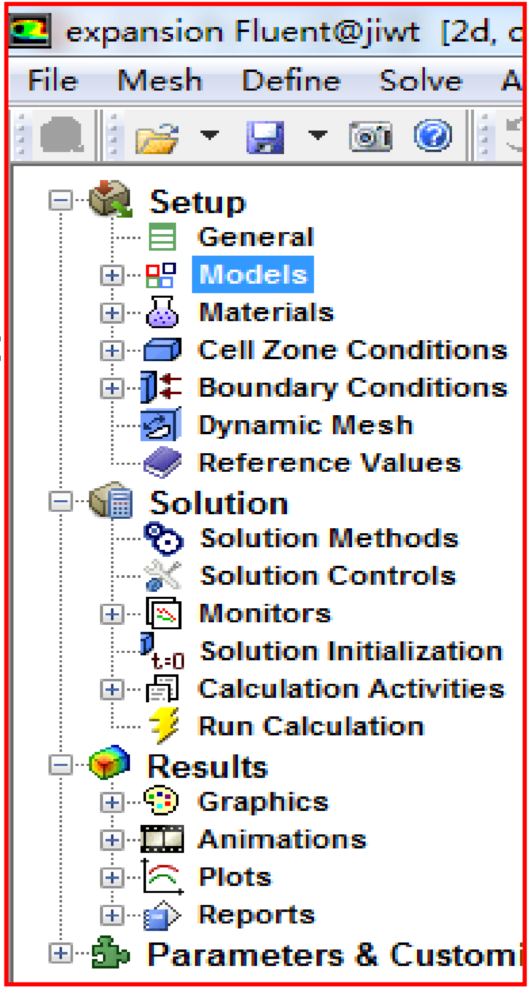
⑦ Results

⑧ Post-processing



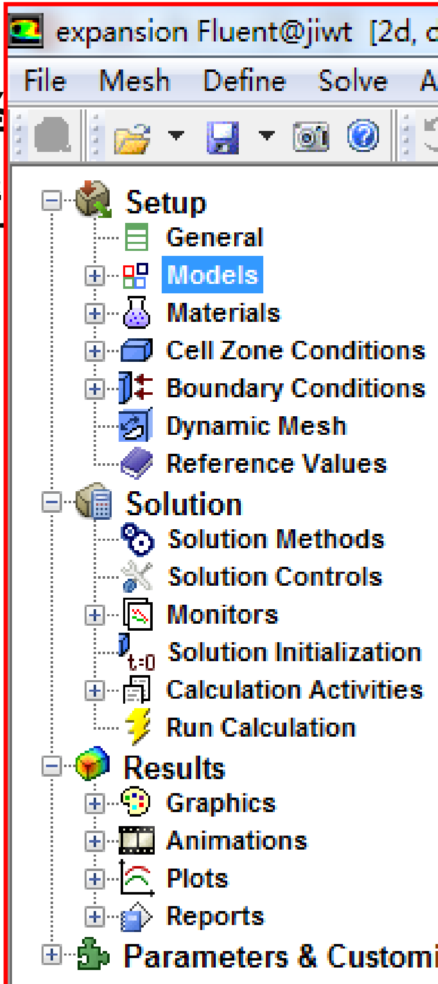
1). Setup：前处理设置

- **General**：一般设置。如设置时间项(瞬态或稳态)、求解器类型(压力基或密度基)等
- **Models**：设置物理模型。如湍流模型、多相流模型等
- **Materials**：设置固体、流体物性参数
- **Cell Zone Conditions**：设置计算域属性
- **Boundary Conditions**：设置边界条件
- **Dynamic Mesh**：设置动网格
- **Reference Values**：设置参考值



2). Solution：求解器设置

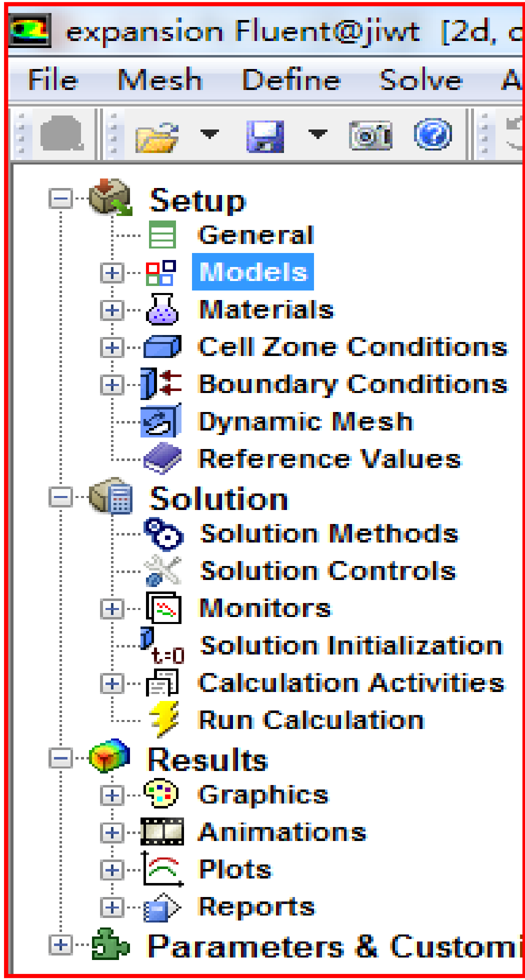
- **Solution Methods：** 求解算法设置，如算法、离散方法选择
- **Solution Controls：** 求解控制参数设置，如亚松弛因子设置
- **Monitors：** 监视器设置
- **Report Definitions：** 定义计算过程中的输出报告
- **Report Files：** 列出定义的报告文件
- **Report Plots：** 定义报告的输出形式
- **Solution Initialization：** 初始化
- **Calculation Activities：** 定义求解中的参数，如自动保存、动画输出等
- **Run Calculation：** 开始计算



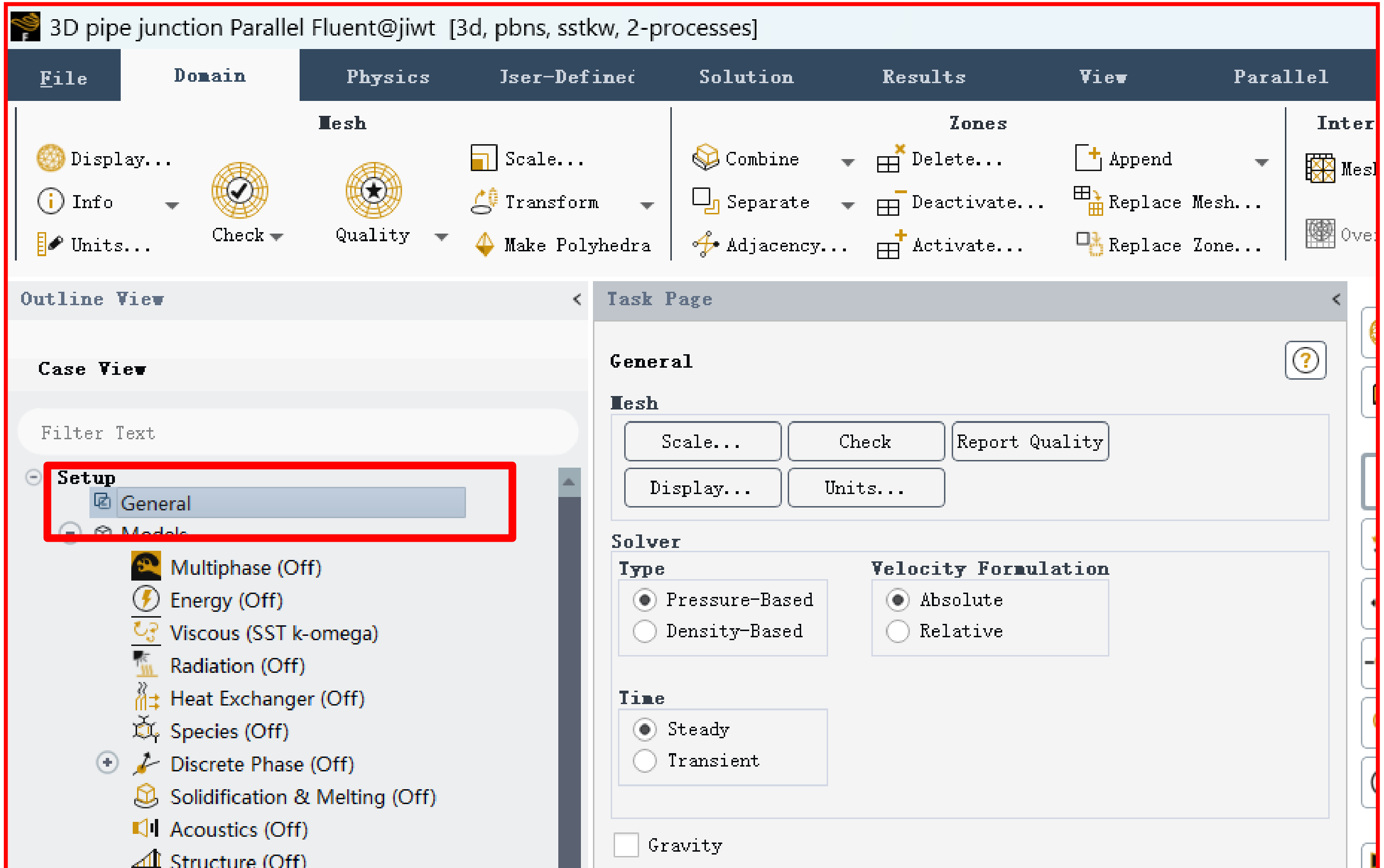
3). Result：计算后处理

- **Graphics：** 显示出各种量化图，如云图、矢量图、流线图等
- **Animations：** 显示动画
- **Plots：** 显示各种线图
- **Reports：** 显示报告

4). Parameters & Customizations：参数化及自定义列表



1. General setting



Mesh files can be created with the mesh generators (ICEM, Fluent meshing and TGrid), or by several third-party CAD packages. The mesh file contains the coordinates of all the nodes, connectivity information that tells how the nodes are connected to one another to form faces and cells, and the zone types and numbers of all the faces.

File → Read → Mesh...

The mesh file does not contain any information on boundary conditions or flow parameters.

◆ Mesh Check Report

```
Mesh Check

Domain Extents:
  x-coordinate: min (m) = -4.000000e-002, max (m) = 2.550000e-001
  y-coordinate: min (m) = 0.000000e+000, max (m) = 2.500000e-002
Volume statistics:
  minimum volume (m3): 2.463287e-009
  maximum volume (m3): 4.508038e-007
   total volume (m3): 4.190433e-004
  minimum 2d volume (m3): 3.000589e-007
  maximum 2d volume (m3): 3.019523e-006
Face area statistics:
  minimum face area (m2): 4.199967e-004
  maximum face area (m2): 2.434403e-003
Checking mesh........................
Done.
```

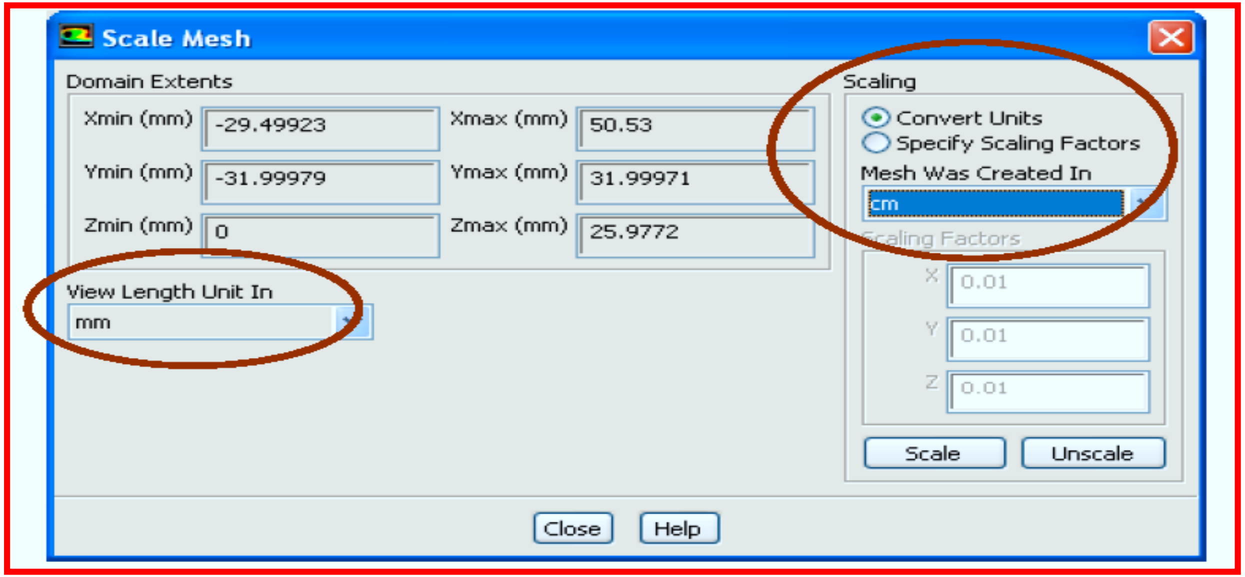
① The mesh check report begins by listing the domain extents. The domain extents include the minimum and maximum x, y, and z coordinates in meters.

② It also display warnings based on the results of the checks previously described.

③ Then the volume statistics are provided, including the minimum, maximum, and total cell volume in m^3. A negative value for the minimum volume indicates that one or more cells have improper connectivity. Cells with a negative volume(负体积) can often be identified using the Iso-Value Adaption dialog box to mark them for adaption and view them in the graphics window.

④ Next, the mesh report lists the face area statistics, including the minimum and maximum areas in m^2. A value of 0 for the minimum face area indicates that one or more cells have degenerated. As with negative volume cells, such faces must be eliminated. It is also recommended to correct cells that have non-zero face areas, if the values are very small.

◆ Scaling the Mesh and Selecting Units

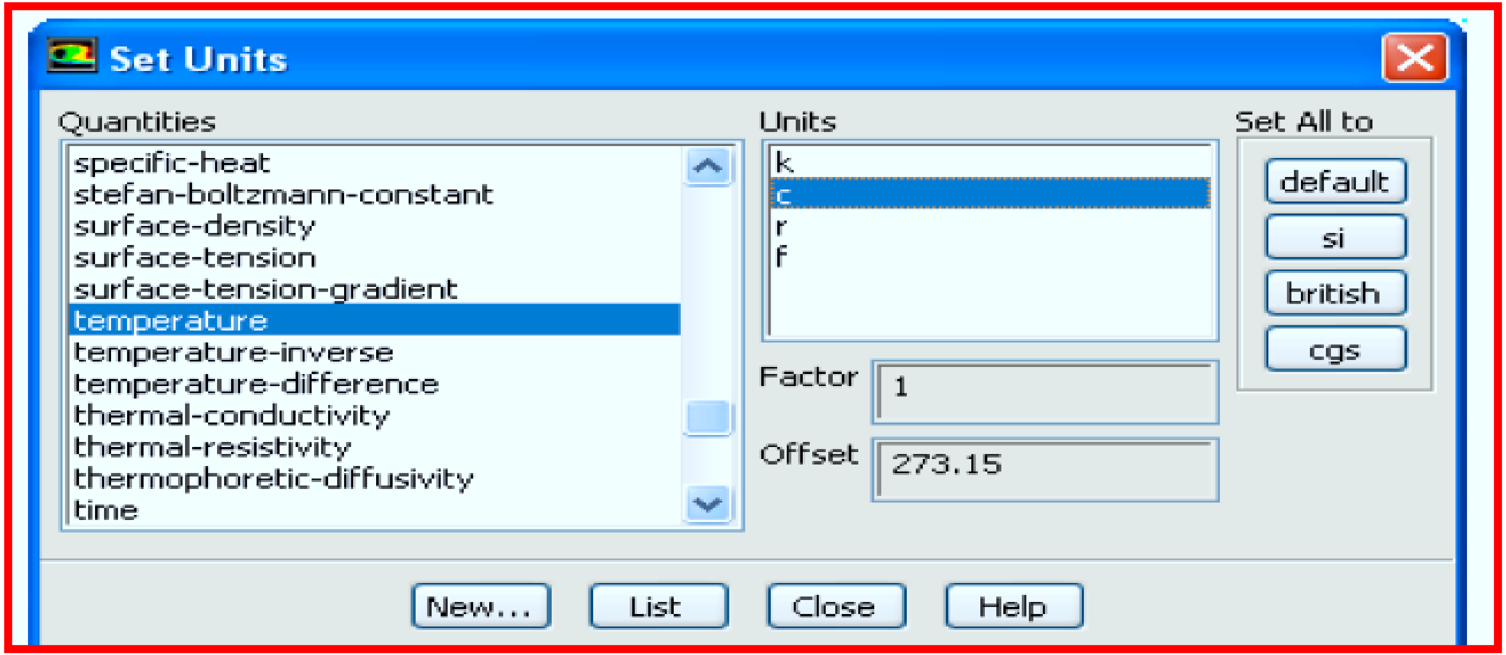


• **When FLUENT reads a mesh file(.msh), all dimensions are assumed to be in units of meters.**

① If our model was not built in meters, then it must be scaled.
② Always verify that the domain extents are correct.

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

① By default, FLUENT uses the international system of units (SI).
② Any units can be specified in the Set Units panel, accessed from the top menu.



Reading and Writing Case and Data Files

Information related to the ANSYS FLUENT simulation is stored in both the case file and the data file.

Case files contain the mesh, boundary and cell zone conditions, and solution parameters for a problem. It also contains the information about the user interface and graphics environment.

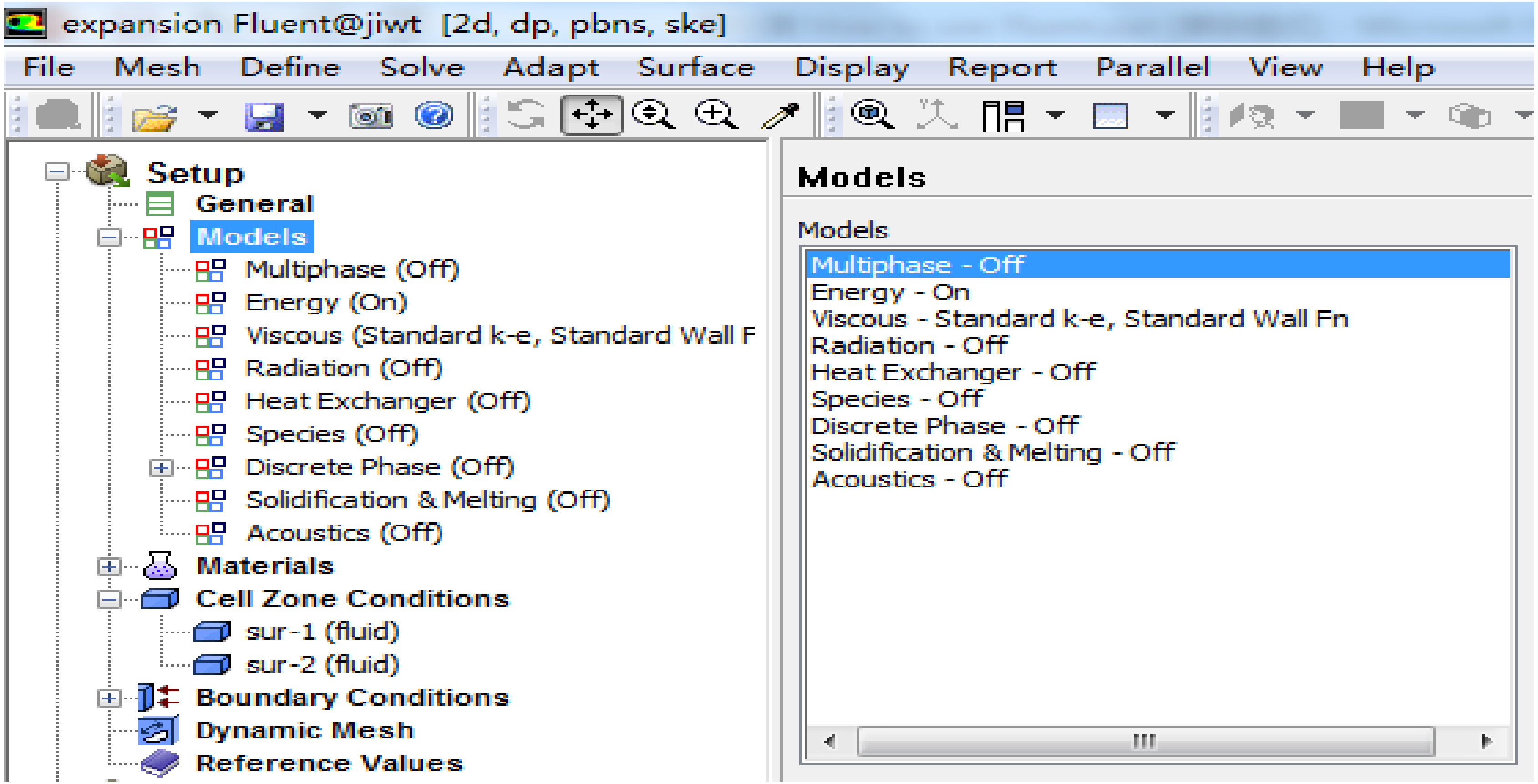
Data files contain the values of the specified flow field quantities in each mesh element and the convergence history (residuals) for that flow field.

File → Read/Write → case/data...

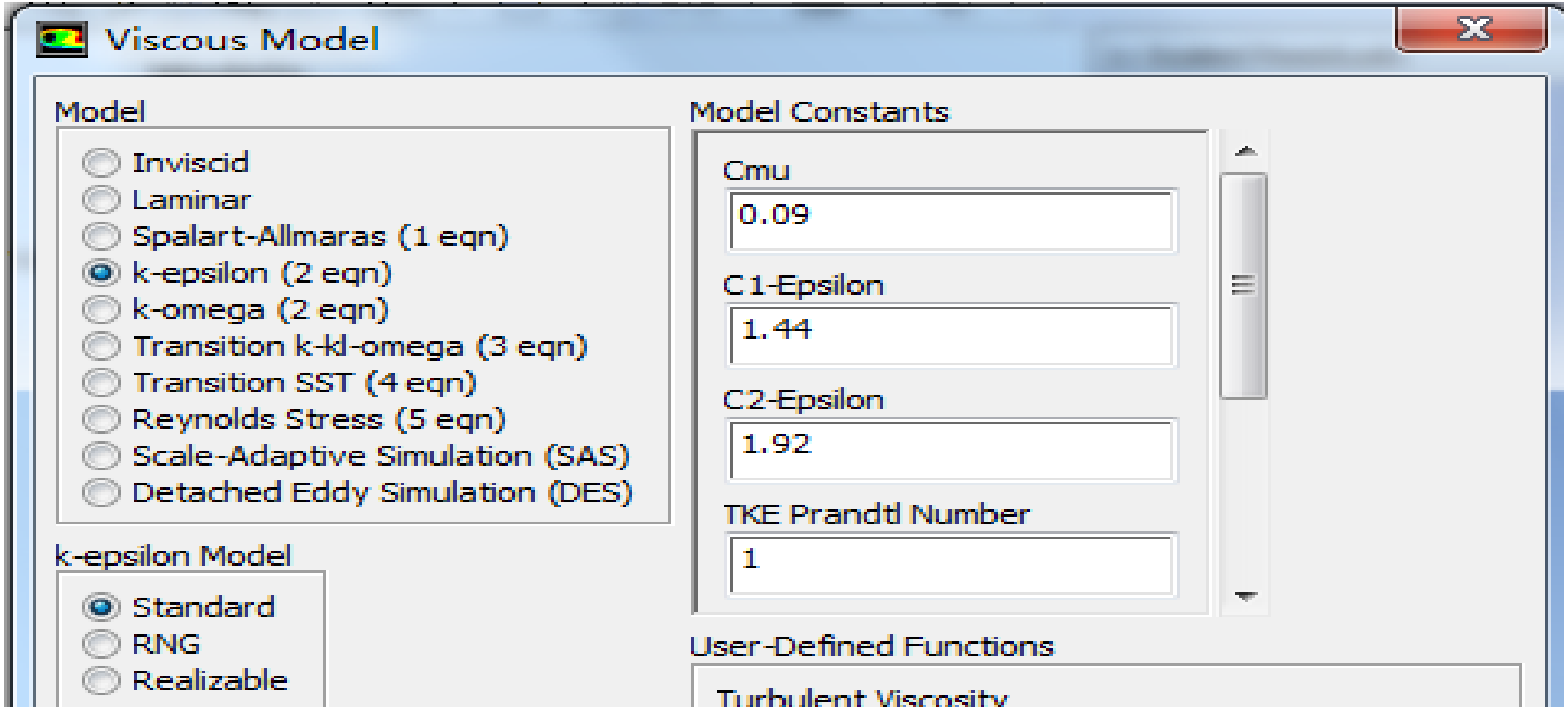
We can read a case file and a data file together:

File → Read/Write → case&data...

2. Setup the Models



◆ Turbulence Models Available in FLUENT(Chapter 9)



It should be noted that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics of the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for our application, we need to understand the capabilities and limitations of the various options.

Compared to laminar flows, simulations of turbulent flows are more challenging in many ways. Since the equations for mean quantities and the turbulent quantities are strongly coupled in a highly non-linear fashion, it takes more computational effort to obtain a converged turbulent solution than to obtain a converged laminar solution.

RANS based models

One Equation Model
Spalart-Allmaras
Two Equation Model
Standard k–ε
RNG k–ε
Realizable k–ε
Standard k–ω
SST k–ω
Two More Equation Models
Reynolds Stress Model
k–kl–ω Transition Model
SST Transition Model
Detached Eddy Simulation
Large Eddy Simulation

Increase in Computational Cost Per Iteration

Chapter 9

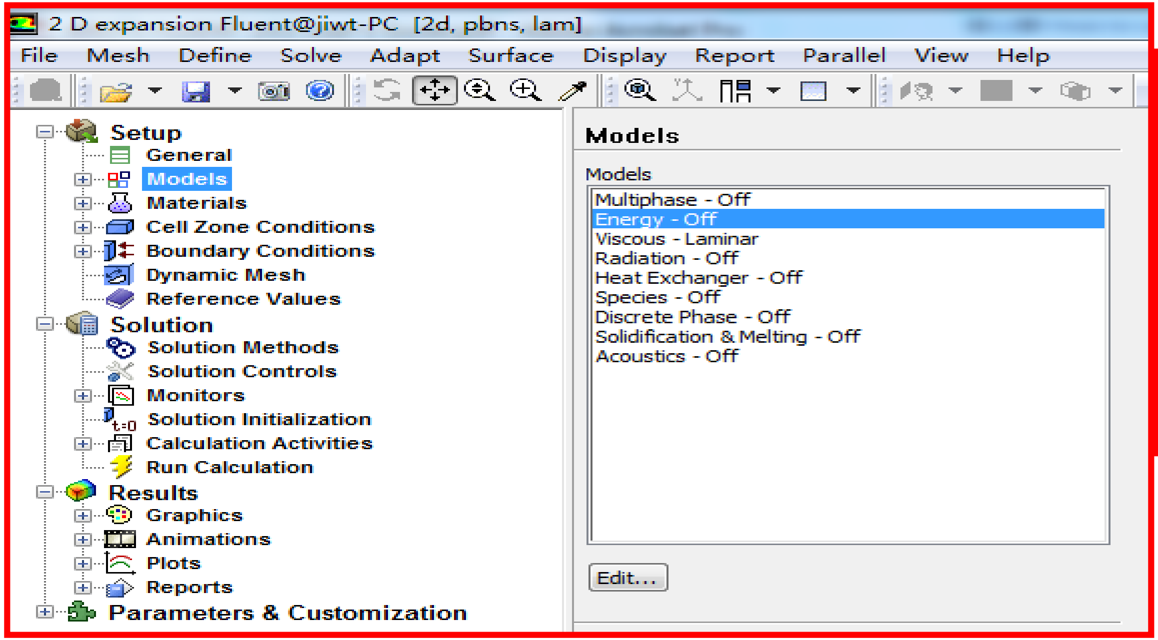
Modeling Heat Transfer

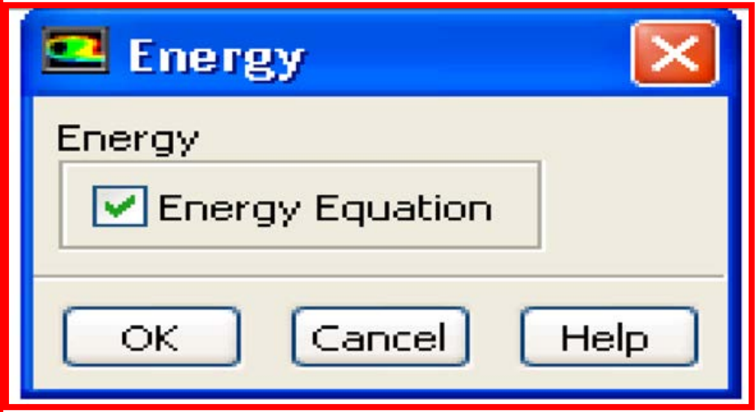
When the ANSYS FLUENT model includes heat transfer we will need to activate the relevant physical models, supply thermal boundary conditions, and input material properties (which may vary with temperature) that govern heat transfer.

Physical models involving conduction and/or convection only are the simplest. While buoyancy-driven flow or natural convection, and flow involving radiation are more complex. Depending on our problem, ANSYS FLUENT will solve a variation of the energy equation that takes into account the heat transfer methods we have specified.

To activate the calculation of heat transfer, enable the Energy Equation option in the Energy dialog box

Models ⟶ Energy ⟶ Edit...





When we are solving the energy equation, we need to define thermal boundary conditions at wall boundaries. Five types of thermal conditions are available:

- **Fixed heat flux**
- **Fixed temperature**
- **Convective heat transfer**
- **External radiation heat transfer**

At flow inlets and exits we can set the temperature of fluid.

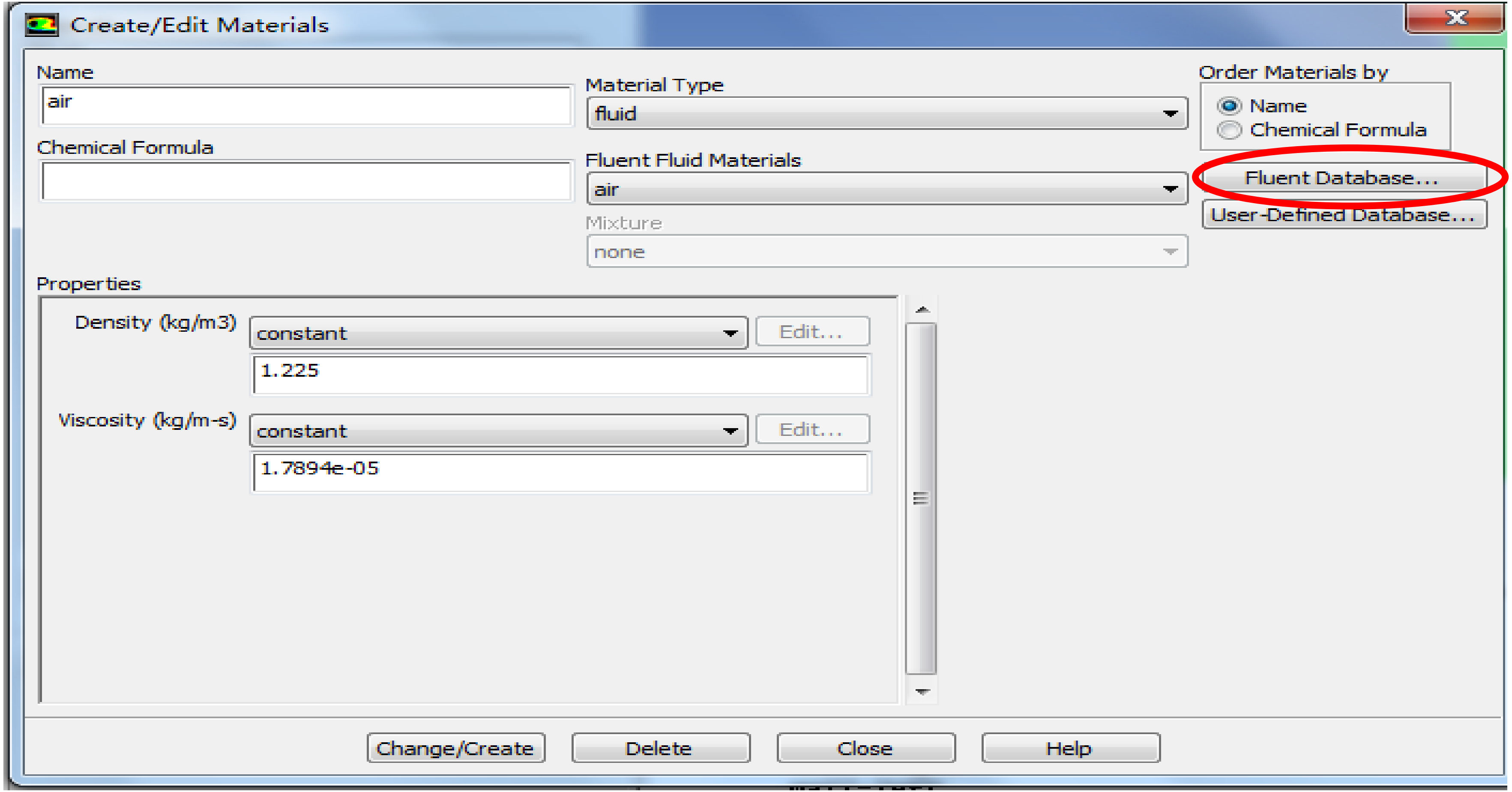
The default thermal boundary condition at inlets is a specified temperature of 300 K; At walls the default condition is zero heat flux (adiabatic).

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:

① Constants
② Functions of temperature
③ Other built in functions following common relationships
④ 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

These properties may include:

① Density and/or molecular weights

② Viscosity

③ Heat capacity

④ Thermal conductivity

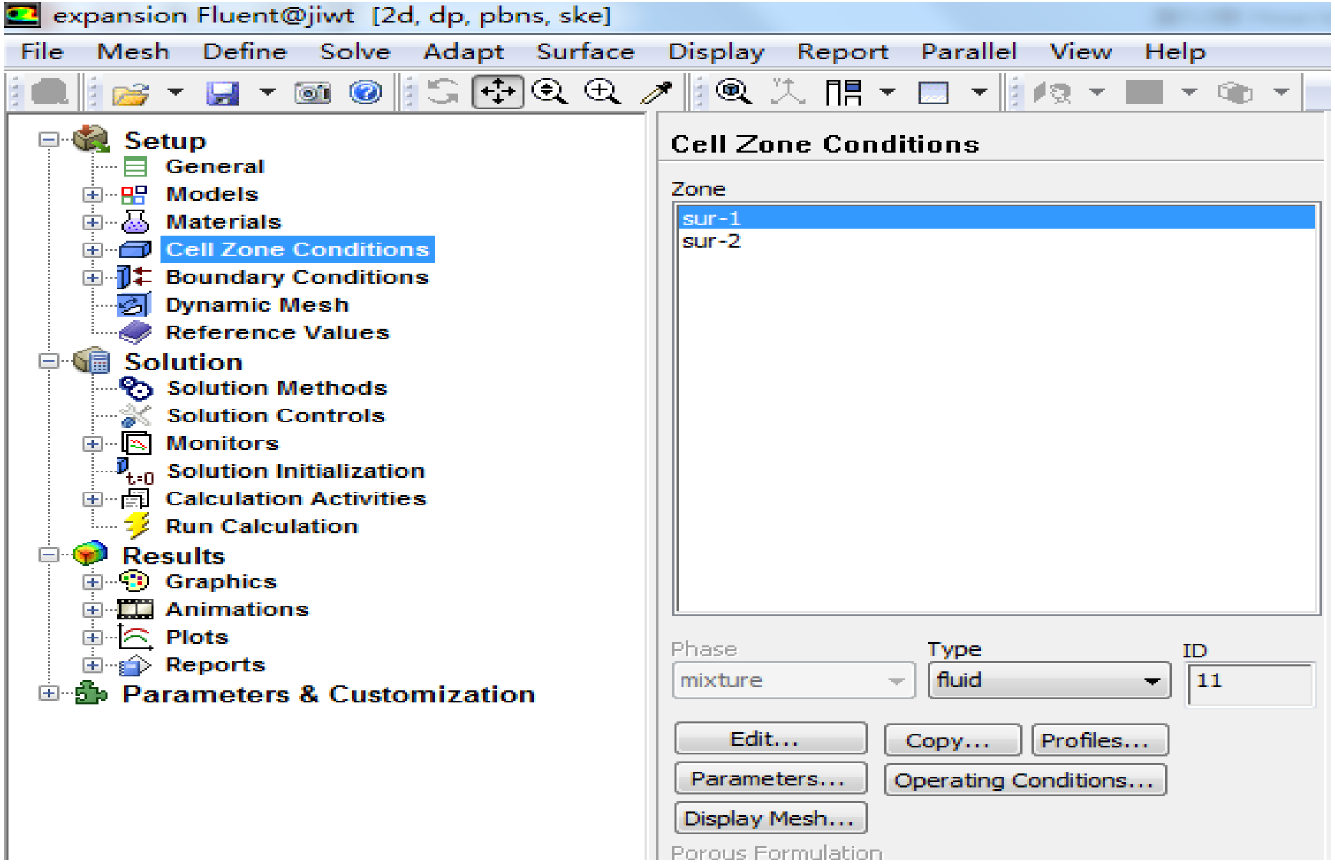
⑤ User-defined scalar diffusivity

⑥ Mass diffusion coefficients

⑦ Standard state enthalpies

⑧ Kinetic theory parameters

4.Cell Zone Conditions

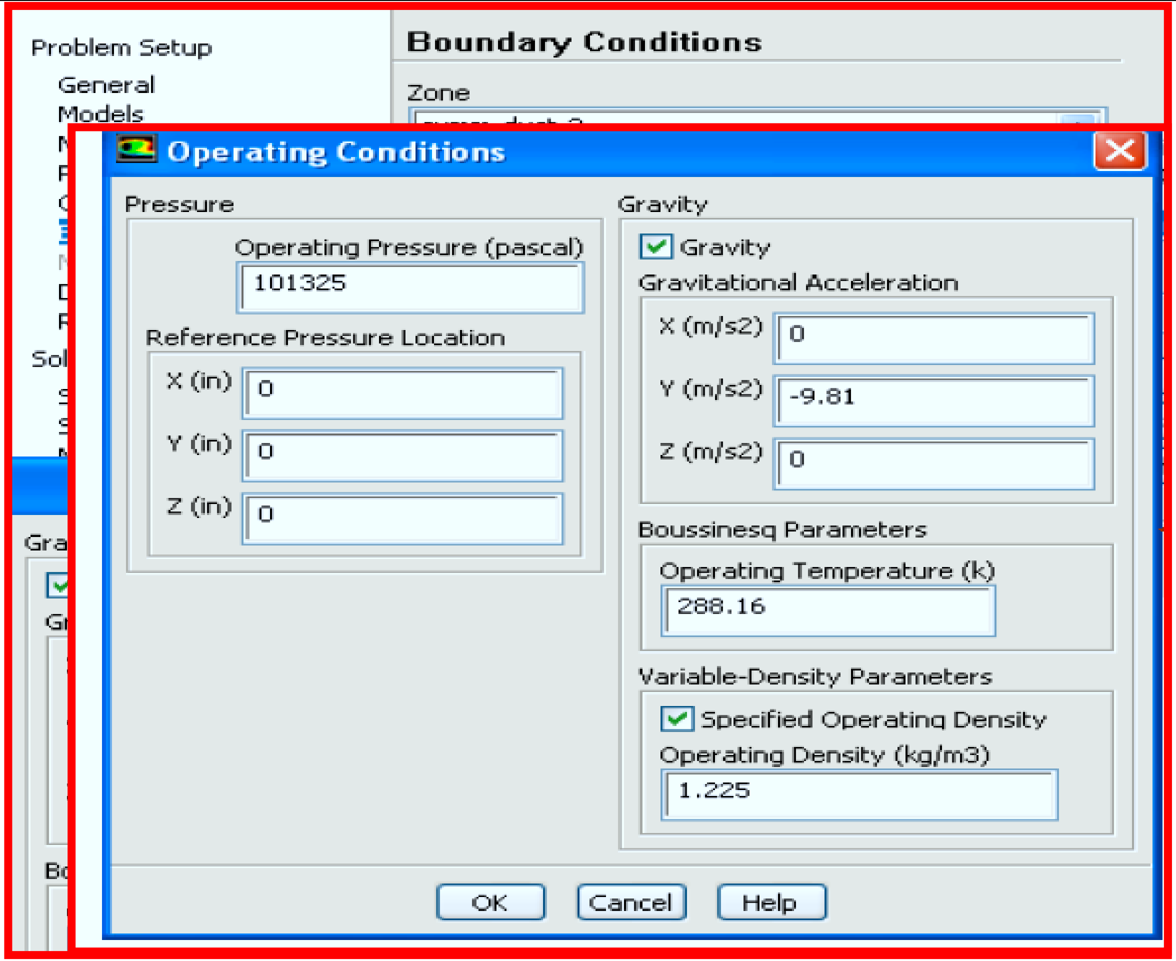


◆ Operating Conditions(操作/运行工况)

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, we must define:

1）Cell zones

① These relate to the middle of the grid cells

② Typically this always involves setting up which material (fluid) is in that cell

③ Other values（heat sources, etc.)

2）Boundary conditions

①Where fluid enters or leaves the domain, the conditions must be set (velocity/pressure/temperature)

②Other boundaries also need declaring, like walls (smooth/rough, heat transfer?)

③There 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.

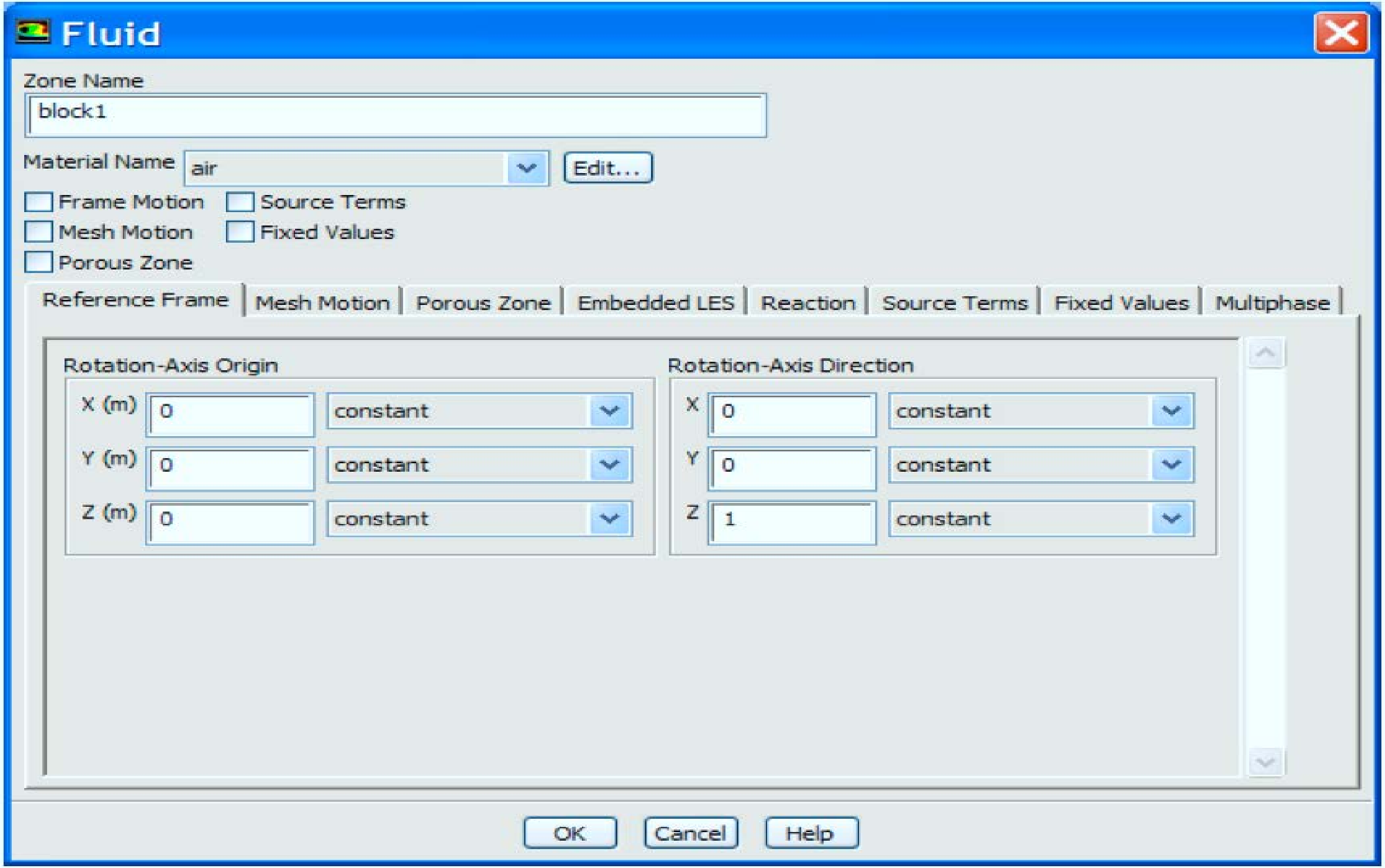
- Optional inputs

① Moving zones

② Porous region

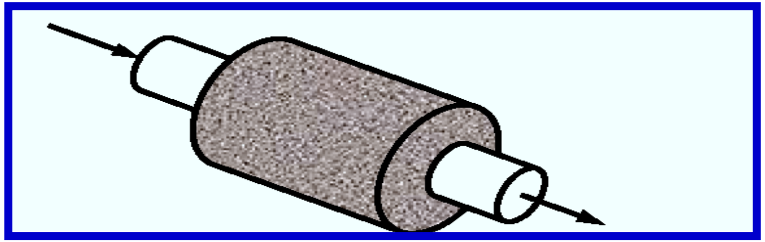
③ Source terms

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



The porous media model incorporates an empirically determined flow resistance in a region of our model defined as “porous”. In essence, the porous media model adds a momentum sink in the governing momentum equations.

◆ Cell Zones – Solid

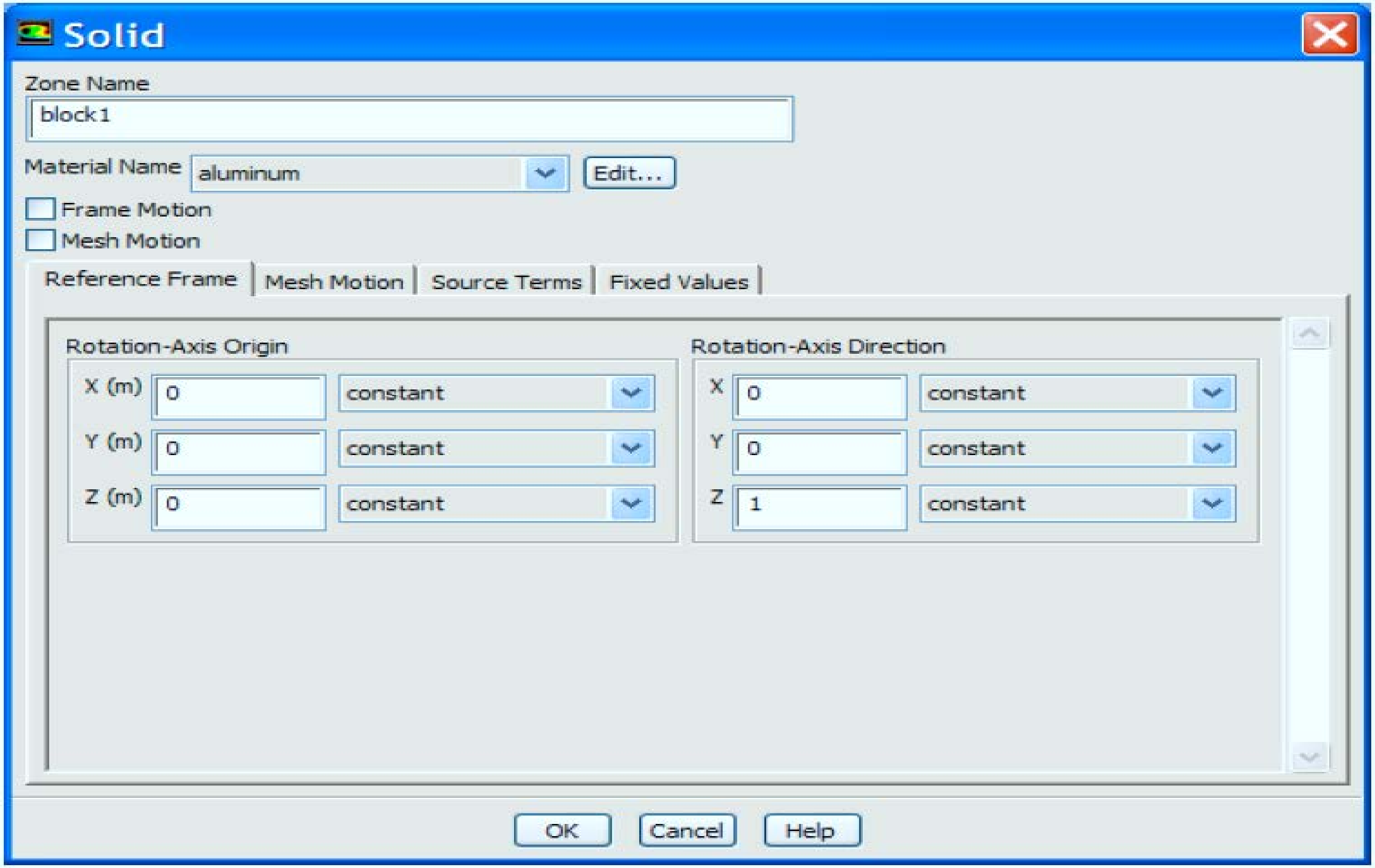
1) A solid zone is a group of cells for which only the energy equation is solved.

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

3) Optional inputs allow us 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

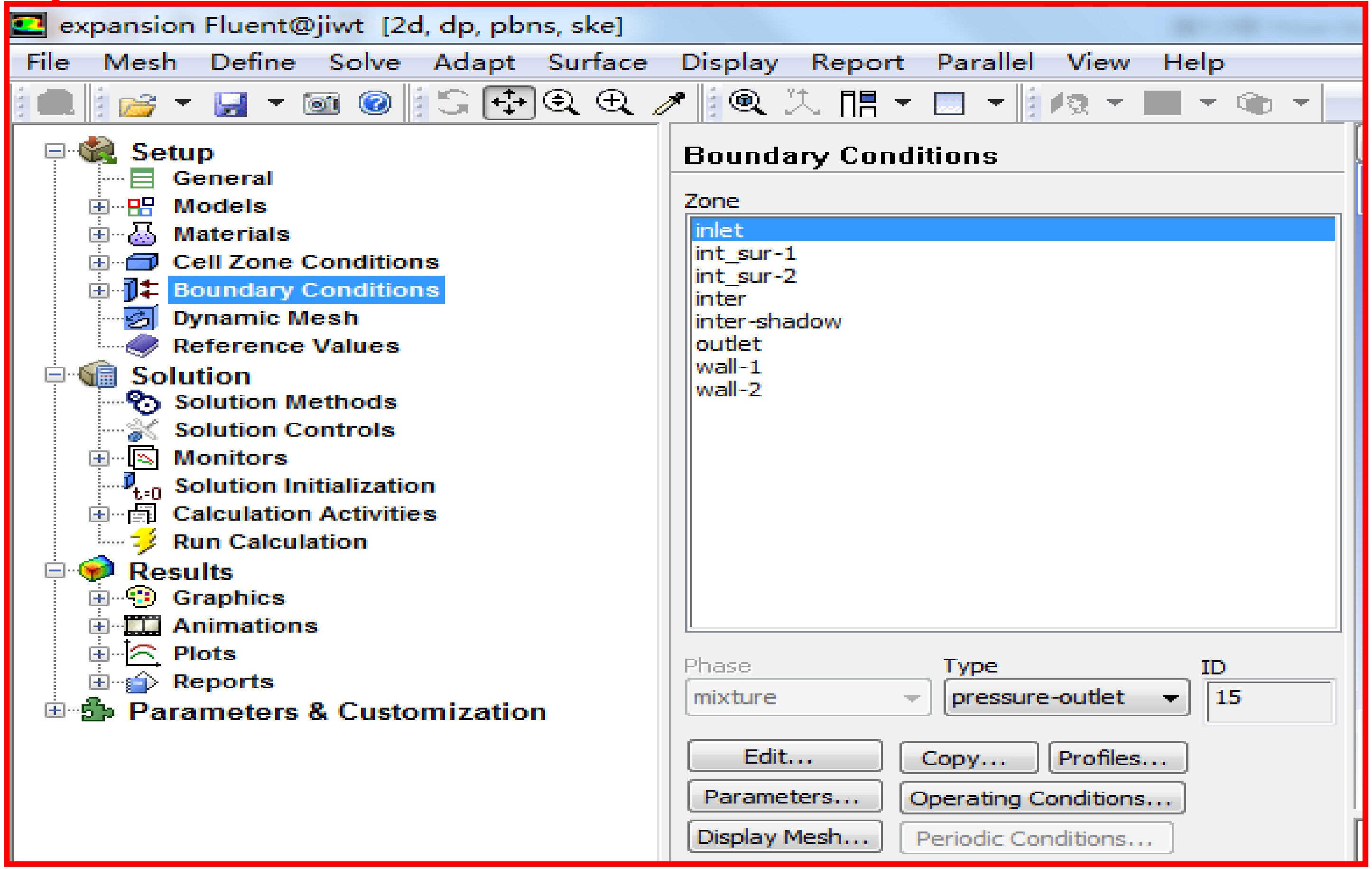

Solid
Zone Name
block1
Material Name
aluminum
Edit...
Frame Motion
Mesh Motion
Reference Frame
Mesh Motion
Source Terms
Fixed Values
Rotation-Axis Origin
X (m)
0
constant
Y (m)
0
constant
Z (m)
0
constant
Rotation-Axis Direction
X
0
constant
Y
0
constant
Z
1
constant
OK
Cancel
Help

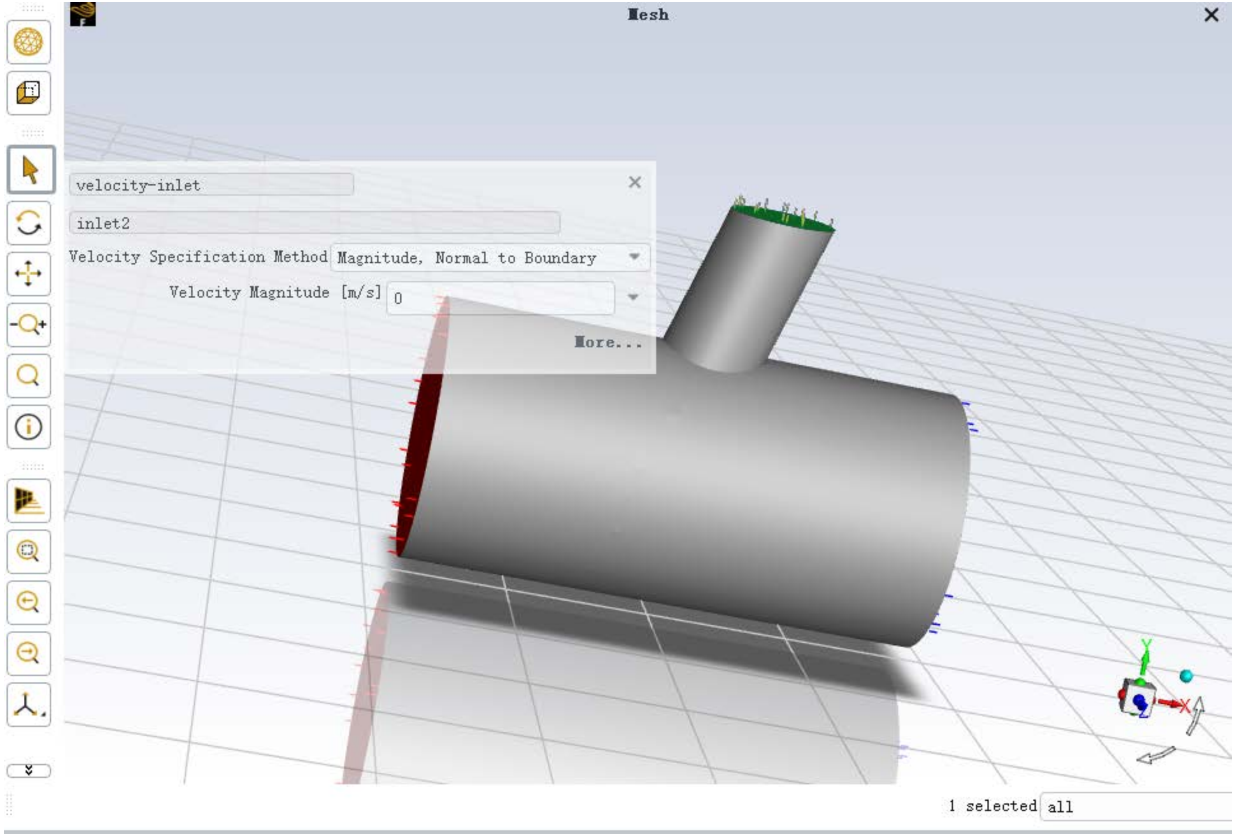
③ The data required at a boundary depends upon the boundary condition type and the physical models employed.

④ Be aware of the information that is required of the boundary condition, and locate the boundaries where the information on the flow variables are known or can be reasonably approximated

- Poorly defined boundary conditions can have a significant impact on the solution

5. Boundary Conditions





It also allows for direct editing of boundary conditions on the geometries

Defining Boundary Conditions

① To define a problem that results in a unique solution, we must specify information on the dependent (flow) variables at the domain boundaries

- Specifying fluxes of mass, momentum, energy, etc. into domain.

② Defining boundary conditions involves:

- Identifying the location of the boundaries (e.g. Inlets, walls, symmetry)
- Supplying information at the boundaries

• General guidelines:

1). If possible, select boundary location and shape such that flow either goes in or out.

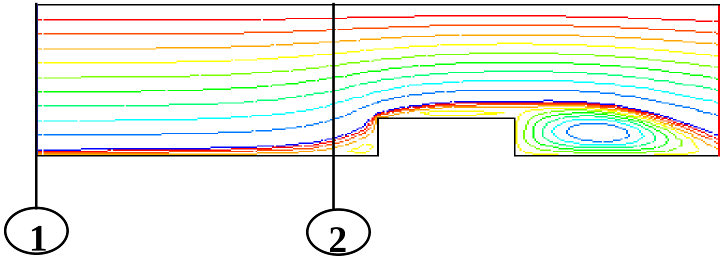
It will typically observe better convergence.

2). Should not observe large gradients in direction normal to boundary.

Indicates incorrect set-up.

3). Minimize grid skewness near the boundary.

Otherwise it would introduce error early in calculation.



Boundary Conditions - Available Types

External Boundaries

1）General

- Pressure Inlet
- Pressure Outlet

2）Incompressible

- Velocity Inlet
- Outflow

3）Compressible

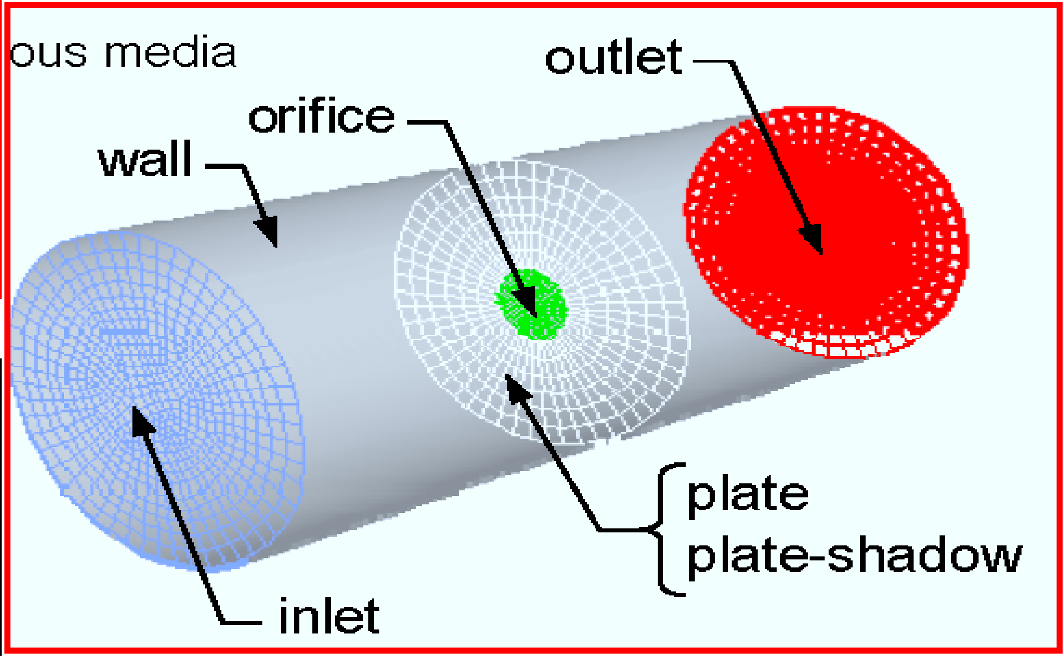
- Mass Flow Inlet
- Pressure Far Field

4）Other

- Wall
- Symmetry
- Axis
- Periodic

- **Internal Boundaries**
  - **Fan**
  - **Interior**
  - **Radiator**
  - **Wall**

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



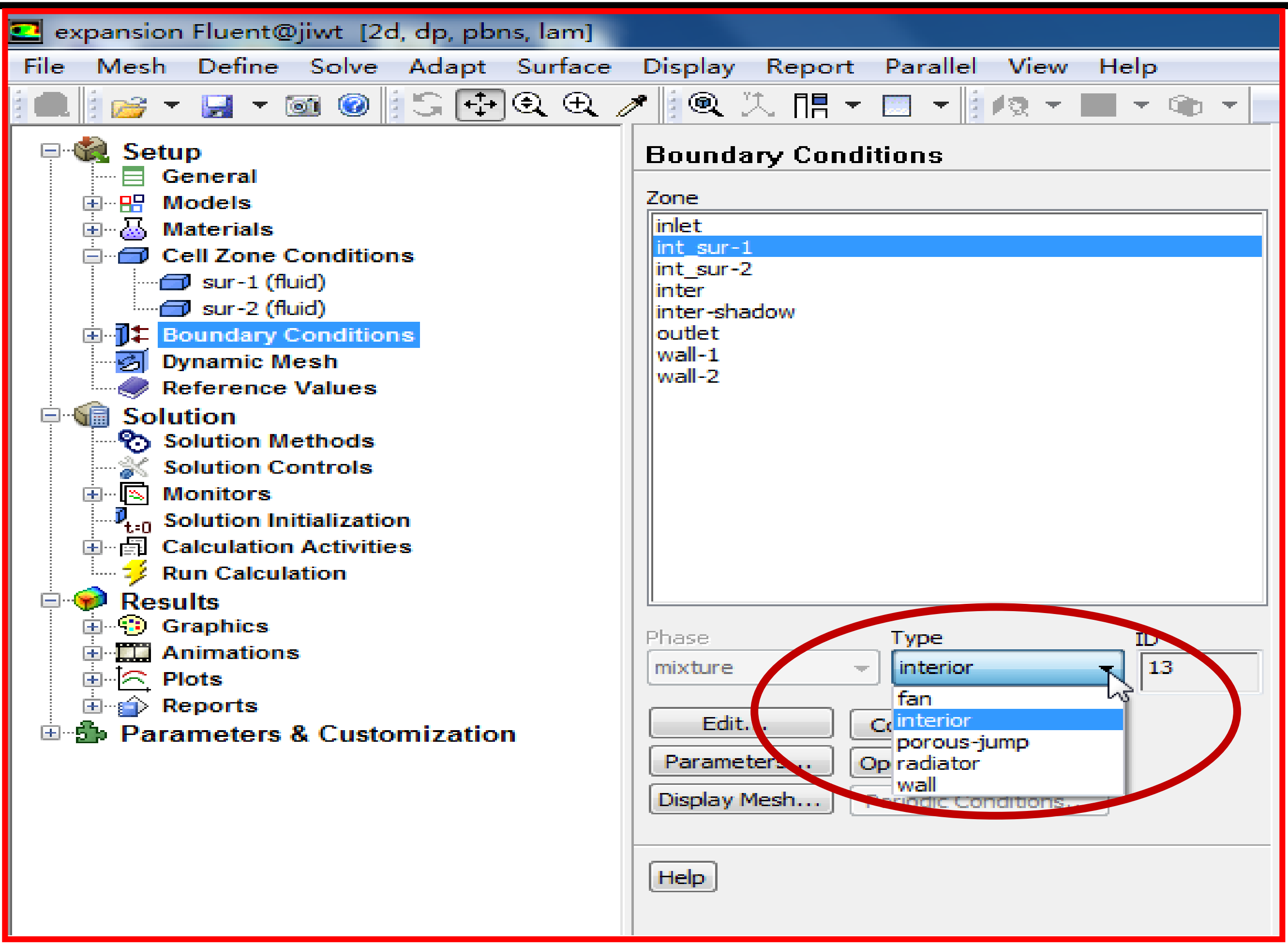
◆ Boundary Conditions – Changing the Types

• **Zones and zone types are initially defined in the pre-processing phase(eg.ICEM).**

• **To change the boundary condition type for a zone:**

① **Choose the zone name in the Zone list.**

② **Select the type we 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

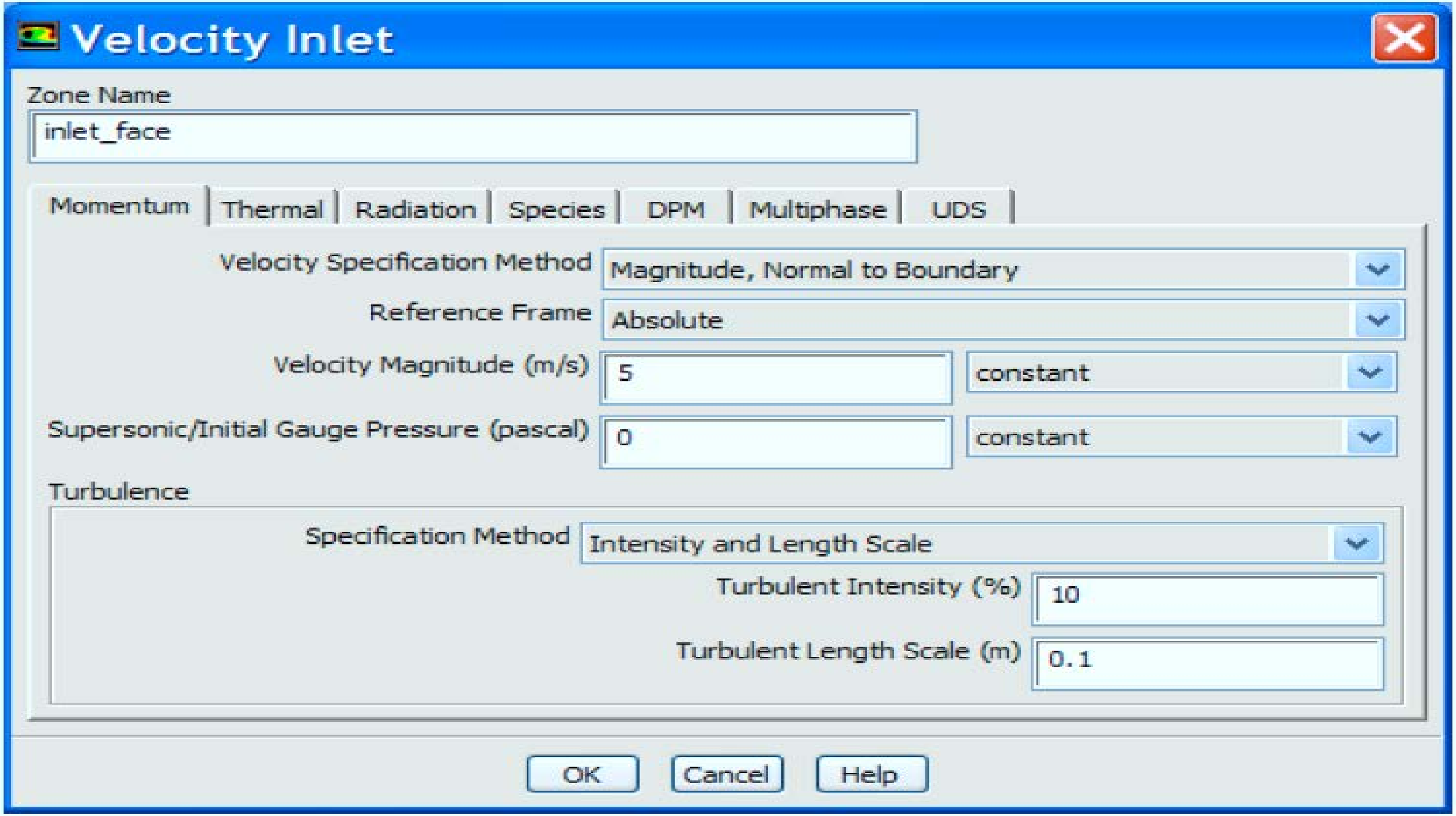
– Magnitude, Normal to Boundary

– 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 is the velocity in the exit.


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

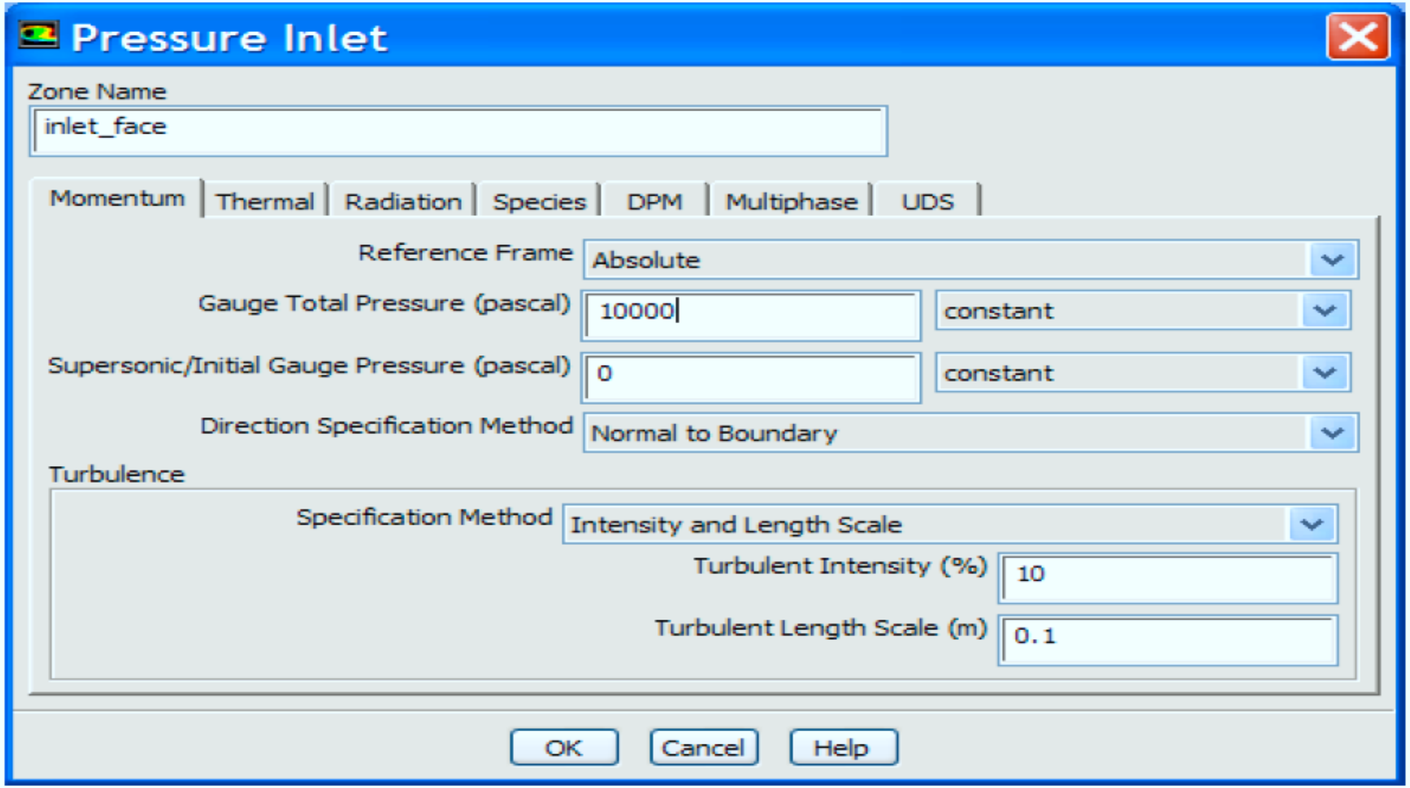
① Gauge Total Pressure

② Supersonic/Initial Gauge Pressure

③ Inlet flow direction

④ Turbulence quantities(if applicable)

⑤ 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.

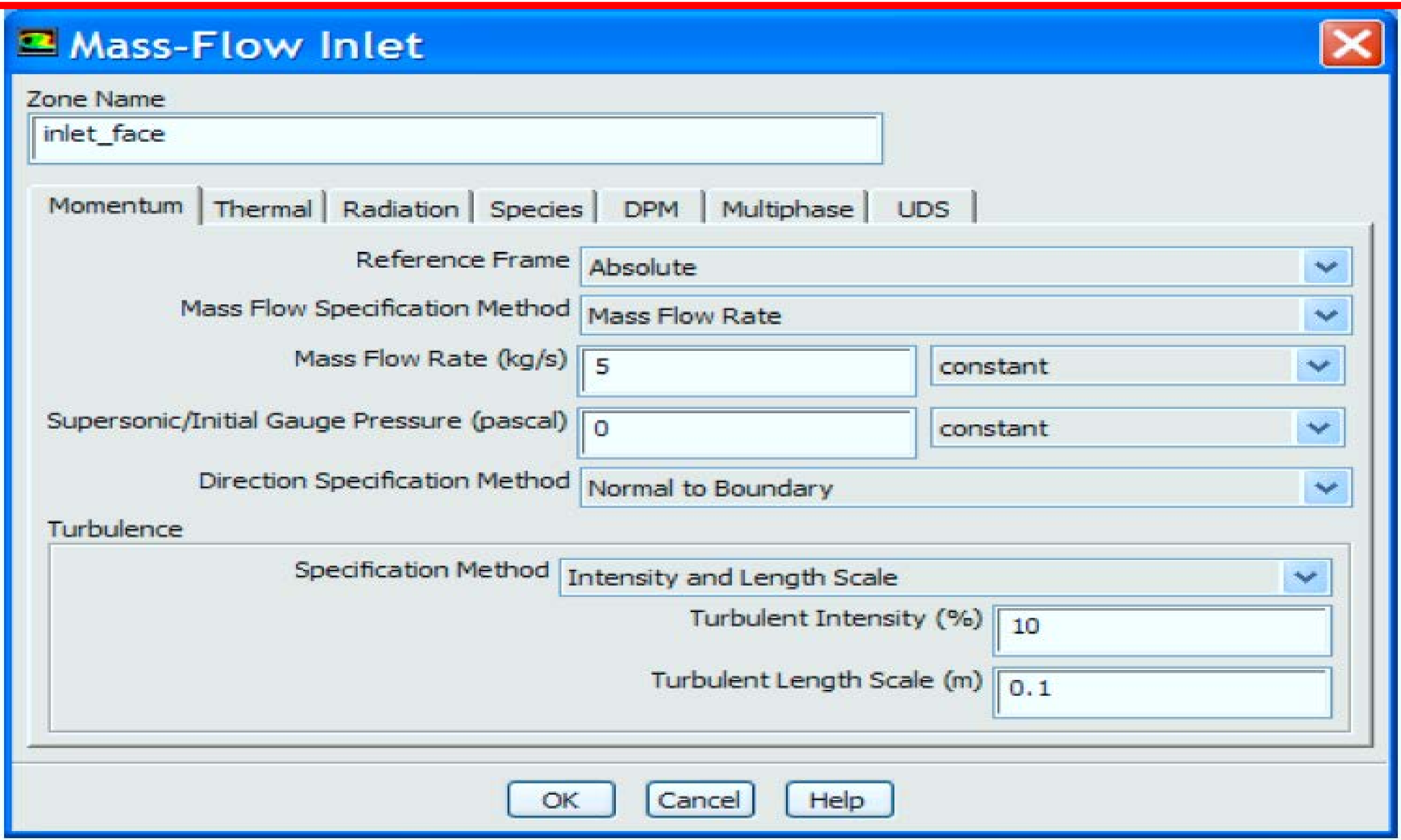
① Total pressure adjusts to accommodate mass flow inputs.

② More difficult to converge than pressure inlet.

2) Required information

① Mass Flow Rate or Mass Flux

② 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

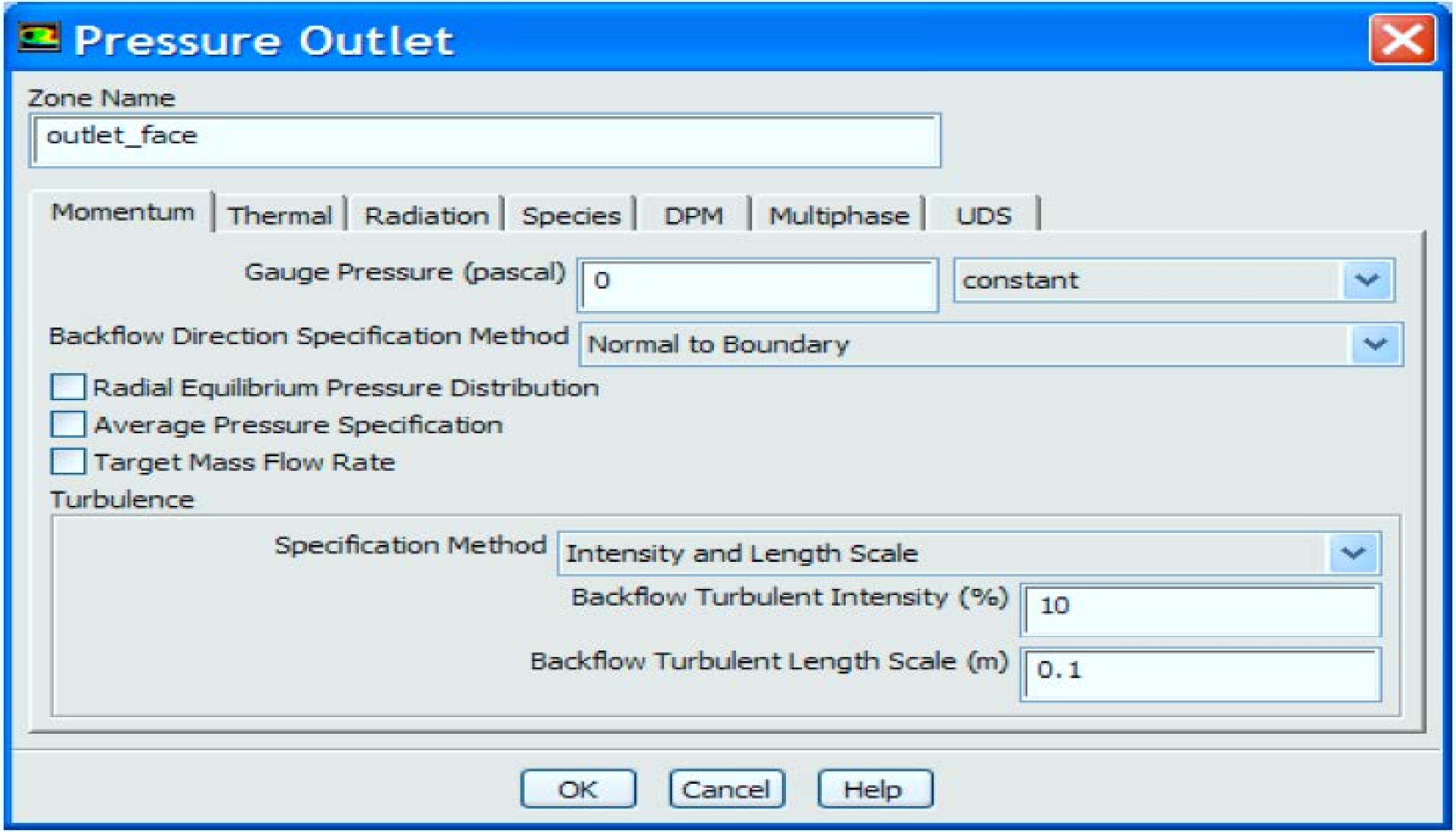
◆ Boundary Conditions - Pressure Outlet

1) Suitable for compressible and incompressible flows.

2) Required information

① Gauge Pressure (static)–static pressure of the environment into which the flow exits.

② 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 - 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

① 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.

② Used to implement various physical models including:

– Fans

– Radiators

③ Preferable over porous media for its better convergence behavior.

– Interior walls

④ The "interior" type of internal face zone does not require any input.

◆ Boundary Conditions - Outflow

① No pressure or velocity information is required.

– Data at exit plane is extrapolated from interior.

– Mass balance correction is applied at boundary.

② Flow exiting outflow boundary exhibits zero normal diffusive flux for all flow variables.

– Appropriate where the exit flow is fully developed.

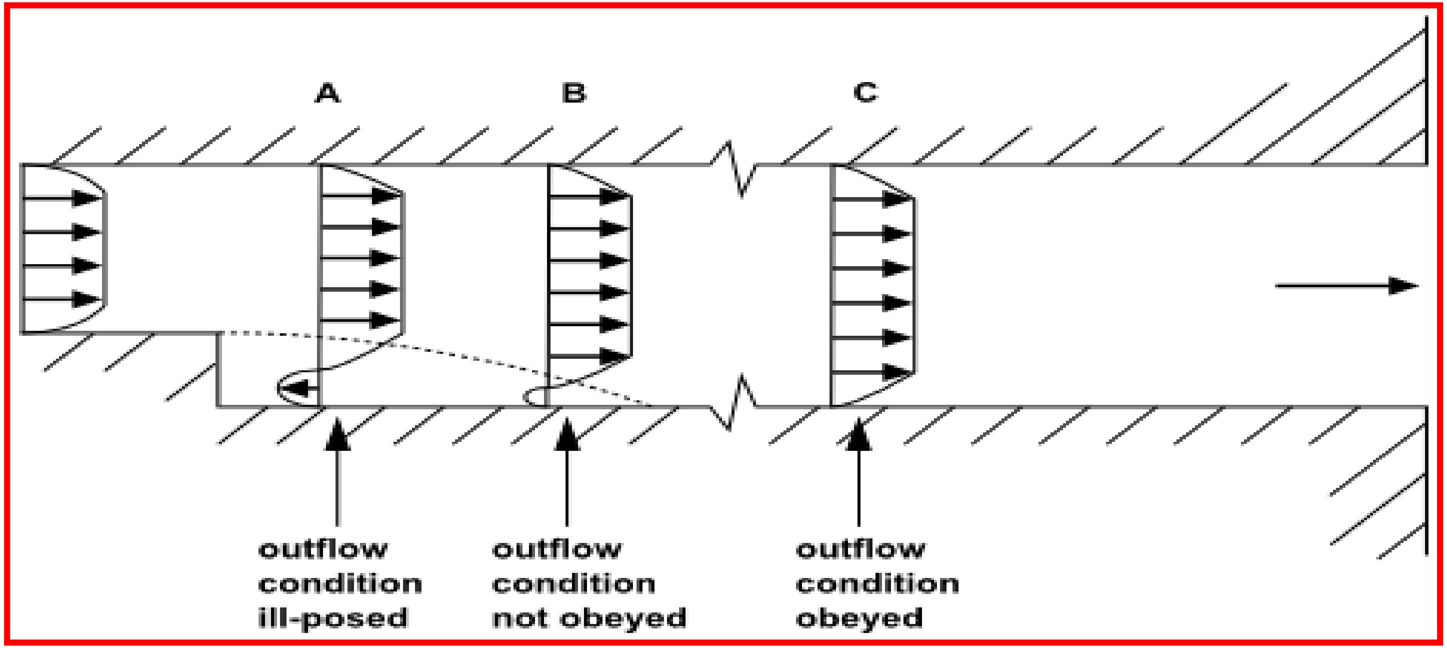
③ The outflow boundary is intended for use with incompressible flows.

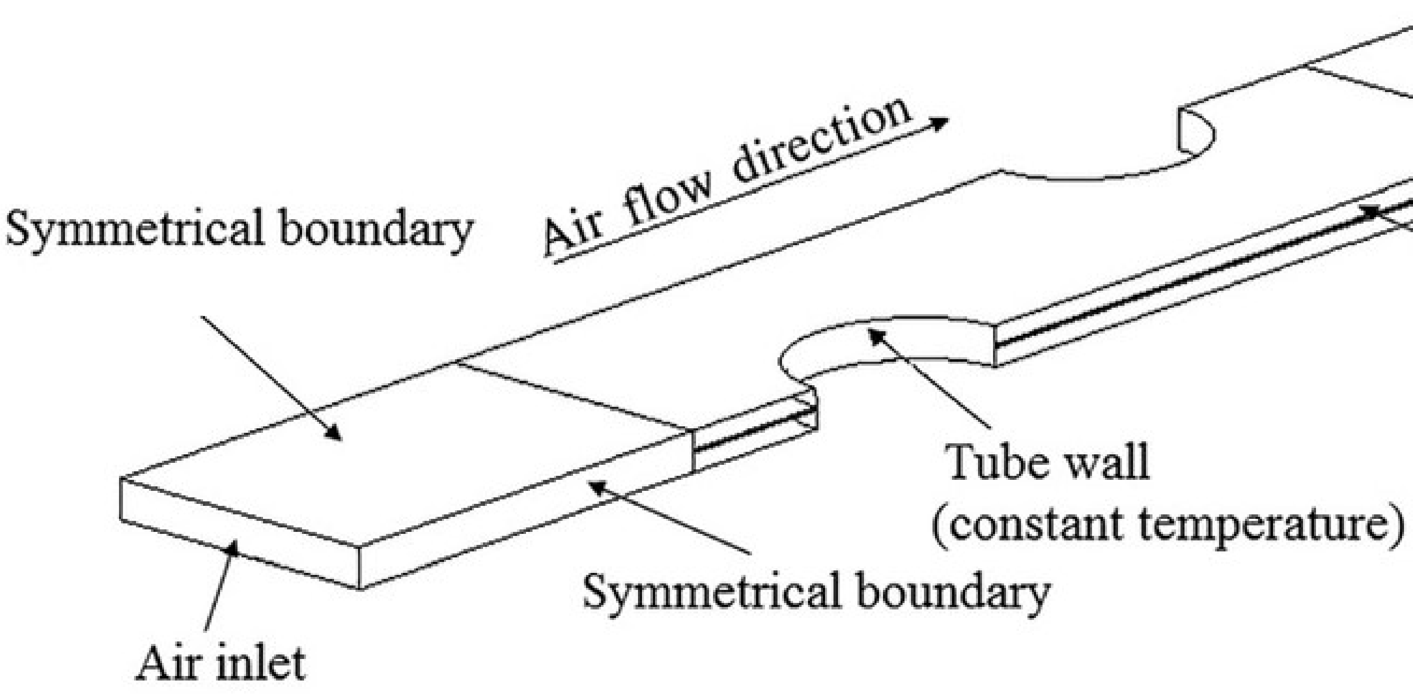
– Cannot be used with a pressure inlet boundary (must use velocity-inlet).

④ **Cannot be used for unsteady flows with variable density.**

⑤ **Convergence rate is poor when backflow occurs during iterations.**

Cannot be used if backflow is expected in the final solution.





Fin and tube heat exchanger simulation

◆ Wall Boundary Conditions

- **Five thermal conditions**

① **Heat Flux**

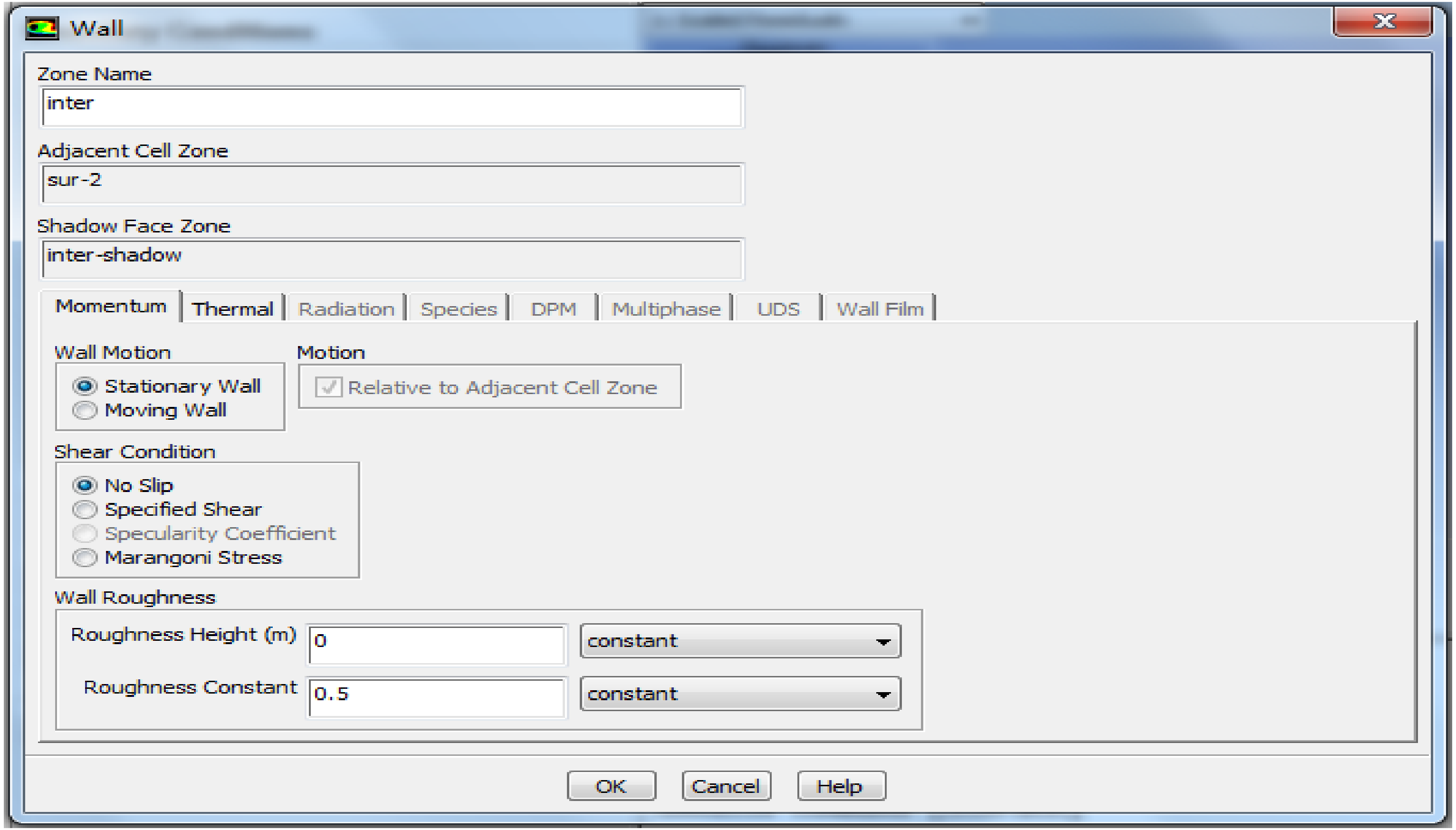
② **Temperature**

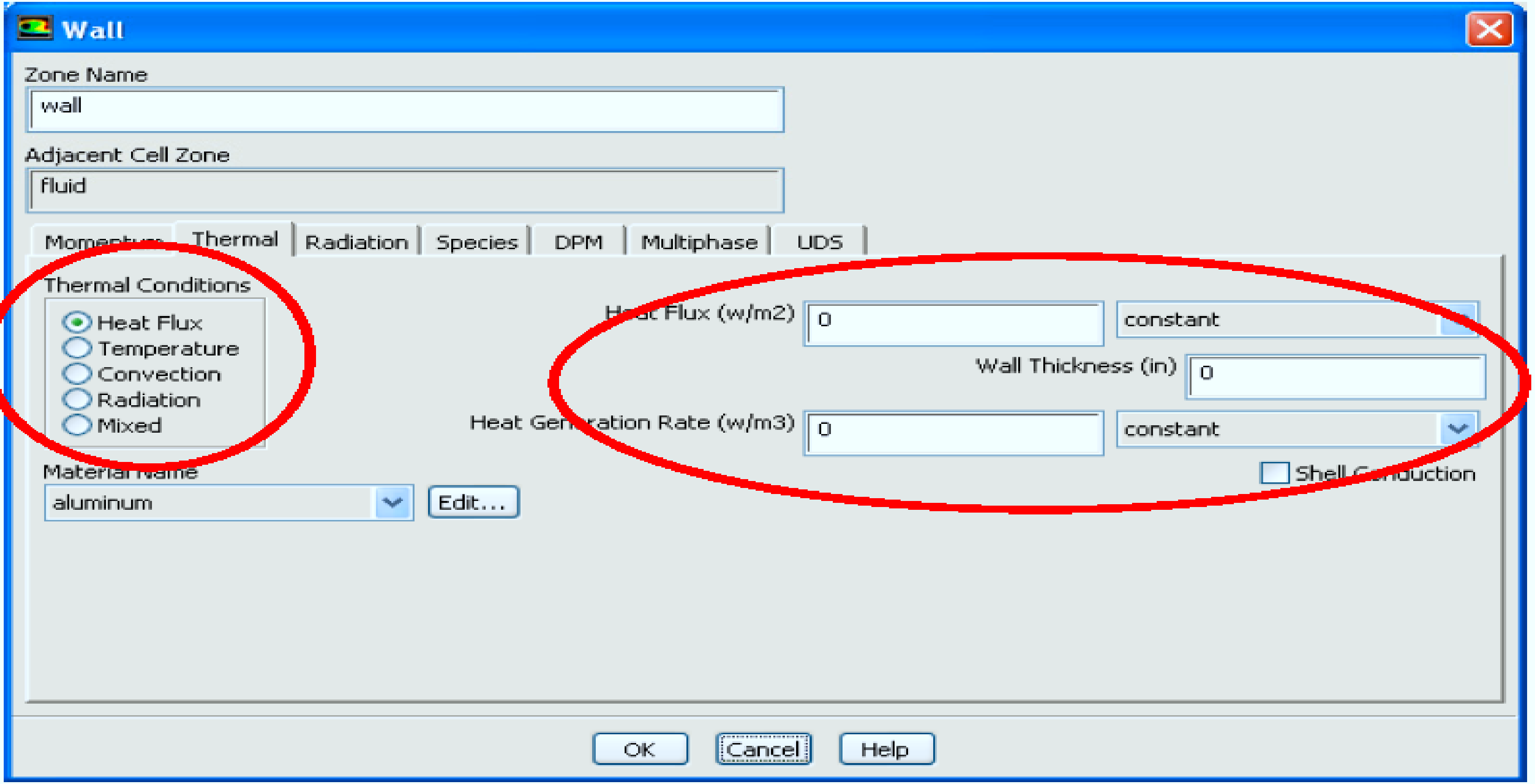
③ **Convection–simulates an external convection environment which is not modeled (user-prescribed heat transfer coefficient).**

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

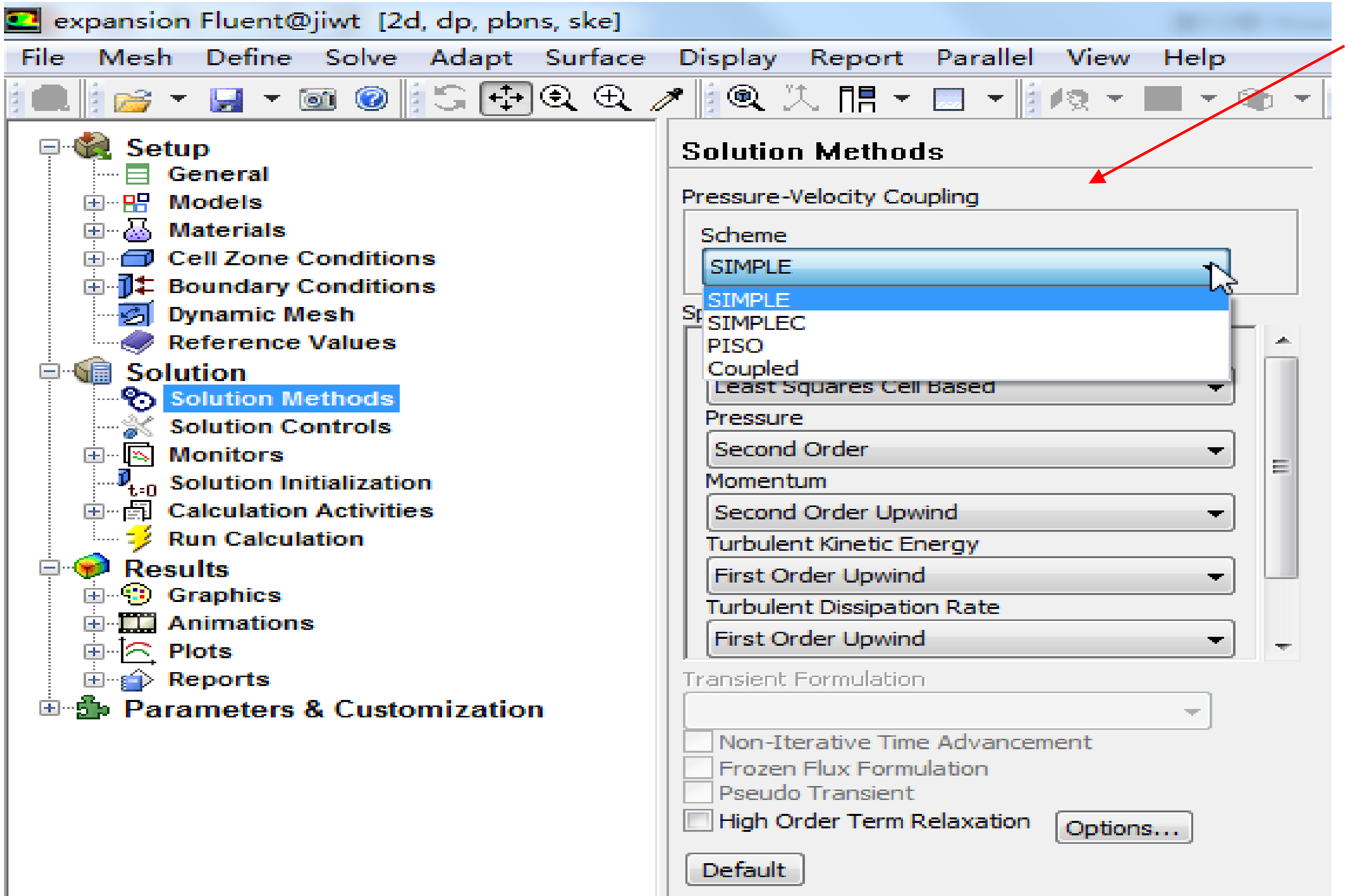
⑤ **Mixed– Combination of Convection and Radiation boundary conditions.**

Momentum 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

6. Solution Methods



SIMPLE：Semi-Implicit Method for Pressure Linked Equations

Chapters 5. Discretized Schemes of Diffusion and Convection Equation；第5章：对流扩散方程的离散格式

Chapter 6：Primitive Variable Methods for Elliptic Flow and Heat Transfer第6章：求解椭圆型流动与换热问题的原始变量法

Solution Procedure Overview

① **Select the solution parameters**

– Choosing the Solver
– Discretization Schemes

} Rather important for simulation result

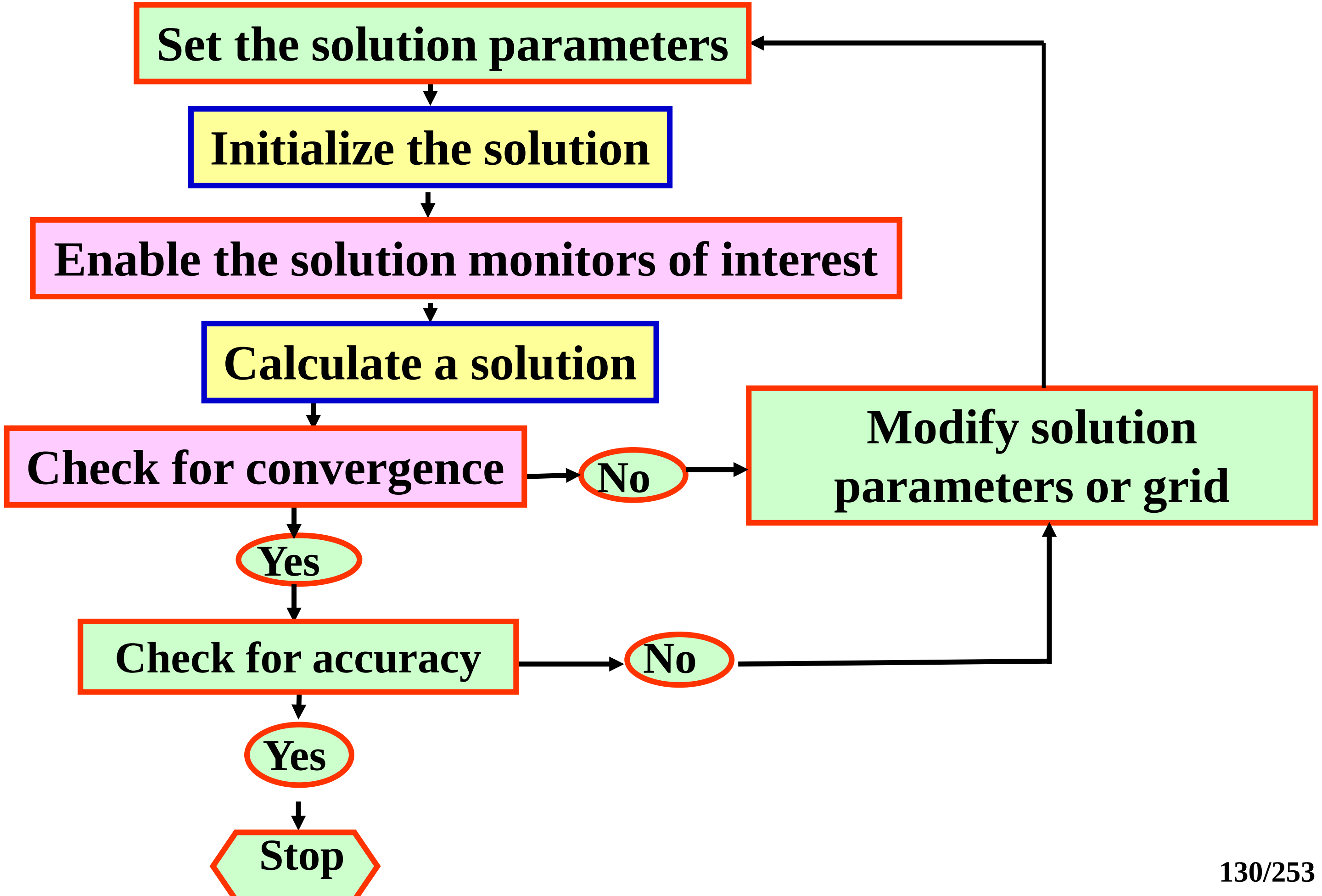
② **Initialization**

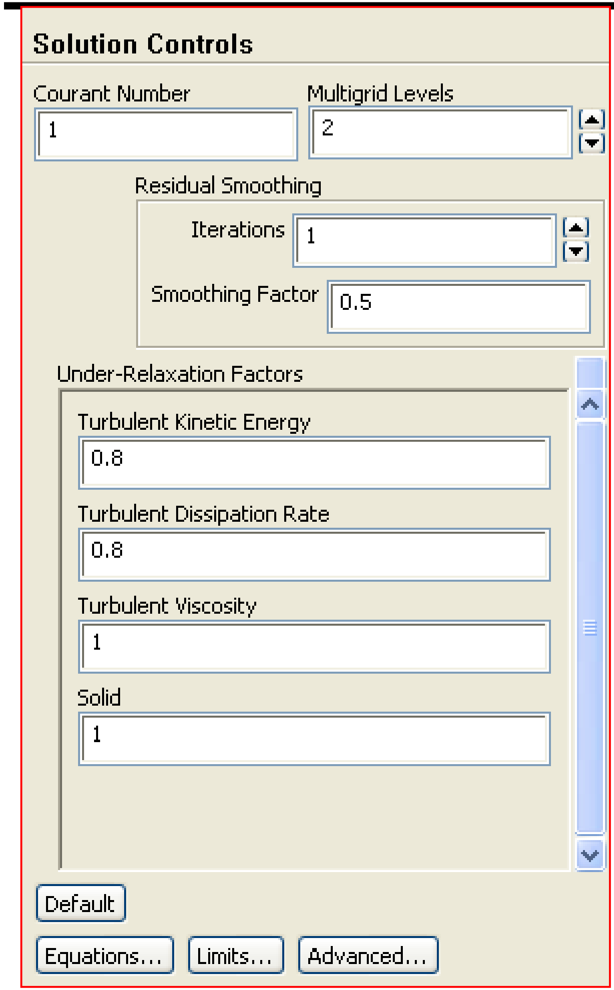
③ **Check the convergence behavior**

- Monitoring Convergence Stability
  - Setting Under-relaxation factor
  - Setting Courant number
- Accelerating Convergence

④ **Evaluation the accuracy of computation result**

- Grid Independence
- Adaption


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



The Solution Controls for Density-Based Explicit Formulation

Courant number is a measure of how fast information traverses *(u)* a computational grid cell (Δx) in a given time-step (Δt).

- **When we select Explicit from the Formulation drop-down list, in the Solution Methods task page, ANSYS FLUENT will automatically set the Courant Number to 1;**
- **When we select Implicit from the Formulation drop-down list, the Courant Number will be changed to 5 automatically.**

Solver Settings

- **By modifying the solver settings we can improve both:**

① **The convergence rate of the simulation. (Chapter 6 求解椭圆形流动)**

② **The accuracy of the computed result.(Chapter 5 对流-扩散方程的离散格式)**

To Consider:

a. The choice of solver
b. Discretization schemes
c. Checking convergence
d. Assessing accuracy

◆ Available Solvers

① There are two kinds of solvers available in FLUENT:

– Pressure based
– Density based

② The pressure-based solvers take momentum and pressure (or pressure correction) as the primary variables(such as SIMPLE Algorithm)

③ 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).

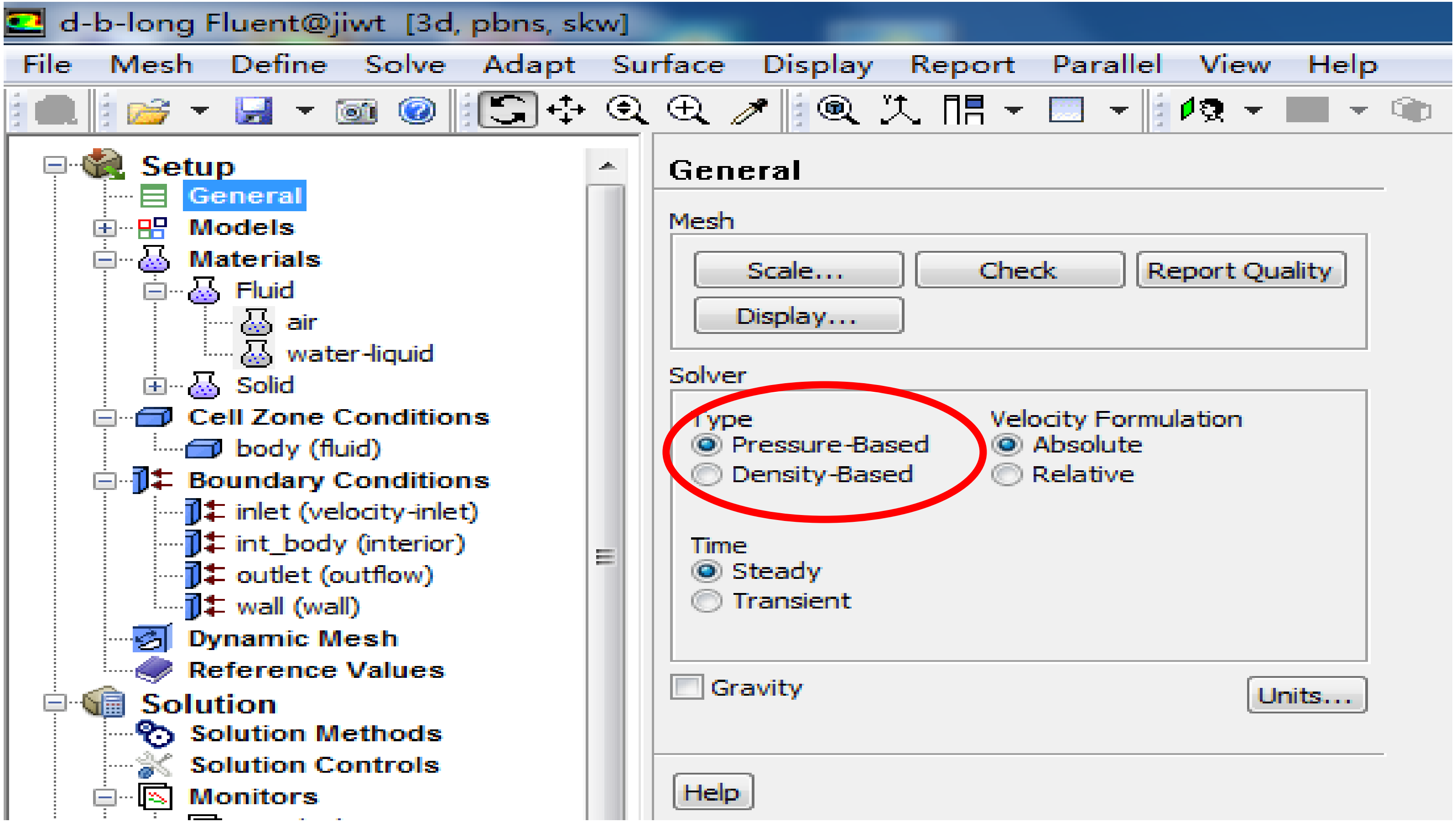
④ Density-Based Coupled Solver

– 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

① The pressure-based solver(PBS) 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.

② The pressure-based coupled solver(PBCS) is applicable for most single phase flows, and yields superior performance to the standard pressure-based solver.

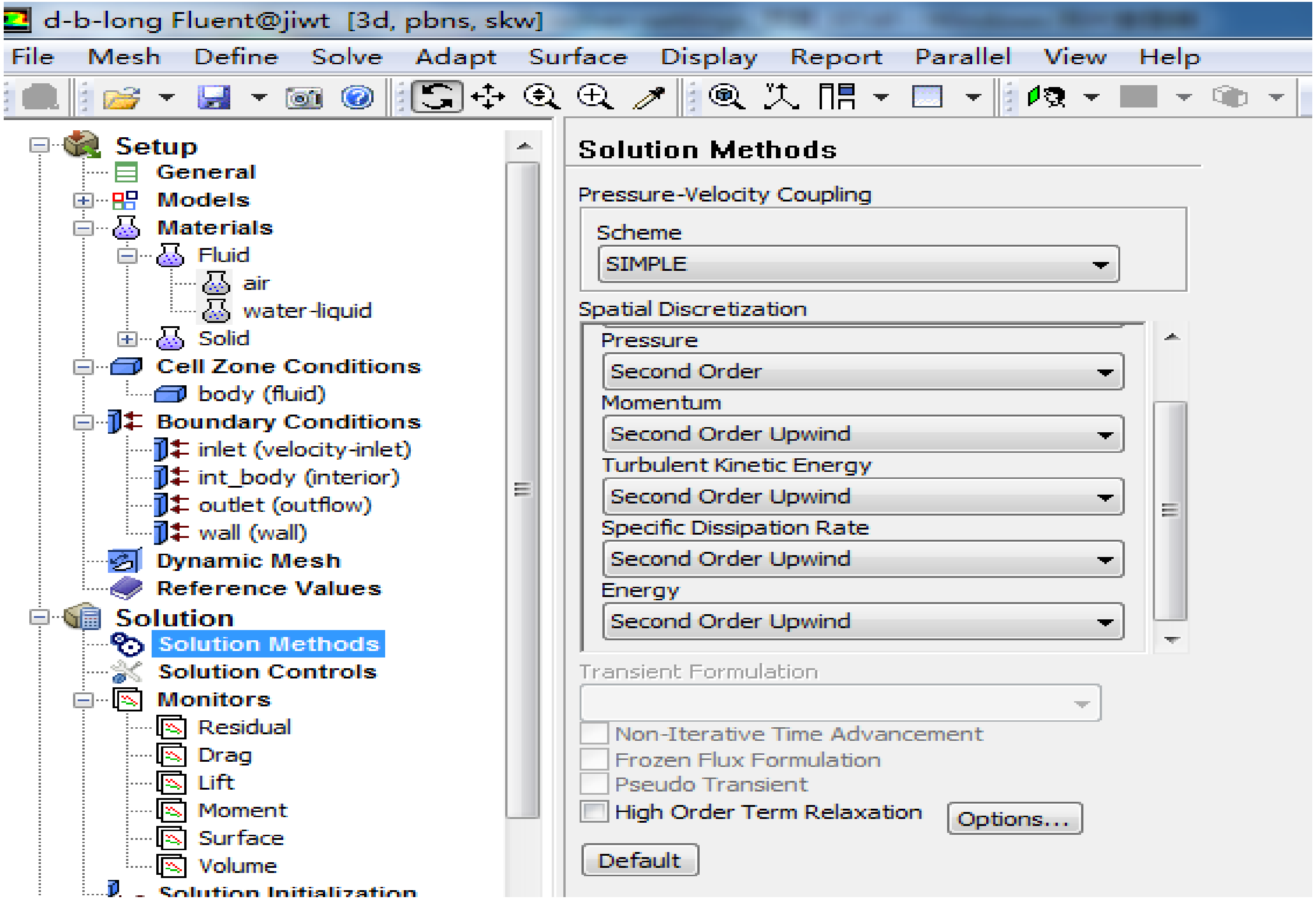
- Requires 1.5–2 times more memory than the segregated solver.

③ **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).**

Note: the pressure-based solvers are 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
Residual
Drag
Lift
Moment
Surface
Volume
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 convection term(Chapter 5:离散格式):

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

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

③ **Second-Order Upwind** –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 discretization 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.

• Higher order schemes will be more accurate. They will also be less stable and will increase computational time.

• It is recommended to always start calculations with first order upwind and after 100 iterations or so to switch over to second order upwind. This provides a good combination of stability and accuracy.

• The central differencing scheme should only be used for transient calculations involving the large eddy simulation (LES) turbulence models in combination with grids that are fine enough that the Peclet number is always less than one.

• It is recommended to only use the power law or QUICK schemes if it is known that those are somehow especially suitable for the particular problem being studied.

Interpolation Methods (Gradients)(插值方法)

① Gradients of solution variables are required in order to evaluate diffusive fluxes, velocity derivatives, and for higher-order discretization schemes.

② 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/computationally 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

④ **Second-Order:** Use for compressible flows; not to be used with porous media, jump, fans, etc. or VOF/Mixture multiphase models

⑤ **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.

① Semi-Implicit Method for Pressure-Linked Equations (SIMPLE)

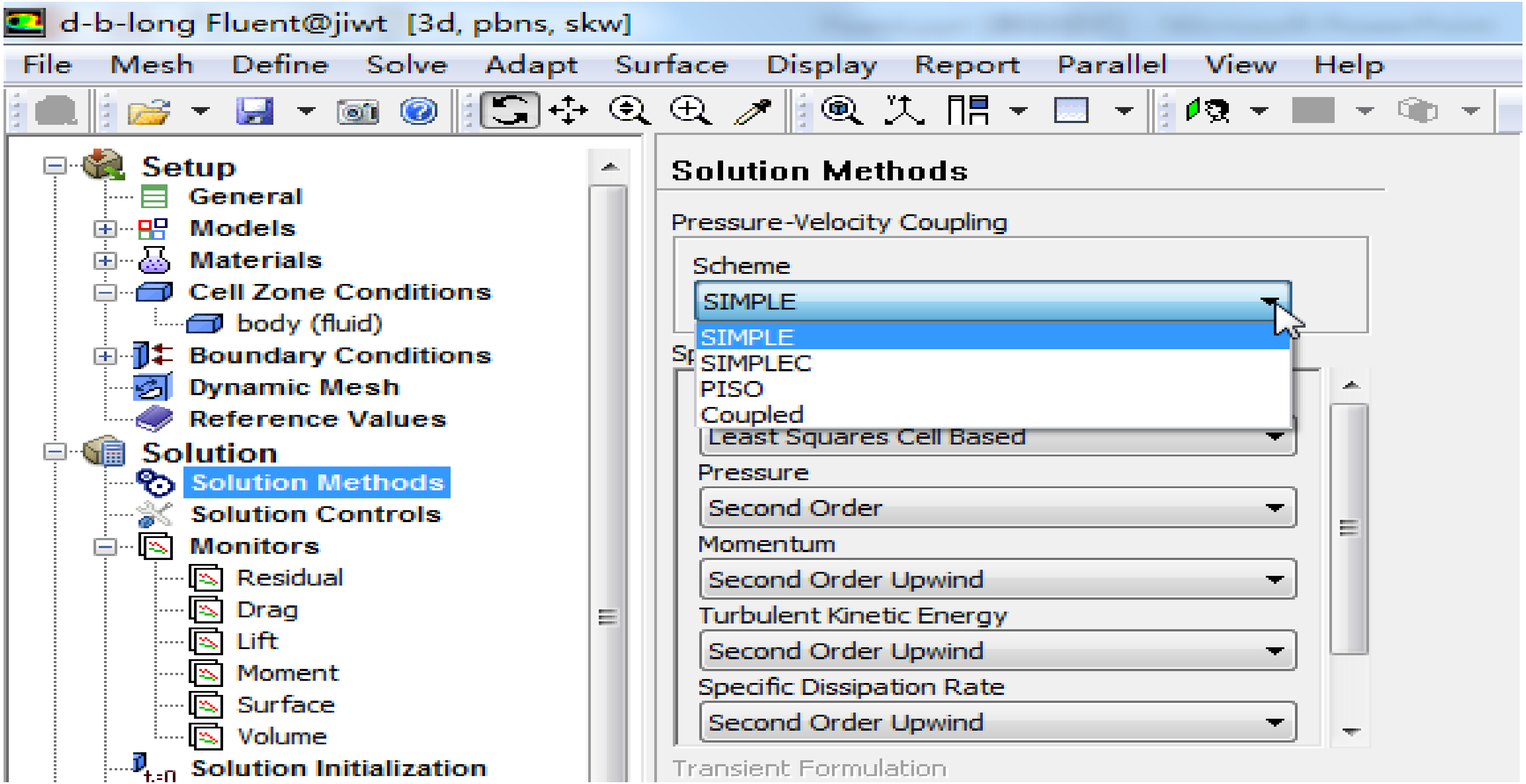
The default scheme, robust

② SIMPLE-Consistent (SIMPLEC)

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

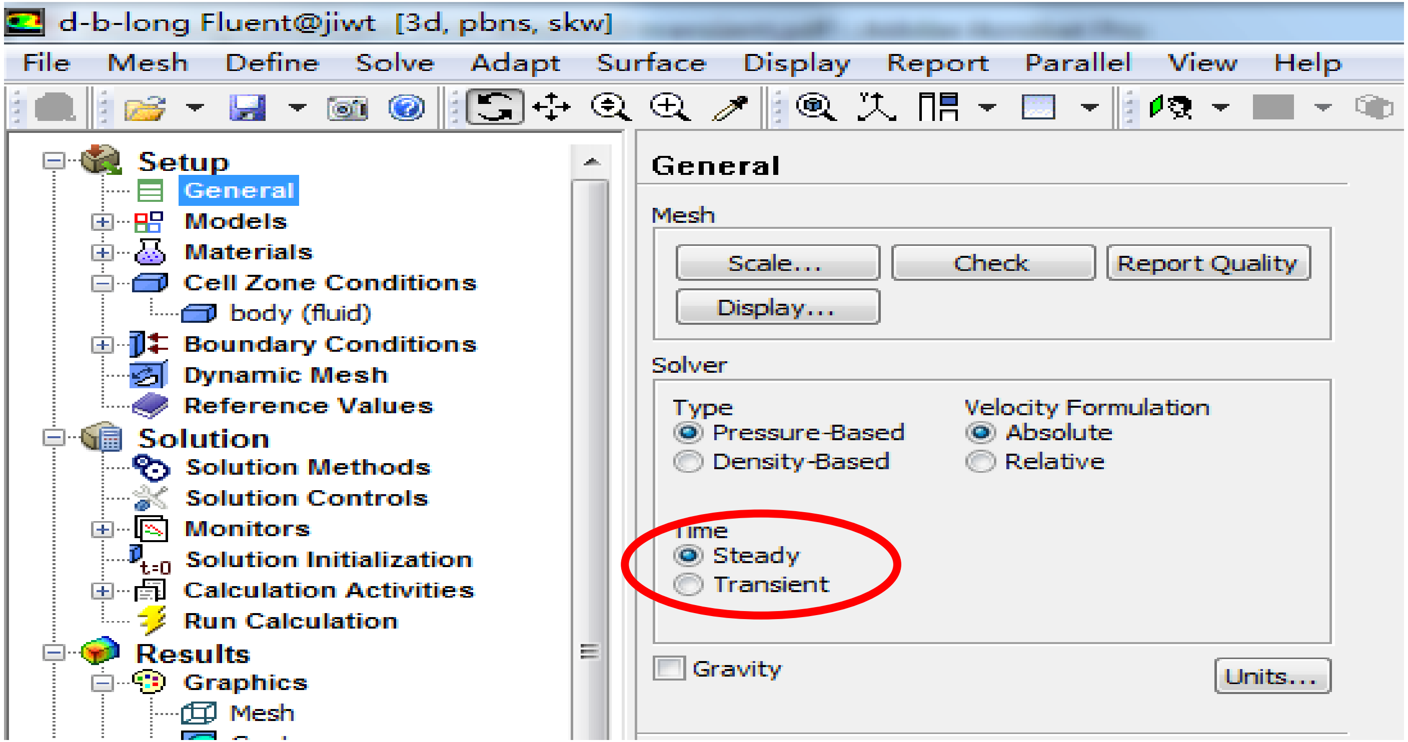
③ Pressure-Implicit with Splitting of Operators (PISO)

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(非稳态问题)

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



① **Before performing iterations, we will need to set some additional controls.**

- **Solver settings**
- **Animations**
- **Data export/Autosave options**

② **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:**

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

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 A good way to judge the choice of Δt is to observe the number of iterations. FLUENT needs to converge at each time step. The ideal number of iterations per time step is 5–10. If FLUENT needs substantially more, the time step is too large. If FLUENT needs only a few iterations per time step, it should be increased.

② It is often wise to choose a conservatively small for the first 5–10 time steps. Then gradually increased it as the calculation proceeds.

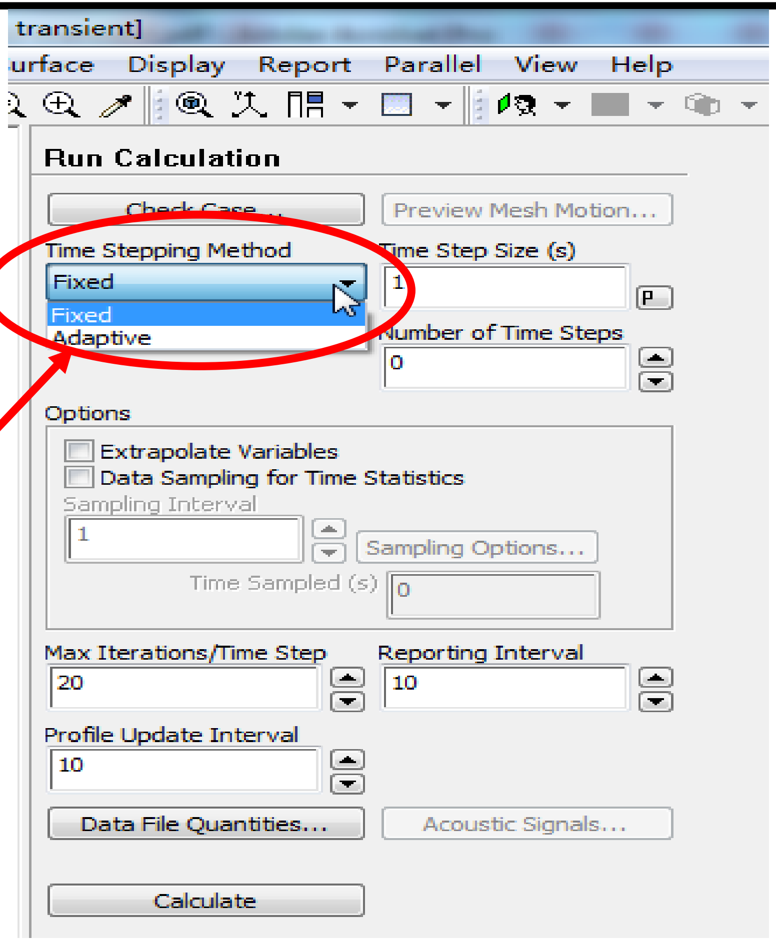
③ To iterate without advancing in time, specify zero time steps. This will instruct the solver to converge the current time step only.

④ The PISO scheme may aid in accelerating convergence for many transient flows form.

Transient Flow Modeling Options

- **Adaptive Time Stepping**

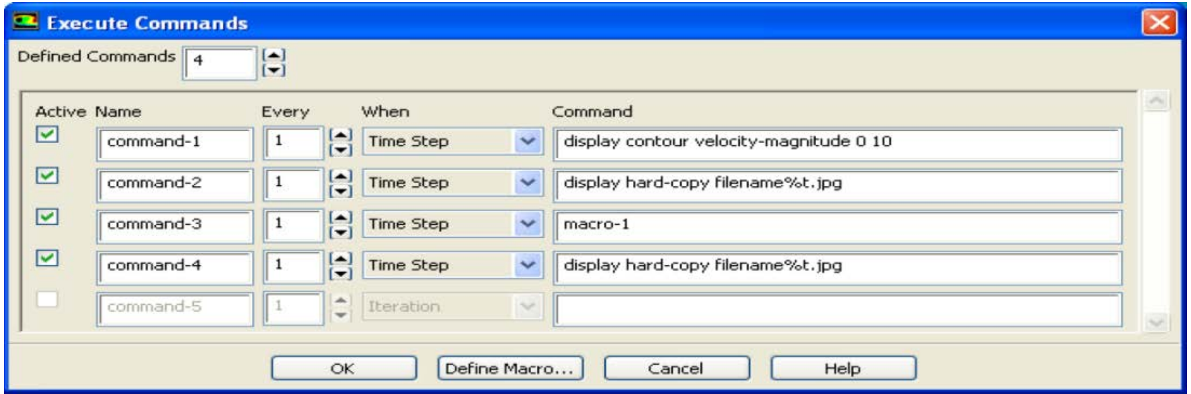
Automatically adjusts time-step size based on local truncation error analysis



Creating Animations

This approach is very useful in creating high-quality animations of CFD results.

- **A command is defined which generates an animation frame (contour plot, vector plot, etc.) and then writes that frame to a hard copy file.**
- **Third-party software can then be used to link the hard copy files into an animation file(AVI, MPG, GIF, etc.)**

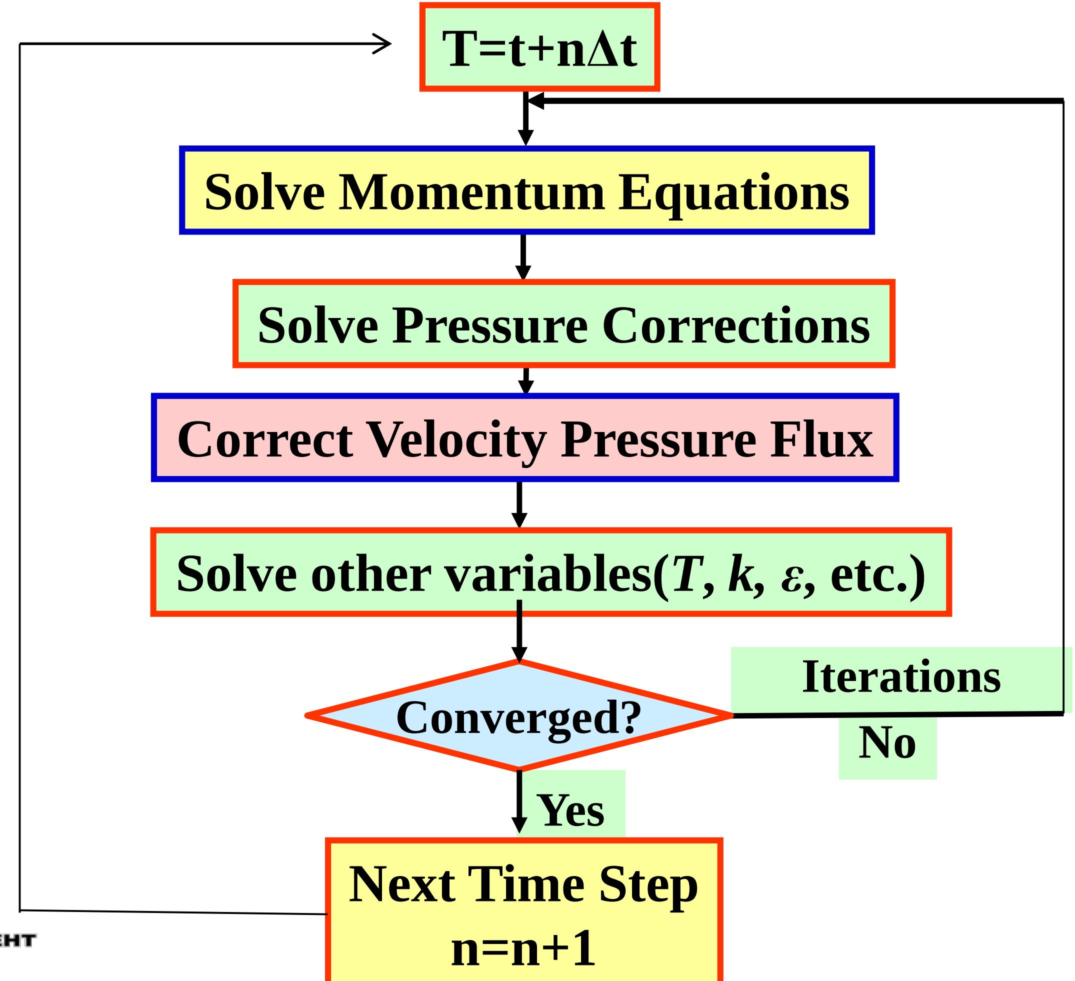


◆ Performing Iterations

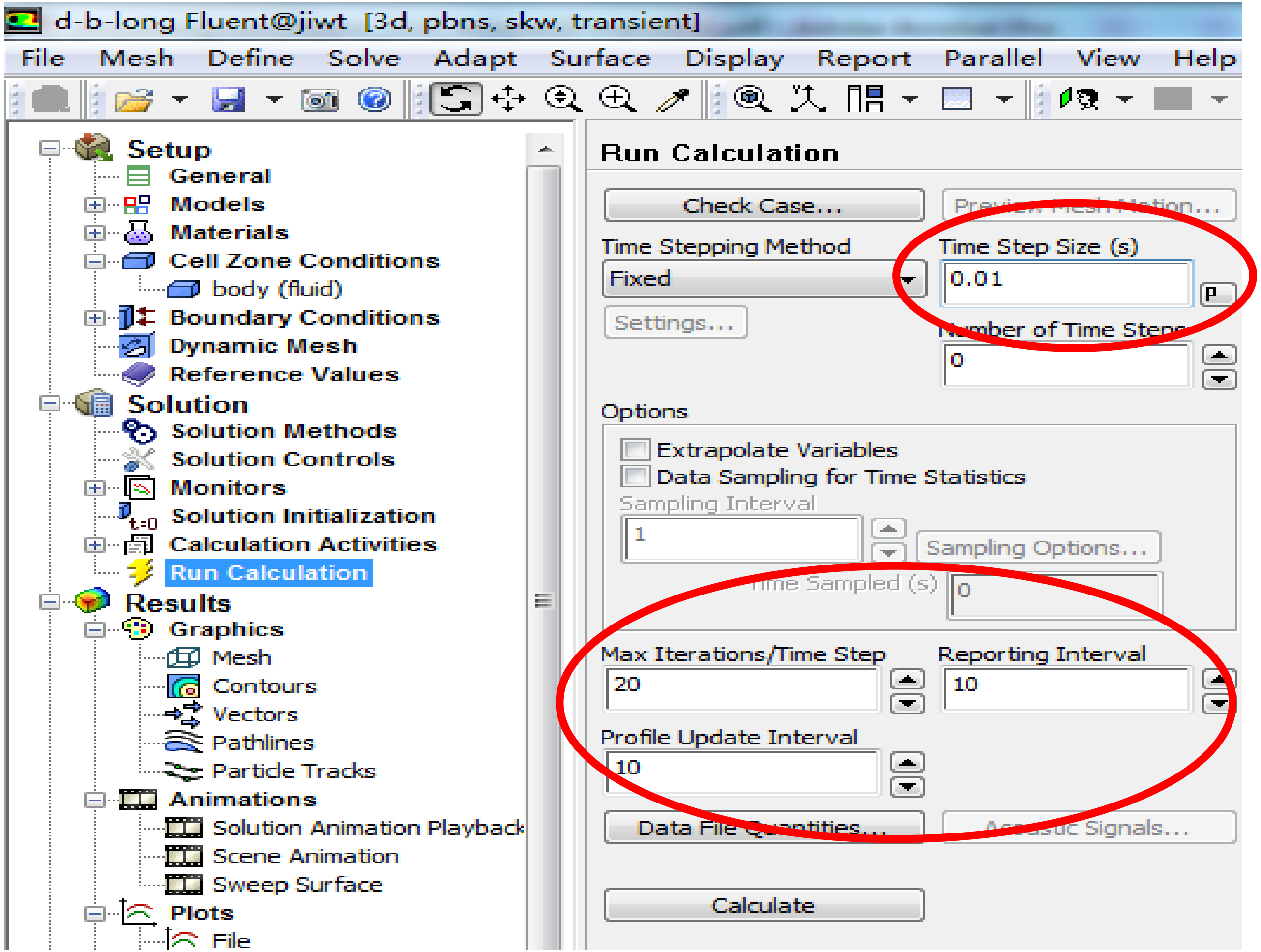
① The most common time advancement scheme is the iterative scheme.

- 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.

② 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.



Overview of the Iterative Time Advancement Solution Method for the Segregate Solver

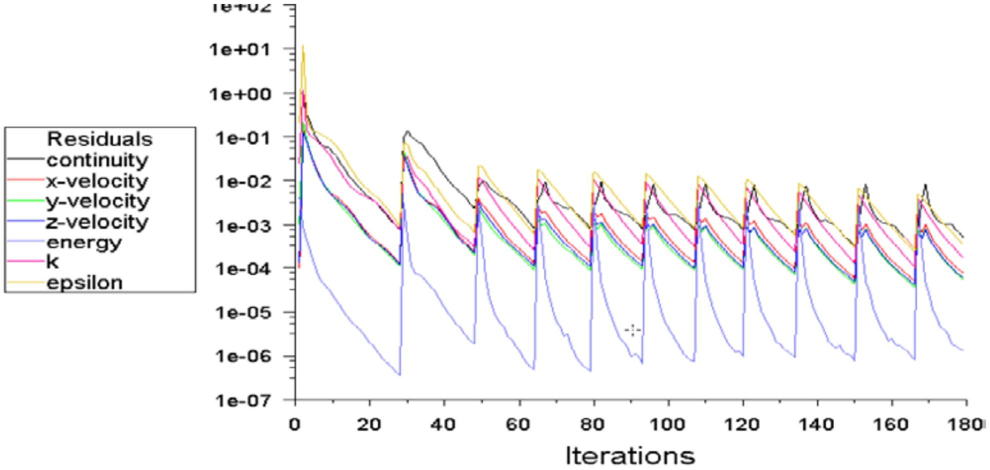

d-b-long Fluent@jiwt [3d, pbns, skw, transient]
File Mesh Define Solve Adapt Surface Display Report Parallel View Help
Setup
General
Models
Materials
Cell Zone Conditions
body (fluid)
Boundary Conditions
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
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
Time Step Size (s)
0.01
Settings...
0
Options
Extrapolate Variables
Data Sampling for Time Statistics
Sampling Interval
1
Sampling Options...
0
Max Iterations/Time Step
20
Reporting Interval
10
Profile Update Interval
10
Calculate

◆Convergence Behavior

① **Residual plots for transient simulations are not always indicative of a converged solution.**

② **We 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

① Use PISO scheme for Pressure-Velocity Coupling-this scheme provides faster convergence for transient flows than the standard SIMPLE approach.

② Select the time step size so that the solution converges three orders of magnitude for each time step(convergence behavior is also problem-specific).

③ **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.**

④ Remember that accurate initial conditions are just as important as boundary conditions for transient problems-initial condition should always be physically realistic!

⑤ Configure any animations we wish to see before running the calculations.

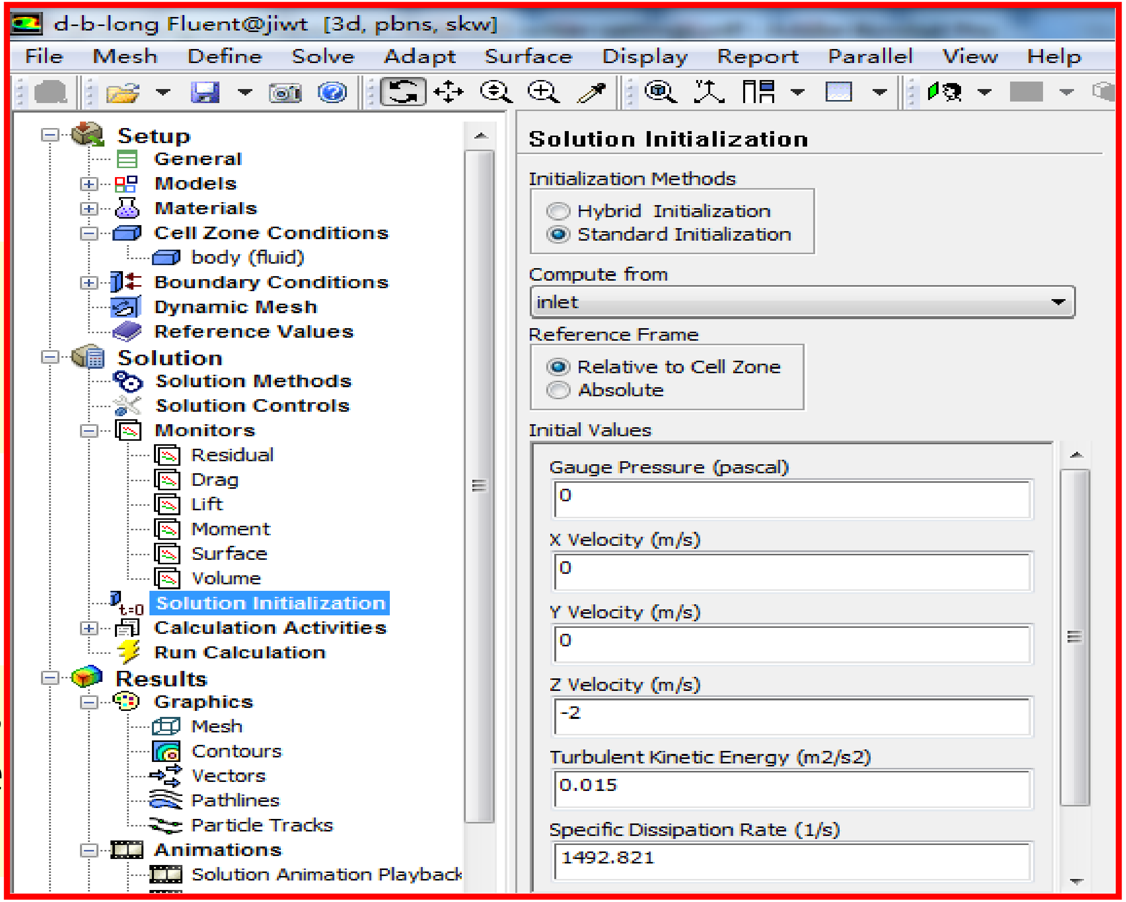
7.Solution Initialization

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.

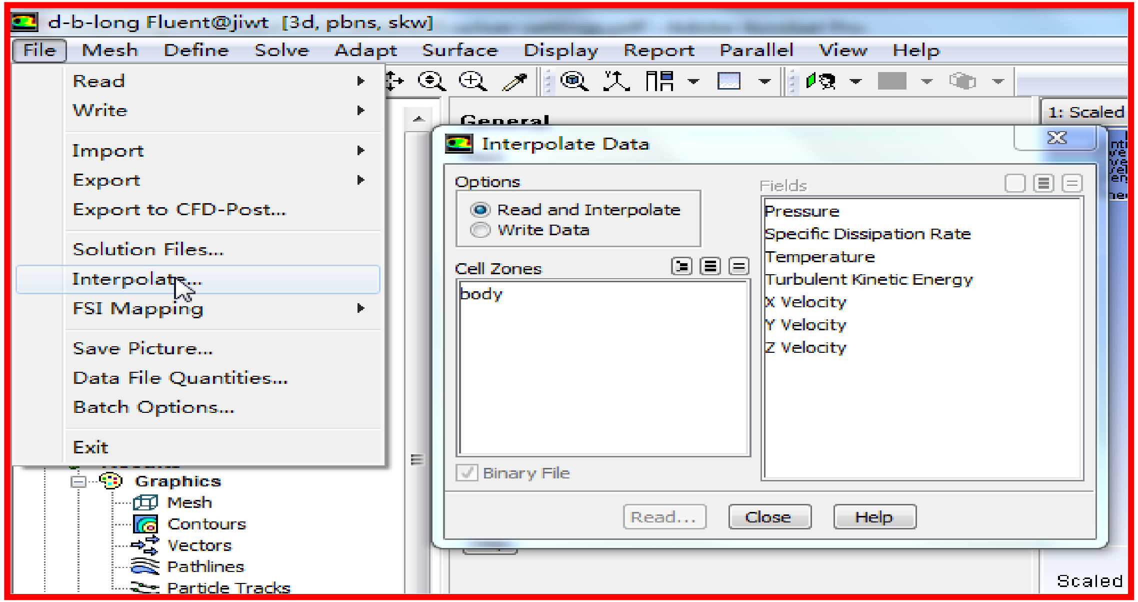
3)Setting this value is called 'Initialization'.

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

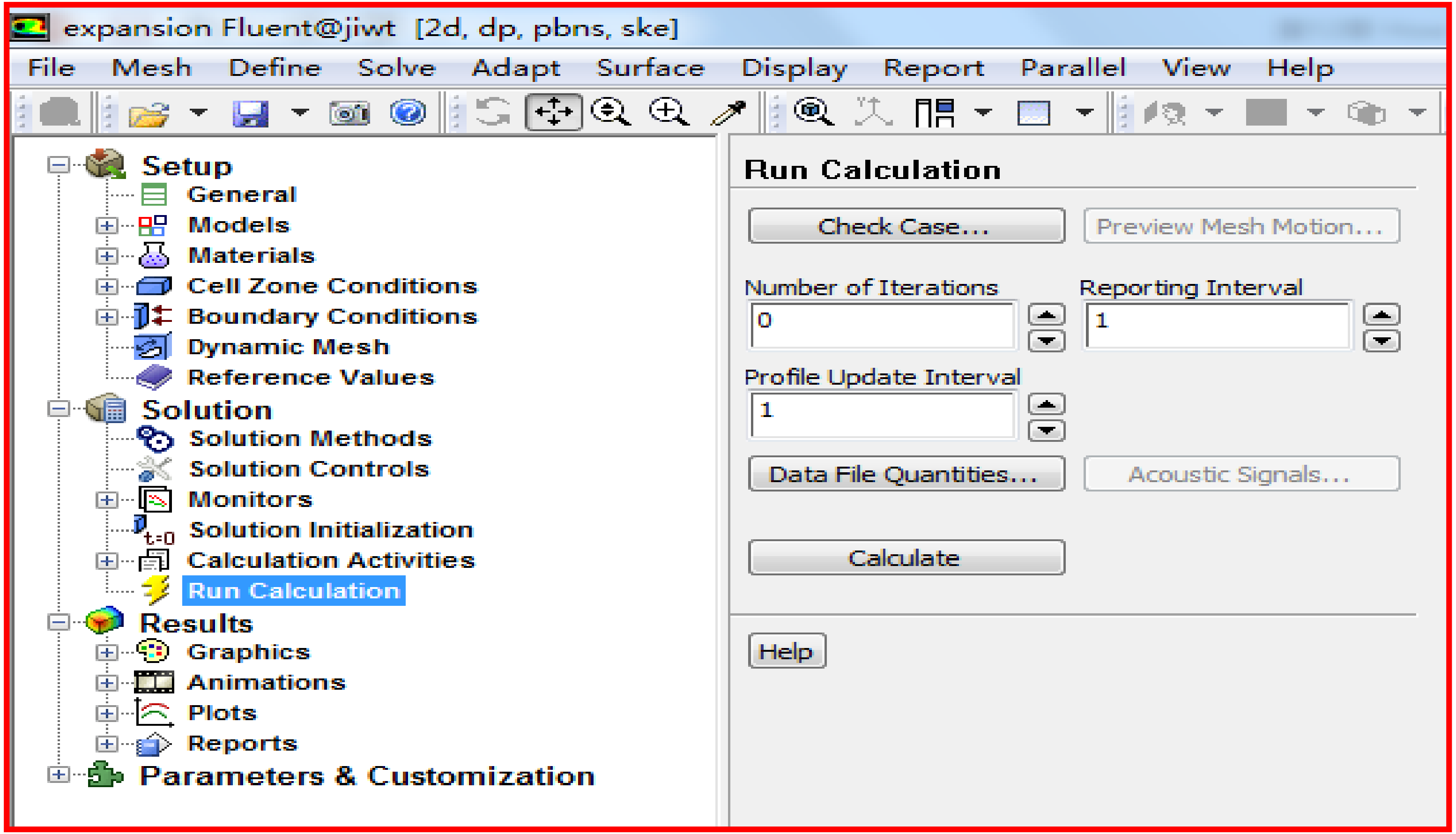


◆ Starting from a Previous Solution

- Convergence rates are dependent on how good the starting point is.
- Therefore if we already have a similar result from another simulation, we 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

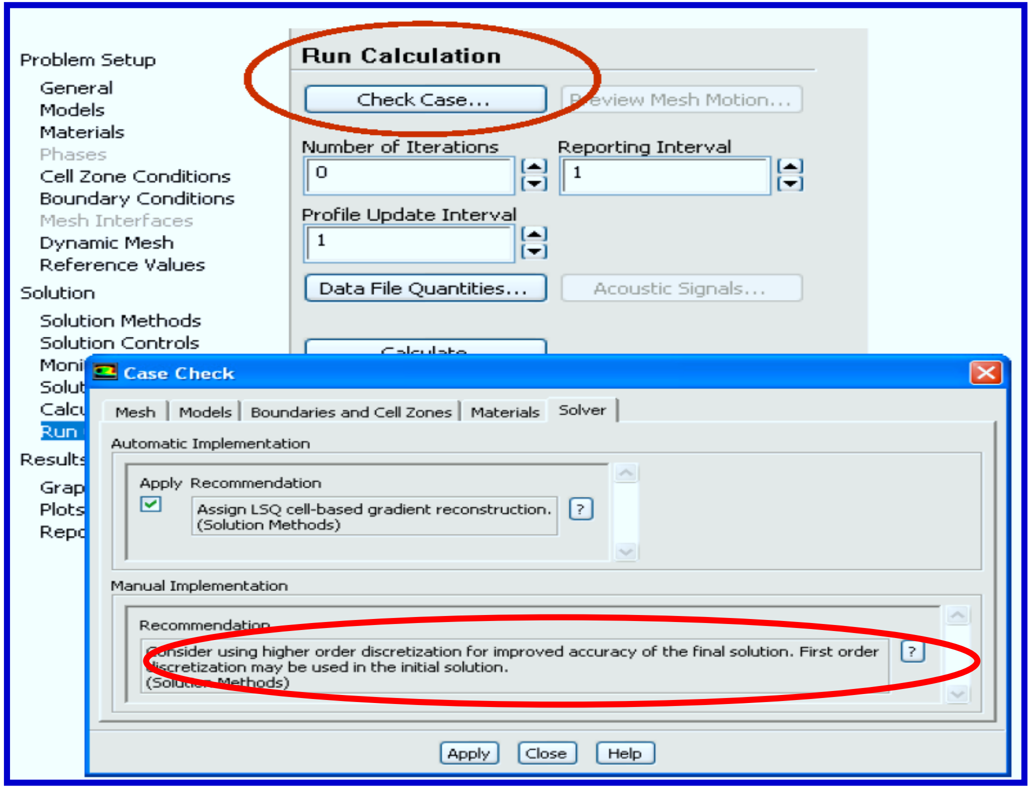


◆ Case Check

①Case Check is a utility in FLUENT which searches for common setup errors and inconsistencies.

②It provides guidance in selecting case parameters and models.

③Contain recommendations which the user can optionally apply or ignore.



◆ Convergence

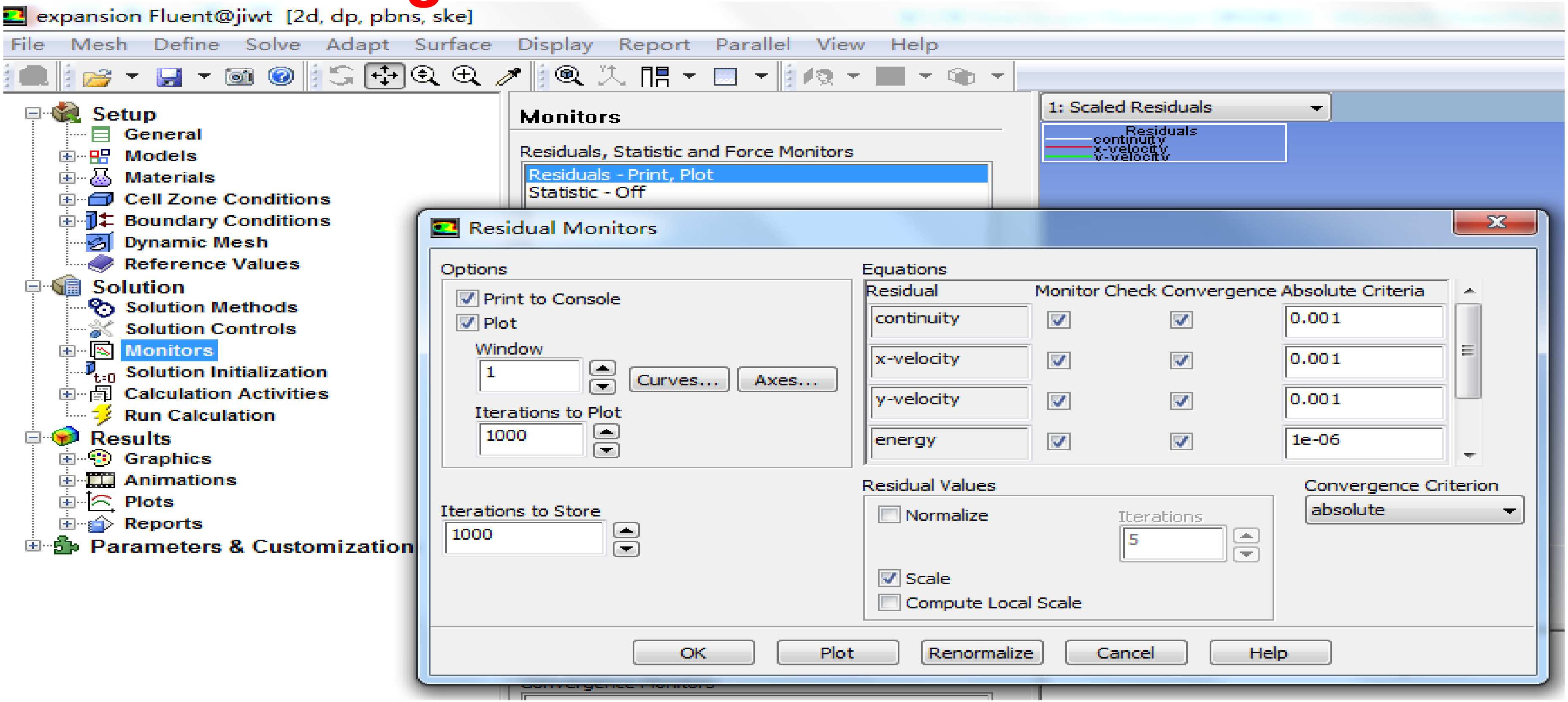
- The solver should be given sufficient iterations so that the problem is converged.
- **At convergence, the following should be satisfied:**

① The solution no longer changes with subsequent iterations.

② Overall mass, momentum, energy, and scalar balances are achieved.

③ All equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance.

9. Monitoring



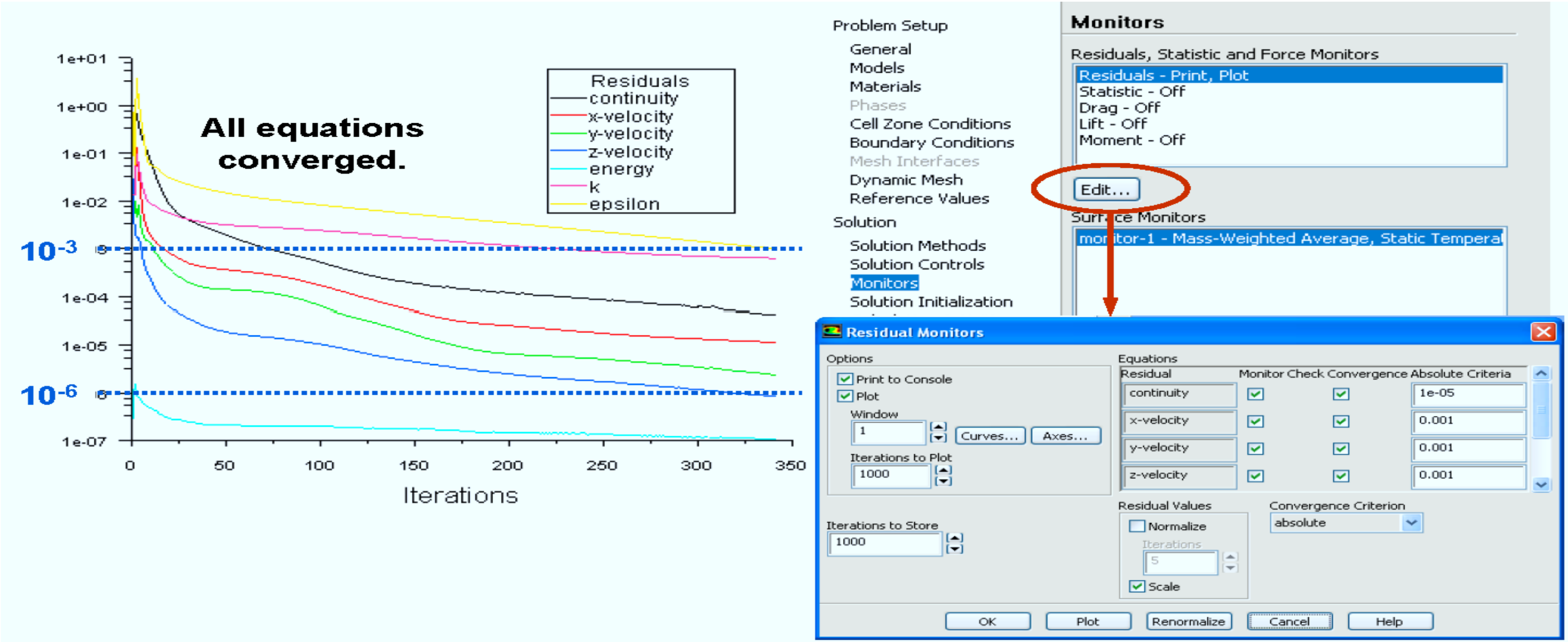
• Monitoring convergence using residual history:

① 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.

② Scaled energy residual should decrease to 10^{-6} (for the Pressure-based solver).

③ Scaled species residual may need to decrease to 10^{-5} to achieve species balance.

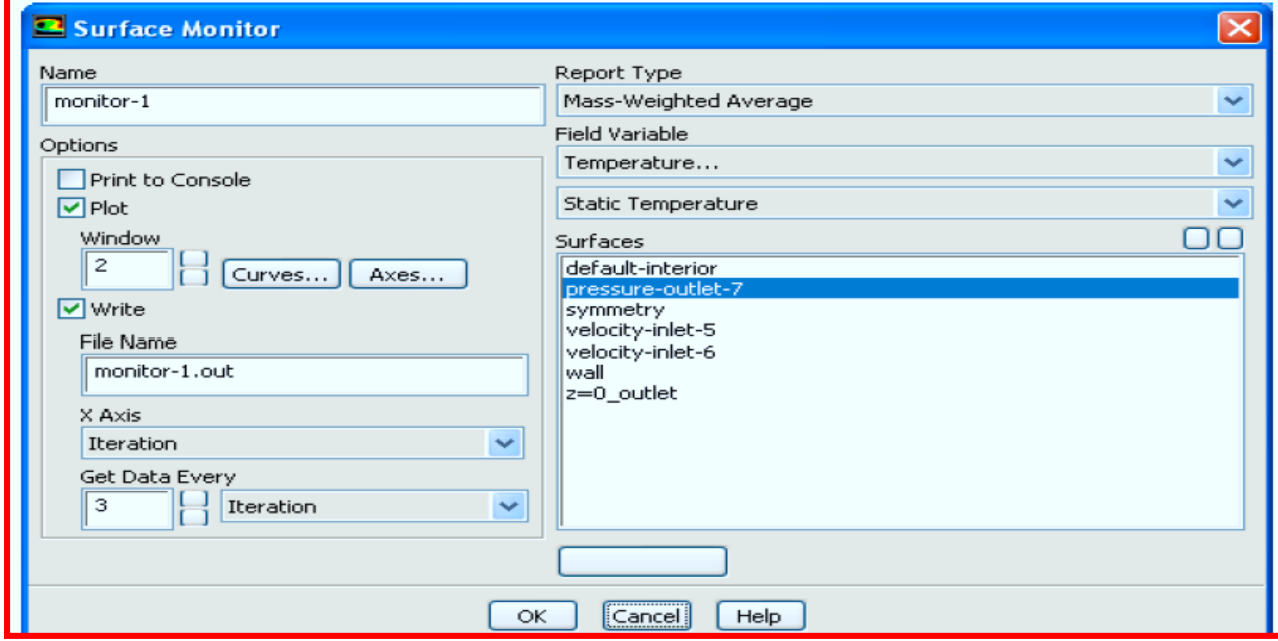
◆Convergence Monitors – Residuals



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

◆ Convergence Monitors – Forces and Surfaces

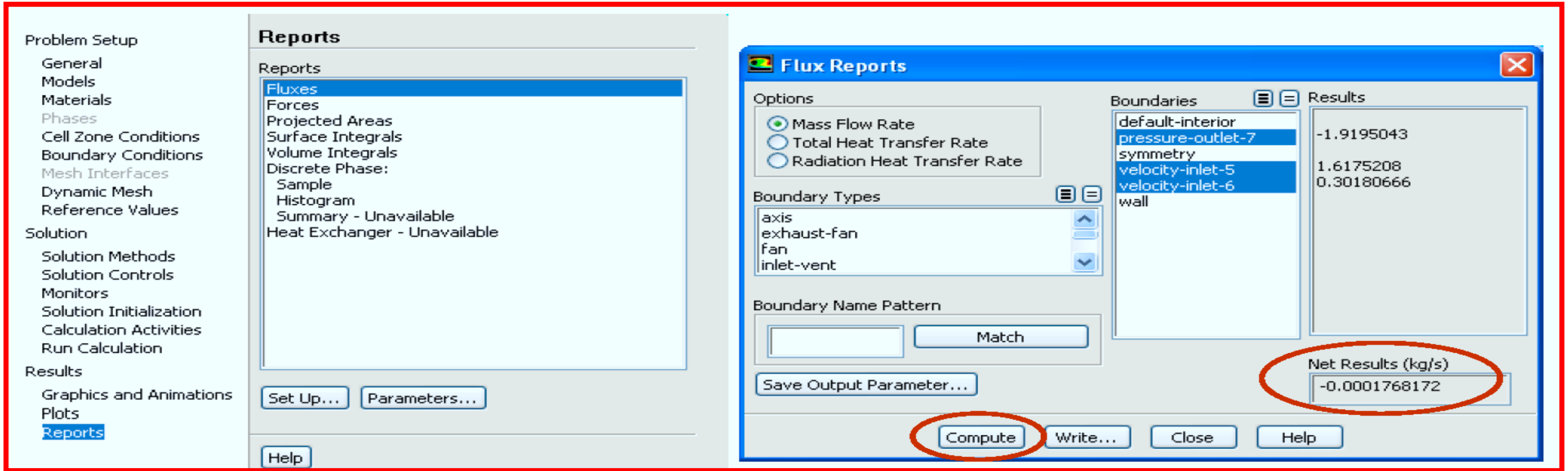
If there is a particular value we are interested in (lift coefficient, average surface temperature etc.), it is useful to plot how that value is converging.



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, we need to:

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

– Continue iterations until the solution converges.

Convergence Difficulties

① **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.**

② **Troubleshooting**

Continuity equation convergence trouble affects convergence of all equations.

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

b. Alter the under-relaxation or Courant numbers.

c .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. When the solution is converged but the pressure residual is still relatively high, the factors for pressure and momentum can be lowered to further refine the solution.

• The recommendation is to always use underrelaxation factors that are as high as possible, without resulting in oscillations or divergence.

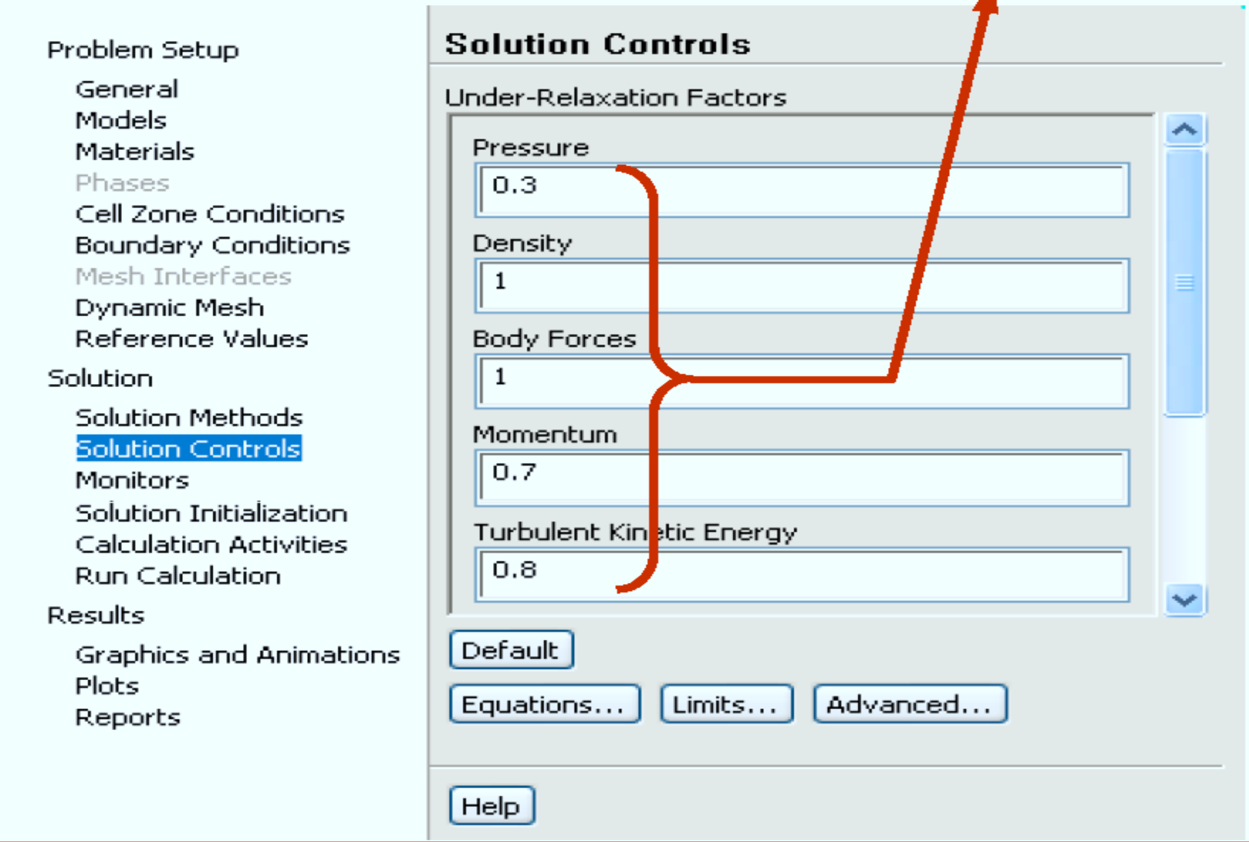
• If value is too high, the model will be unstable, and may fail to converge

• If value is much too low, it will take longer (more iterations) to converge.

– Default settings are suitable for a wide range of problems, we can reduce the values when necessary.

– Appropriate settings are best learned from experience!

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



Solution Accuracy

① **Remember, a converged solution is not necessarily a correct one!**

A. Always inspect and evaluate the solution by using available data, physical principles and so on.

B. Use the second-order upwind discretization scheme for final results.

C. Ensure that solution is grid-independent

② **If flow features do not seem reasonable:**

A. Reconsider physical models and boundary conditions

B. Examine mesh quality and possibly re-mesh the problem

C. Reconsider the choice of the boundaries' location (or the domain): inadequate choice of domain (especially the outlet boundary) can significantly impact solution accuracy.

Mesh Quality and Solution Accuracy

① Numerical errors are associated with calculation of cell gradients and cell face interpolations.

② Ways to reduce the numerical errors:

A. Use higher-order discretization schemes (second-order upwind, MUSCL)

B. Attempt to align grid with the flow to minimize the "false diffusion"

③ Refine the mesh

Refine the mesh

① Sufficient mesh density is necessary to resolve salient(重要) features of flow

- Interpolation(插值) errors decrease with decreasing cell size

② Minimize variations in cell size in non-uniform meshes

A. Truncation error is minimized in a uniform mesh

B. FLUENT provides capability to adapt mesh based on cell size variation

③ Minimize cell skewness and aspect ratio

A. In general, avoid aspect ratios higher than 5:1 (but higher ratios are allowed in boundary layers)

B. Optimal quad/hex cells have bounded angles of 90 degrees

C. Optimal tri/tet cells are equilateral

◆ Grid-Independent Solutions

① It is important to verify that the mesh used was fit-for-purpose.

– Even if the grid metrics(指标) like skewness are showing the mesh is of a good quality, there may still be too few grid cells to properly resolve the flow.

② To trust a result, it must be grid-independent. In other words, if the mesh is refined further, the solution does not change.

③ Typically we should perform this test once for most of our problems.

Determining Grid Independence

Procedure:

① **Obtain new grid:**

Adaption(细化或者粗化网格,可把某些计算区域的网格细化以提高计算精度，也可把某些计算区域的网格粗化以节约计算资源。)

A. **A process by which the mesh is selectively refined in areas that are affected by the adaption criteria specified.**

B. **If we know where large gradients are expected, we need to have fine grids in the original mesh for that region, e.g., boundary layers.**

② **Continue calculation until it converge.**

③ **Compare results obtained with different grids.**

④ **Repeat the procedure if necessary**

10. Results and Analysis: Graphics, Animation and Reports

① Heat flux report:

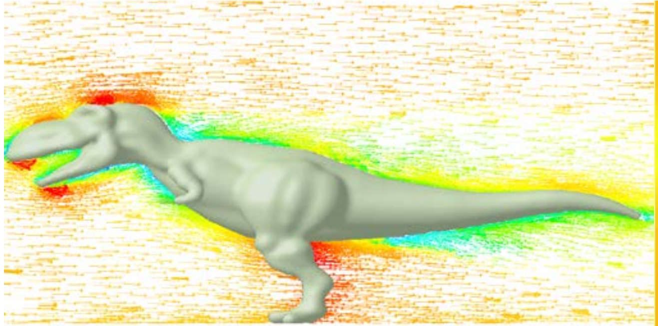
– It is recommended that we perform a heat balance check so to ensure that our solution is truly converged.

② Exporting heat flux data:

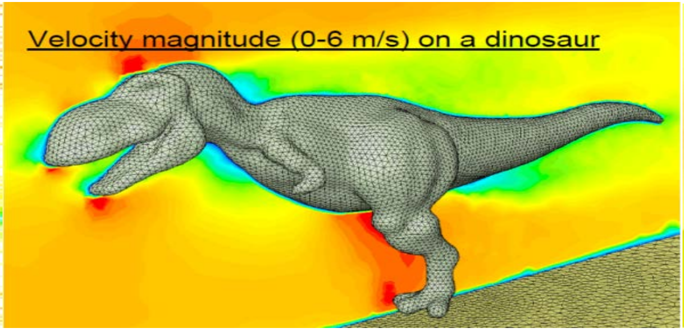
– It is possible to export heat flux data on wall zones (including radiation).

Available Variables for Heat Transfer

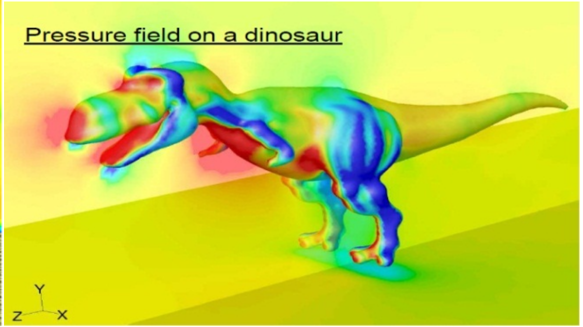
Static Temperature	Total Temperature
Enthalpy	Relative Total Temperature
Rothalpy(滞止焓)	Wall Temperature
Wall Temperature (Thin)	Total Enthalpy
Total Enthalpy Deviation	Entropy
Total Energy	Internal Energy
Total Surface Heat Flux	Surface Heat Transfer Coef.
Surface Nusselt Number	Surface Stanton Number



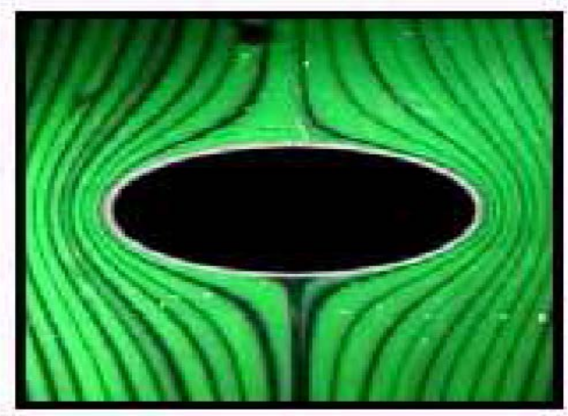
Velocity vectors



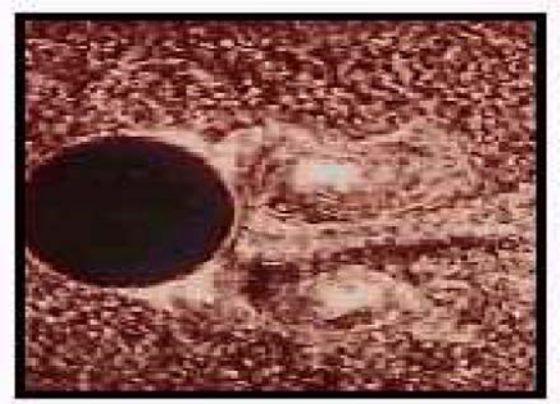
Velocity magnitude



Pressure field



Streamlines



Pathlines

12.4.2. Physical models

Multiphase Flow Modelling

A. Discrete phase model
B. Eulerian model
C. Mixture model
D. Volume-of-Fluid (VOF) model

Reacting Flow Modelling

A. Eddy dissipation model
B. Non-premixed, premixed and partially premixed combustion models
C. Detailed chemistry models
D. Pollutant formation
E. Surface reactions

• Modelling Moving Parts

A. Single and multiple reference frames
B. Mixing planes
C. Sliding meshes
D. Dynamic meshes
E. Six-degree-of-freedom solver

Multiphase Flows

In many flows, there is more than one fluid present in the domain

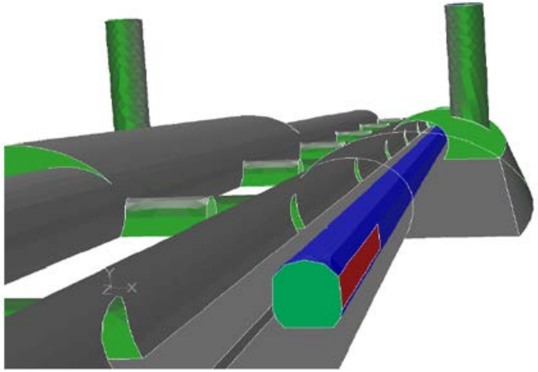
A. Different substances (e.g. oil&water, or water&air)
B. Different phases of same substance (water & steam)

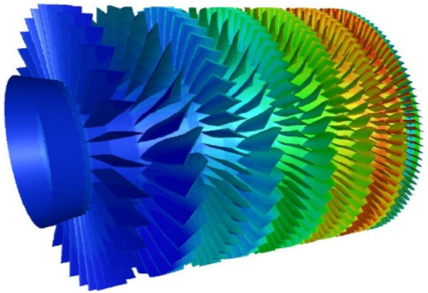
Modelling Moving Parts

- Many flow problems involve domains which exhibit forms of motion.

- Two types of motion are possible – translational and rotational.

- There are two modeling approaches for moving domains:
  - Moving Reference Frames(运动参考坐标系)
  - Moving/Deforming Domains





12.4.3 User Defined Functions

What is a User Defined Function?

① A UDF is a function (programmed by the user) written in C which can be dynamically linked with the FLUENT solver.

- Standard C functions

② Exponential, control blocks, do-loops, file i/o, etc.

- Pre-Defined Macros

③ Allows access to field variable, material property, and cell geometry data and many utilities

Why program UDFs?

Standard interface cannot be programmed to anticipate all needs:

① Customization of boundary conditions, source terms, reaction rates, material properties, *etc.*

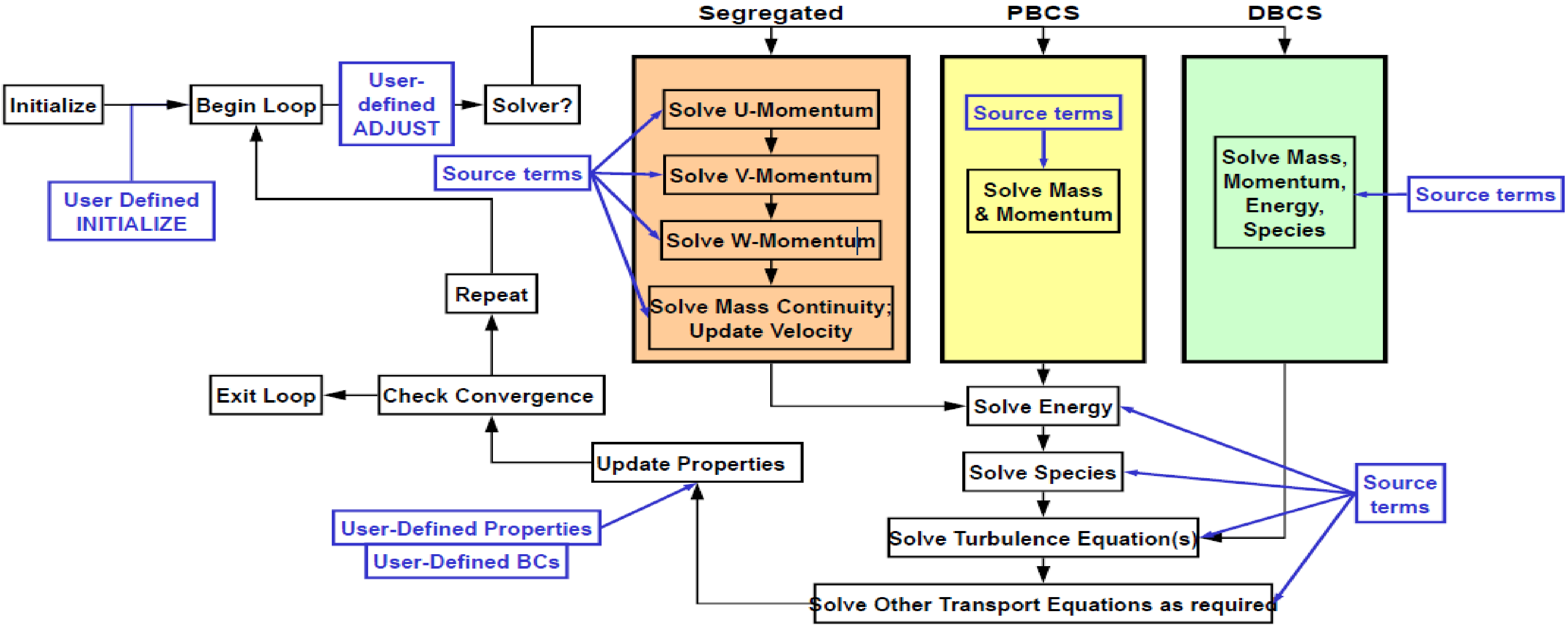
② Customization of physical models

③ User-supplied model equations

④ Adjust functions (once per iteration)

⑤ Execute on demand functions

⑥ Solution initialization



User Access to the FLUENT Solver

12.5 Introduction to ICEM and Meshing with ICEM for structural and unstructural grid

With the continuously growing capabilities of modern computing systems, the demand for more detailed analysis and assessment of fluid behavior is growing. However, the flow domain is in most cases defined by complex geometries, for which it is not always easy to establish a high quality discretized model.

Mesh Generation:

Public domain, downloadable and university codes: more than 100 types.
(http://www.robertschneiders.de/meshgeneration/software.html#Ansys)

Companies offering commercial mesh generation softwares:

ICEM, Ansys meshing, Gambit, Hypermesh, Tgrid, Pointwise, Gridpro, ANSA, turbogrid… ; About 70 software products available.

Which one to choose?

Rule of thumb:

1. Use the mesh generator which is being used by your friends who are available to help you out.

2. ICEM is a good software(Hexa), but it is difficult and takes a lot time to learn.

3. We can also use combination of different meshers. For example: Gambit and ICEM. Use Gambit for geometry cleaning and tetra volume mesh. This mesh was saved in .msh format and imported into ICEM. Where with build topology underlying geometry was reproduced and then prism mesh was extruded from the tetra mesh near to wall surface.

12.5.1 Introduction to ICEM

ICEM CFD mesh types

Mesh Types

- **Unstructured Mesh**
- **Hybrid Mesh**
- **Structured Mesh**

ICEM CFD can generate both structured and unstructured meshes using structured or unstructured algorithms which can be given as inputs to structured as well as unstructured solvers.

- **Mesh**

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

  - **Heat Transfer**
  - **Fluid dynamics**
  - **Other**

- **Nodes**
  - **Point locations of element corners**

2D – Surface/Shell

① Quads(四边形)

② Tris（三角形）

3D-Volume

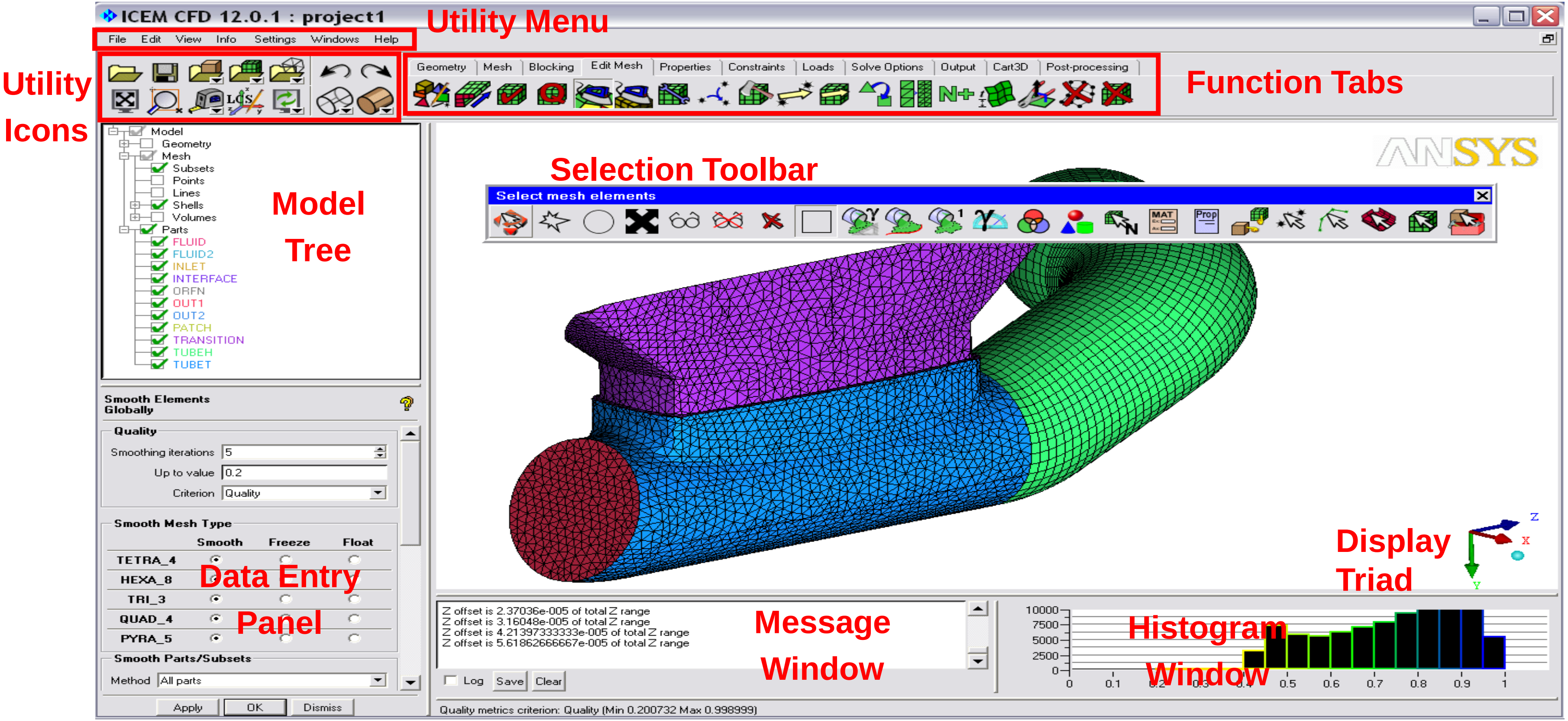
① Tetra（四面体）

② Pyramid（棱锥）

③ Prism（棱柱）

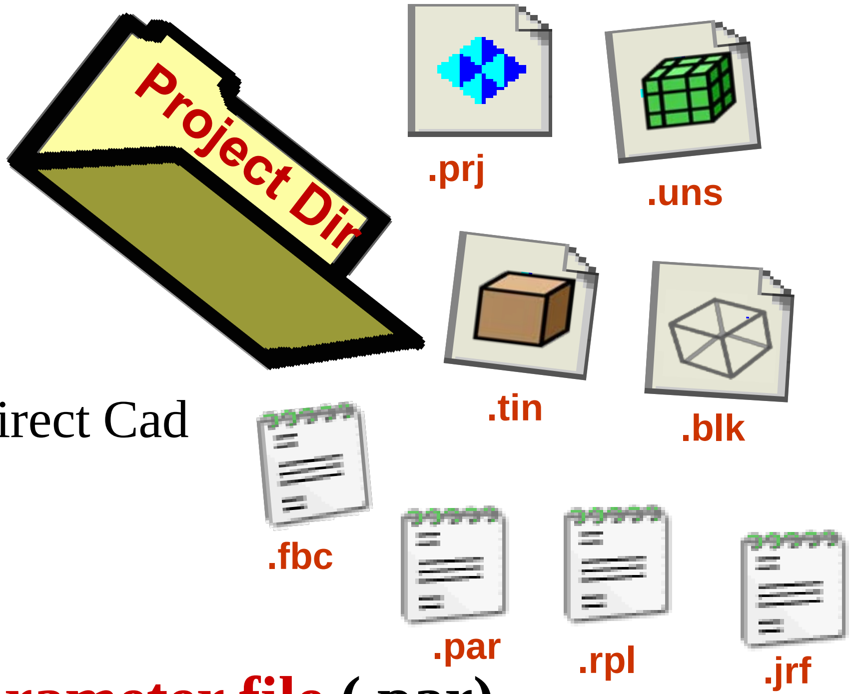
④ Hexa（六面体）

GUI and Layout for ICEM CFD



File and Directory Structure

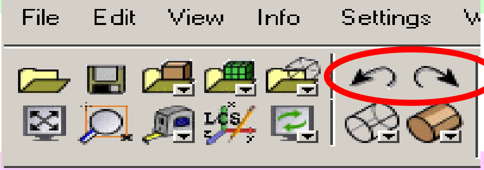
- **Primary file types:**
  - **Tetin (.tin):** Geometry
    - Geometry and material points
    - Part association
    - Global and entity mesh sizes
    - Created in Ansys ICEM CFD or Direct Cad Interface
  - **Domain file (.uns)**
    - Unstructured mesh
  - **Blocking file (.blk)**
    - Blocking 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)

Some Commonly Used Utilities

① Edit > Undo/Redo



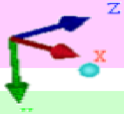
② View

–Fit

•Fit active entities into screen

–Box Zoom

–Standard views

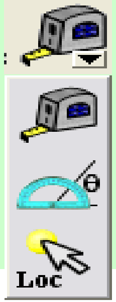


③ Measure

–Distance

–Angle

–Location

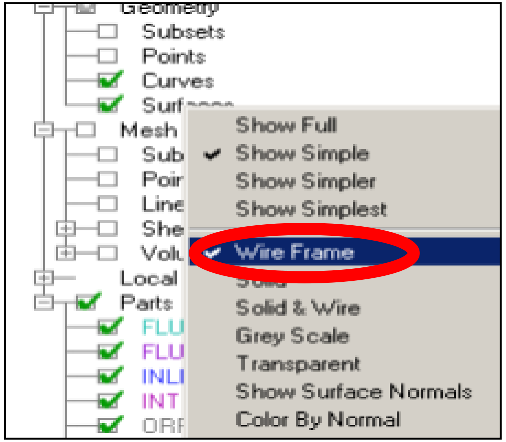


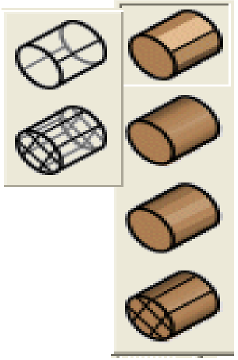
④ Surface display

-Wireframe

-Solid

-Transparent





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

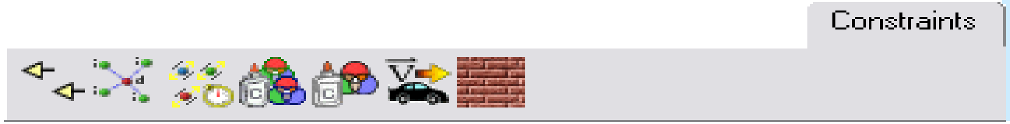
Set Boundary Conditions and Parameters;
Write mesh for 100+ solvers.

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

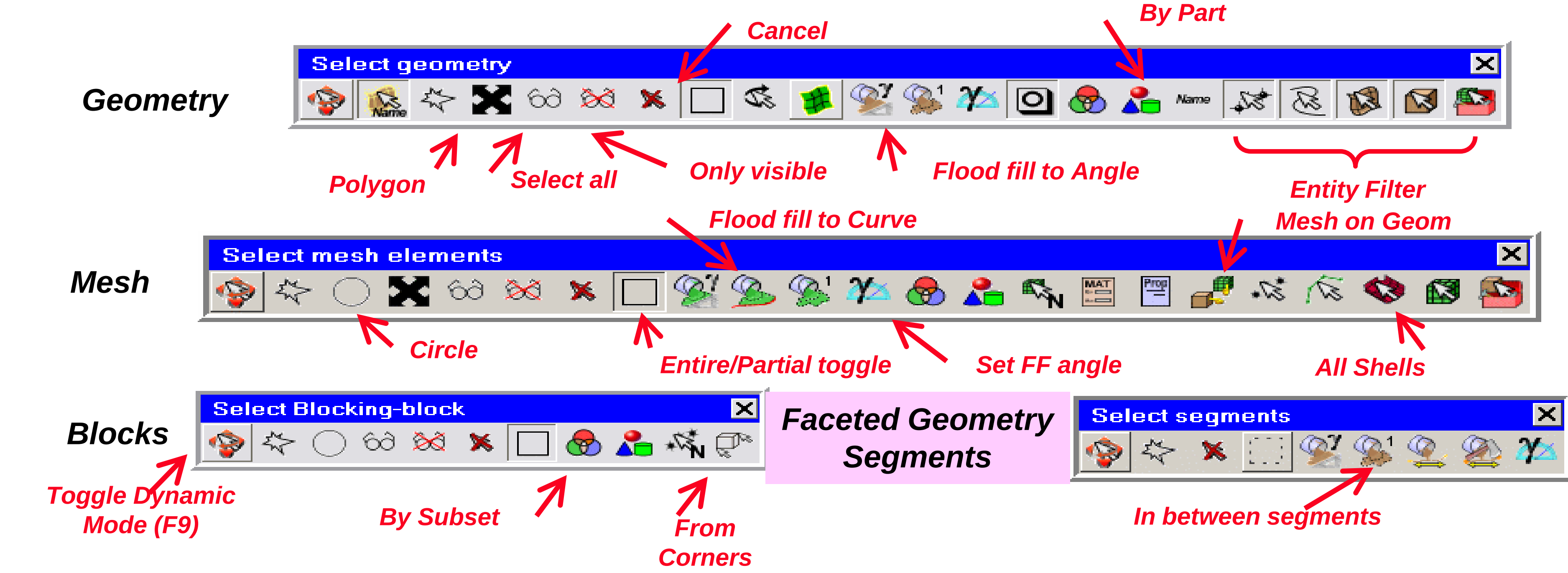
Set parameters, attributes, create subcases, write out input file, run solver

Post Processing

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

Selection Toolbar

- During select mode, selection toolbar appears
  - Filtering of entities
  - Linked to select mode hotkeys



Workflow

Typical ICEM Workflow:

1.Create/open new project

2.Import/Create geometry

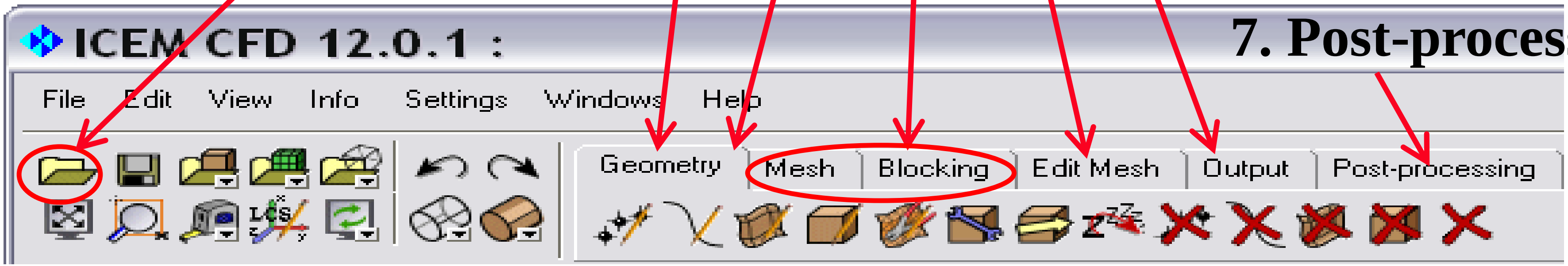
3.Build topology/Clean geometry

4. Mesh model (Possibly Hex Blocking)

5. Check/edit mesh

6. Output to Solver

7. Post-process



Workflow

The unstructured mesh generation procedure:

1. Create/Import geometry
2. Repair geometry ensuring a closed volume
3. Determine global meshing parameter
4. Specify part mesh setup
5. Specify curves and surface mesh size
6. Compute mesh

ICEM was designed to mainly import geometry, not create complicated geometries, although many geometry tools are provided.

ICEM CFD provides:

Geometry import

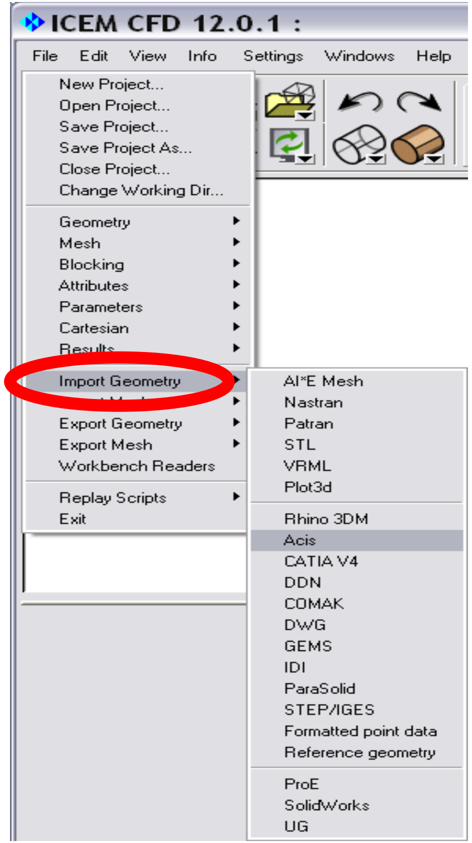
① Directly from CAD package

② 3rd party formats (step, acis, etc…)

③ Via Workbench/Design Modeler

- Surface geometry kernel
  - Imported solids are converted to surfaces
  - Geometry fixing

Geometry Import



CAD from just about any source

① **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

② **Direct import**

–ACIS (.sat) -CATIA V4 -DWG/DXF

–Catia V4 -IDEAS -GEMS

–IDEAS (IDI) –Parasolid -STEP/IGES

–Pro/E -Unigraphics

12.5.2 Introduction to Surface/Shell Meshing with ICEM

Usages of shell meshing:

① **Thin sheet solid modeling (FEA)**

② **2D cross sectional analysis(二维) (CFD)**

③ **Input for volume meshing (FEA/CFD)**

1.General Procedure

First need to decide mesh setup parameters

① Mesh method

– Algorithm used to create mesh

② Mesh type

– quad/tri/mix

③ Mesh sizes

a. Small enough to capture physics, important features

b. Large enough to reduce grid size (number of elements)

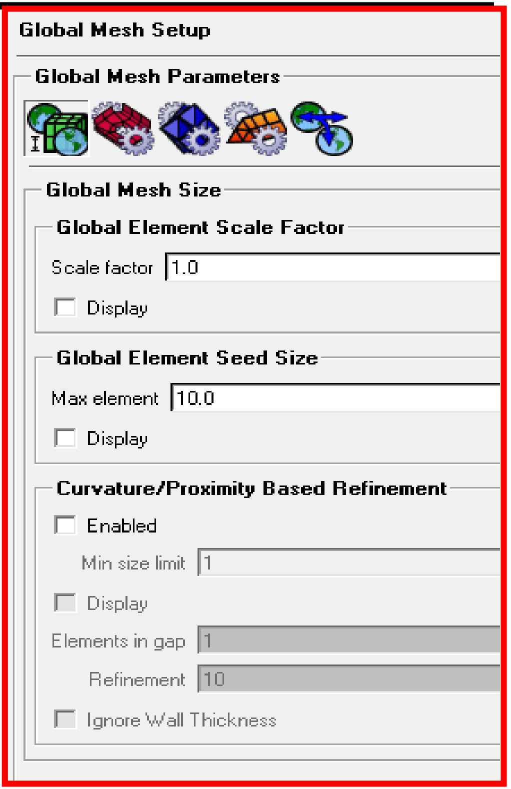
- Memory limitations
- Faster mesh/solver run

a. Set mesh sizes on parts, surfaces, and/or curves

b. Based on edge length

Can have different types/methods set on different surfaces

2. Global Mesh Setup



Global Mesh Size

① For entire model

② Scale factor（全局比例参数）

- Global setting by which many local settings are multiplied
- Good for scaling overall mesh

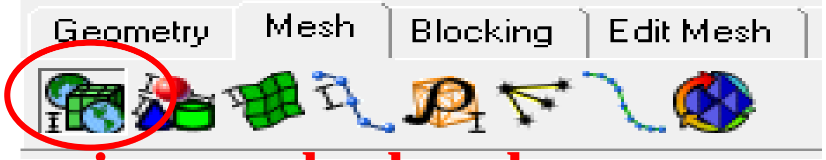
③ *Global Element Seed Size*（生成网格的全局尺寸）

- Maximum possible element size in model
- Default size if don’t wish set sizes

④ *Curvature/Proximity Based Refinement*（根据曲率和邻近区域的网格细化）

- Automatically creates smaller element size to better capture geometry
- Only for Patch Independent method and tetra octree

Global Mesh Setup Mesh tab



① To change defaults globally for size, method and type

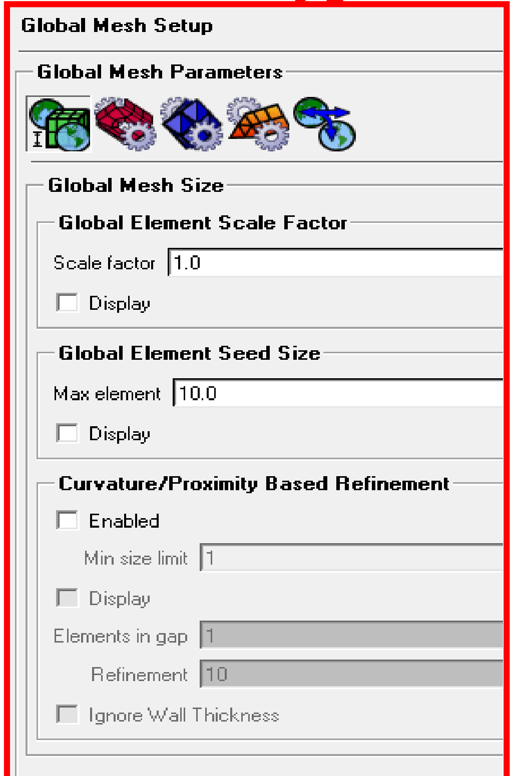
② For entire model

③ For Shells

④ For Volume

⑤ For Prism

⑥ To periodicity



Parameters relative to scale factor

– Max size

– Min size limit

– Height

– Max deviation

3.Global Shell Meshing Parameters

– From *Global Mesh Setup* tab

– *Set surface mesh parameters globally*

- **Defaults for the selected mesh method**

（所选网格方法的默认设置）

– *Method*

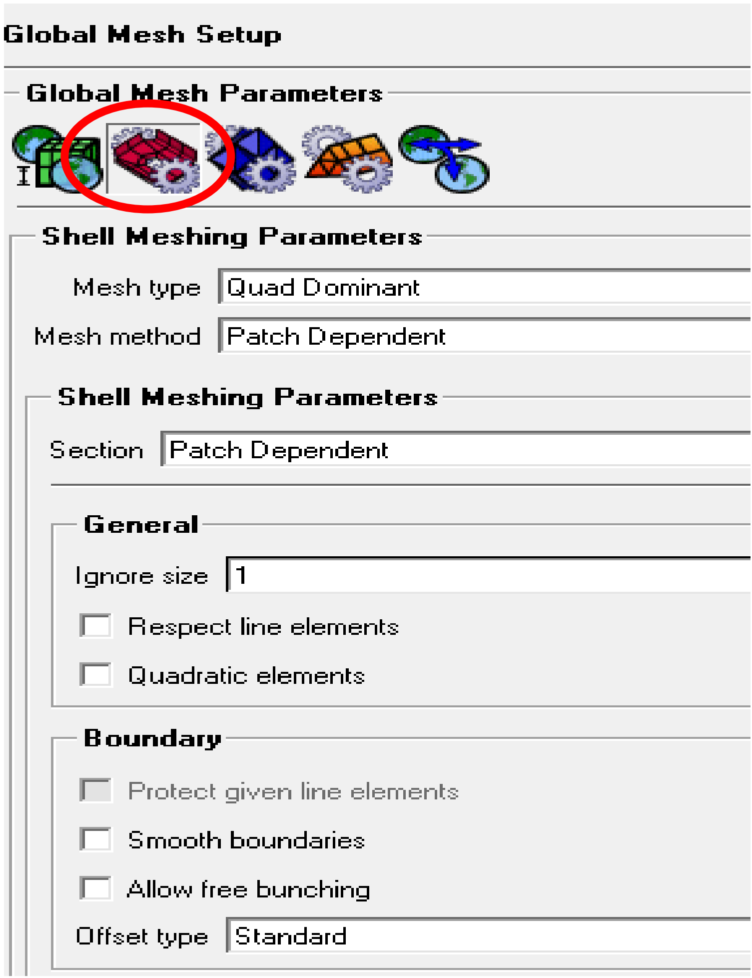
① **Autoblock(自动每个面生成二维 Block后生成网格)**

② **Patch dependent(根据面的轮廓线来生成网格)**

③ **Patch independent(网格生成不严格按照轮廓线)**

④ **Shrinkwrap(忽略大的几何特征)**

⑤ **Delaunay(三角网格生成算法)**



Type

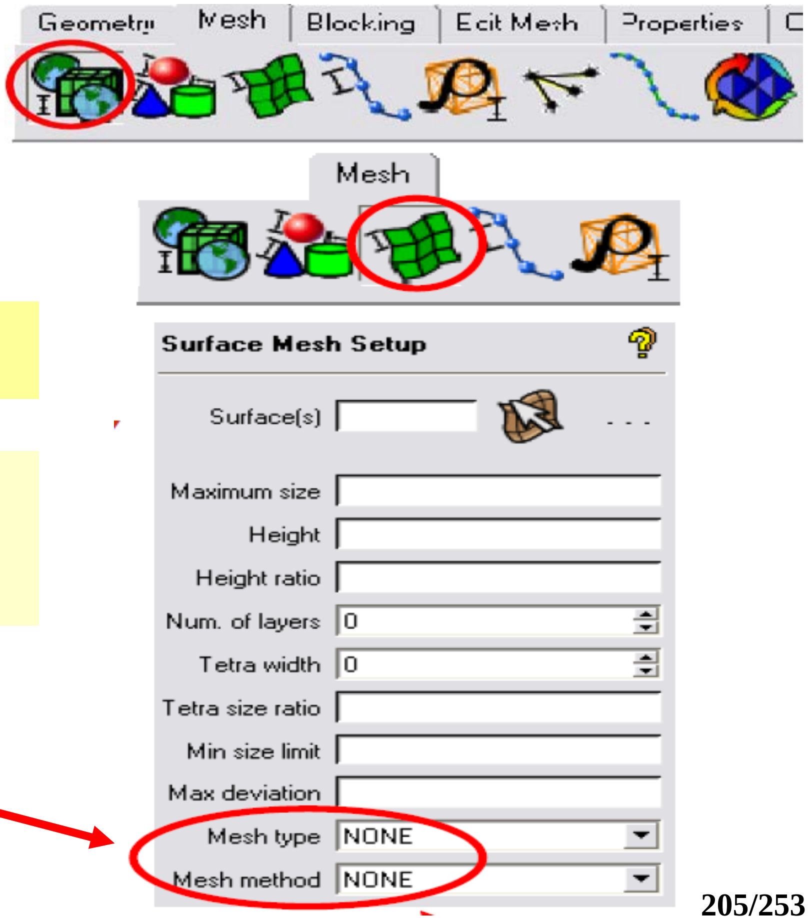
- All Tri, Quad with one tri
- Quad dominant, All quad

– Options for different methods

– Global types and methods can be overridden by:

Local settings

In Surface Mesh Setup

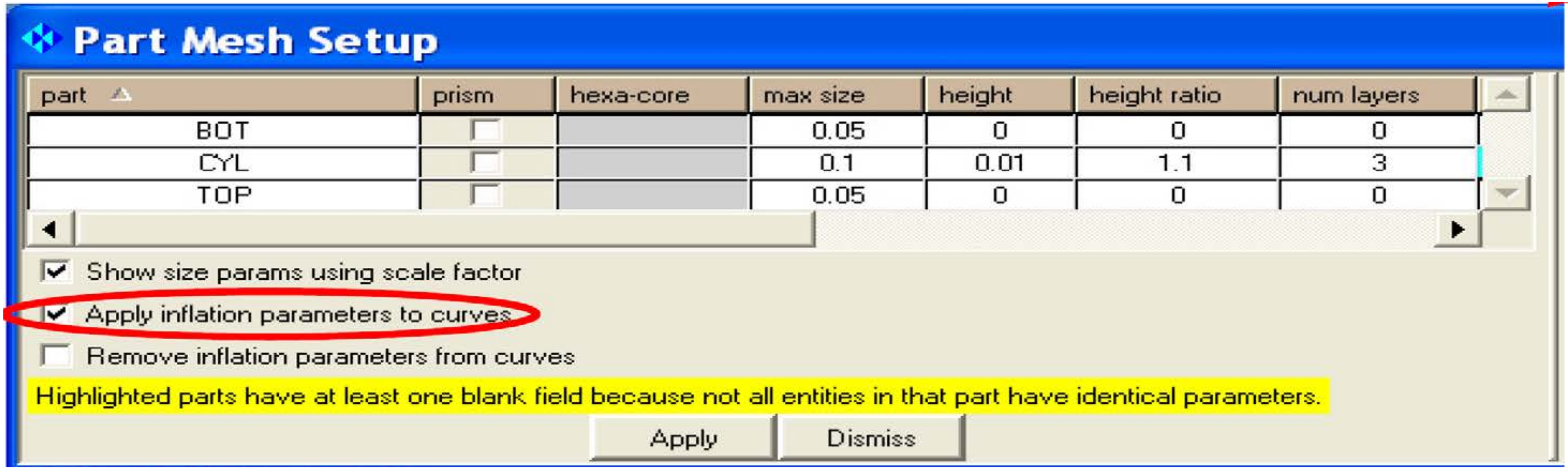


4. Part Mesh Setup

– **Set mesh parameters on all entities within part**

– ***Max. size***

- **Multiplied by global *Scale Factor* = actual size**



Quad layers grown from curves (e.g. rings around holes) , use these 3 parameters:

– ***Height:*** **First layer quad height on curves**

– ***Height ratio:*** **growth ratio which determines the heights of each subsequent layer**

– ***Num layers:*** **Number of rings/inflation layers**

Inflation layers: Mesh orthogonal to surface with faces perpendicular to boundary layer flow direction (膨胀边界层)

Part Mesh Setup

part	prism	hexa-core	max size	height	height ratio	num layers
BOT	☐		0.05	0	0	0
CYL	☐		0.1	0.01	1.1	3
TOP	☐		0.05	0	0	0

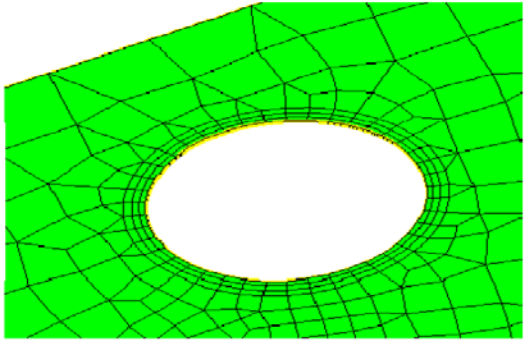
☑ Show size params using scale factor

☑ Apply inflation parameters to curves

☐ Remove inflation parameters from curves

Highlighted parts have at least one blank field because not all entities in that part have identical parameters.

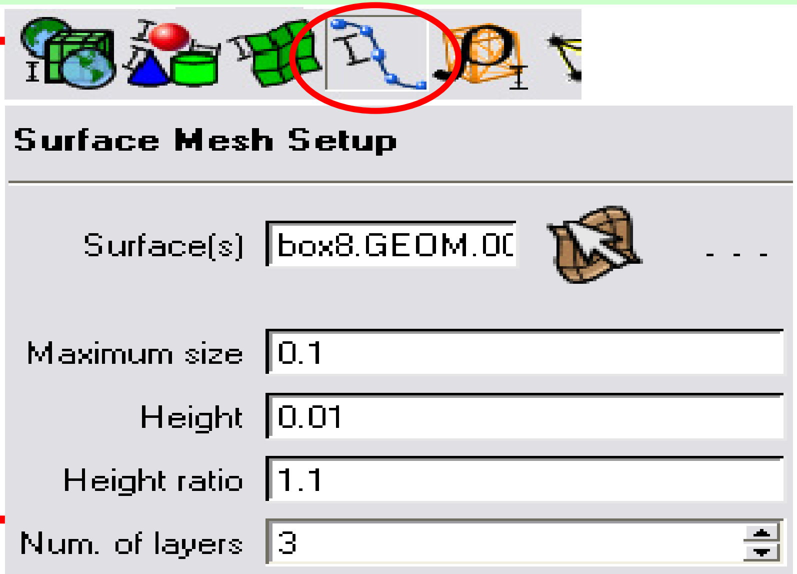
Apply | Dismiss



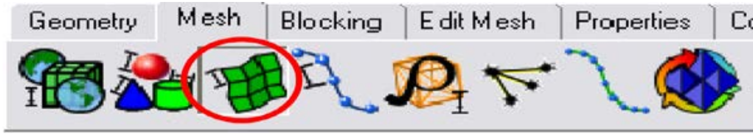
- For quad layers, the minimum required to be set is ***height*** (1 layer) or ***num layers*** (height = max. size) 3-5

- If done in the ***Part Mesh Setup*** interface we must set up the ***Apply inflation parameters to curves***

Or set on individual curves



5. Surface Mesh Setup



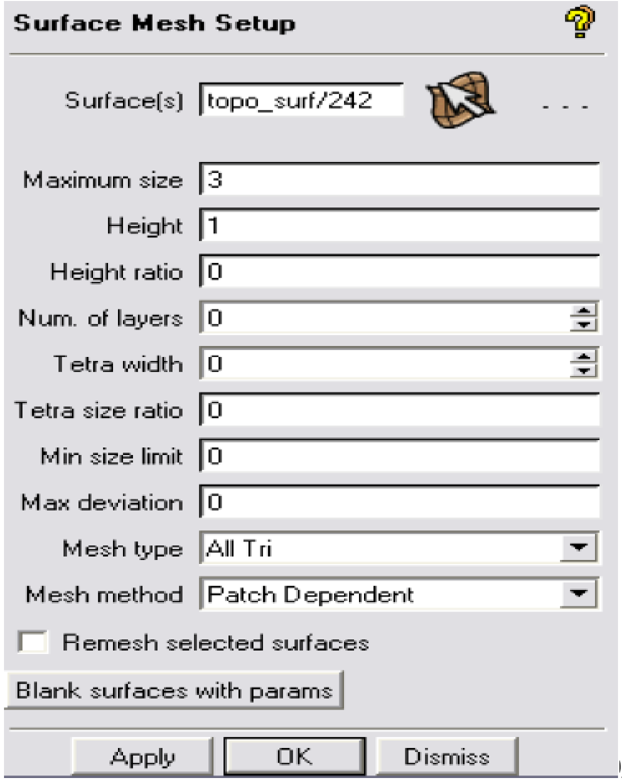
① Same parameters as part mesh setup but also includes:

– *Mesh type*
– *Mesh method*

② Select surfaces first from screen, set sizes/parameters and *Apply*

③ Mesh method/type will override(覆盖) global shell mesh settings for selected surface(s)

④ Will override (覆盖) *Part Mesh Setup* settings if set afterward

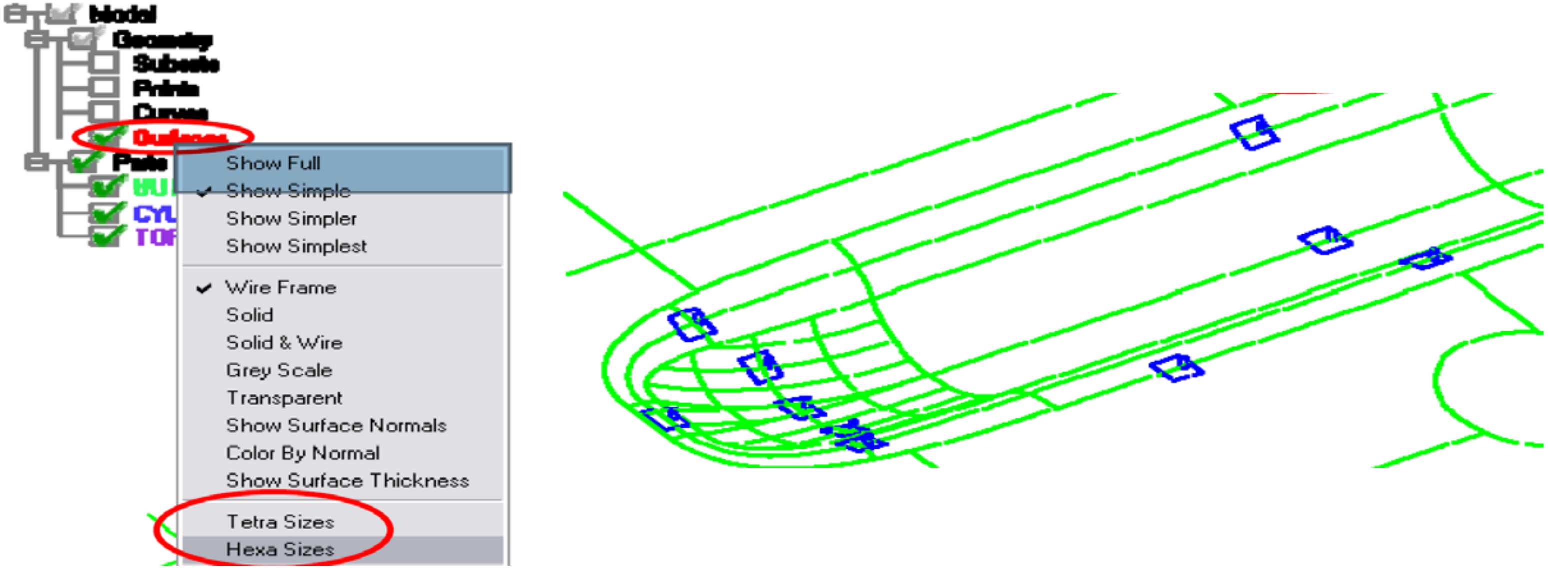


Display

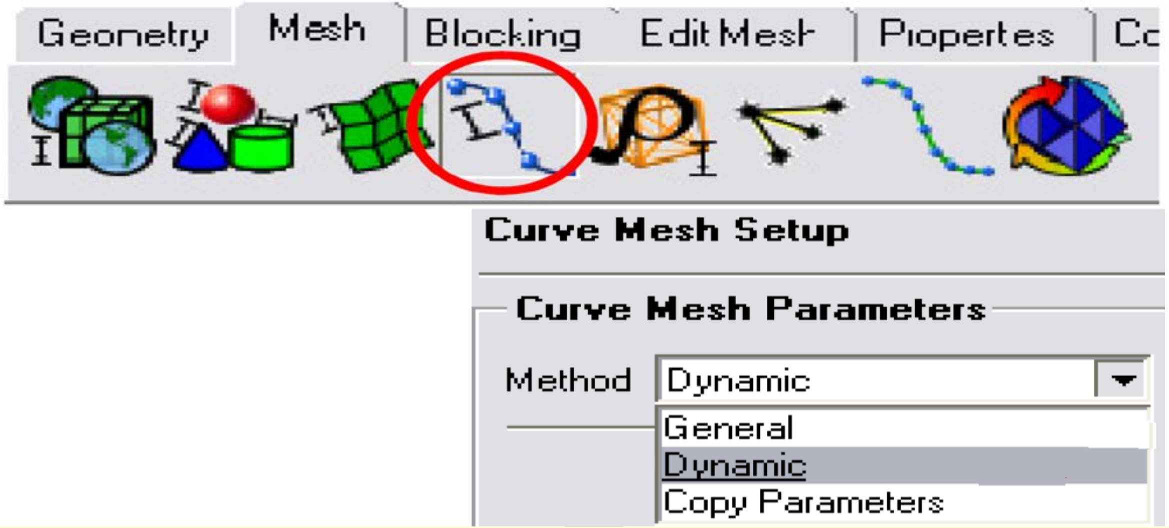
- **Right mouse, select in Model tree on**

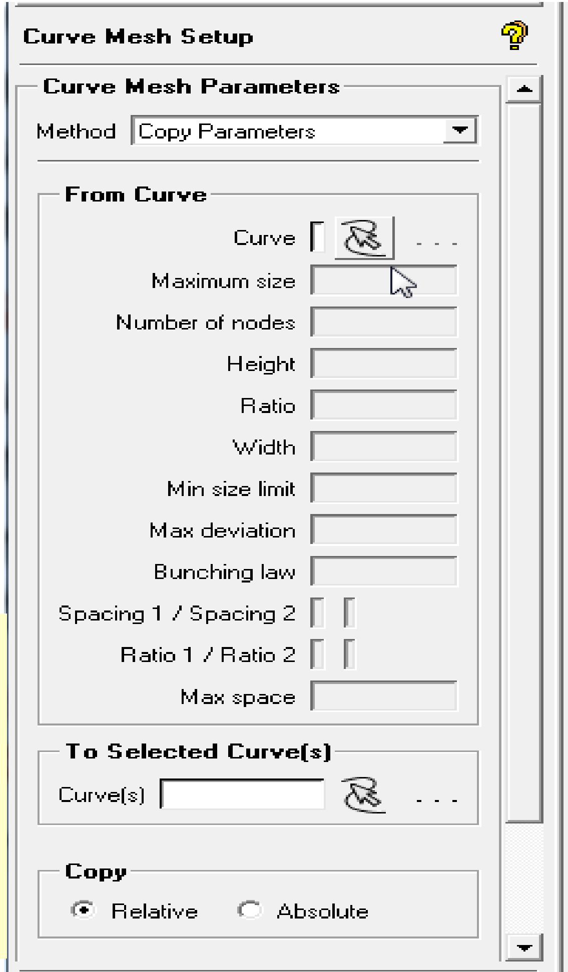
  Surfaces > Tetra/Hexa Sizes

  - **Icon appears for each surface**
  - **Gives a visual estimate of prescribed max. size**



6. Curve Mesh Setup



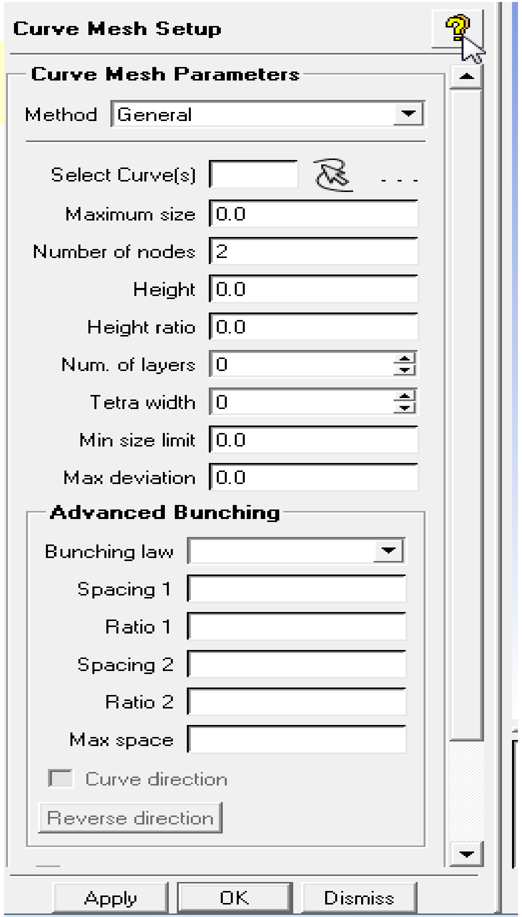


– *General*

- Same as *Surface Mesh Setup*
- But also can prescribe(设置) *Number of nodes*
  - Instead of element size

- **Also includes node biasing along curves(调节节点疏密)**
  - **Initial spacing from either curve end**
  - **Bunching laws(指定节点分布规律)**
  - **Expansion ratios from either curve end**
  - **Matching of node spacing to adjacent curves**

Select curves first, middle mouse to accept selection, then type in parameters/sizes - *Apply*



Curve Mesh Setup

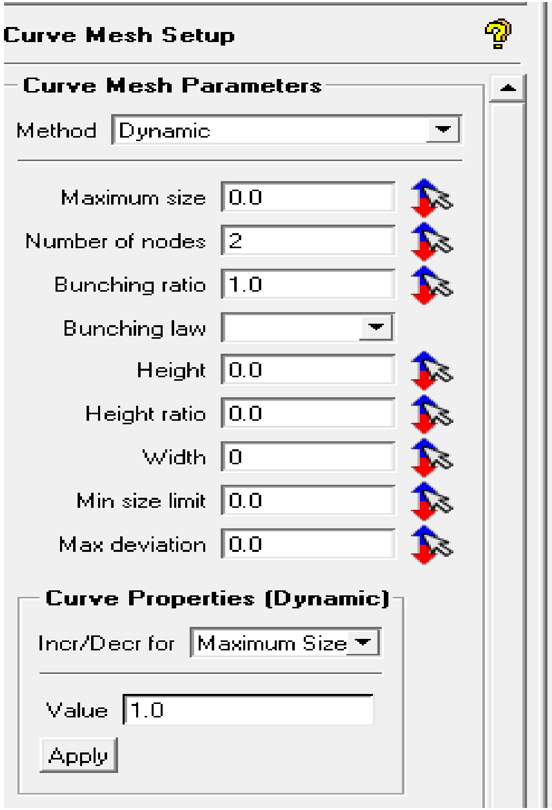
– *Dynamic Method*

- **Adjust mesh parameters on screen**
- **Interactively toggle(切换)displayed values near curve with left (to increase)/right mouse (to decrease) keys**

– *Copy Parameters*

- **Copy parameters set on curve to one others**
  - **e.g. parallel curves, downstream**

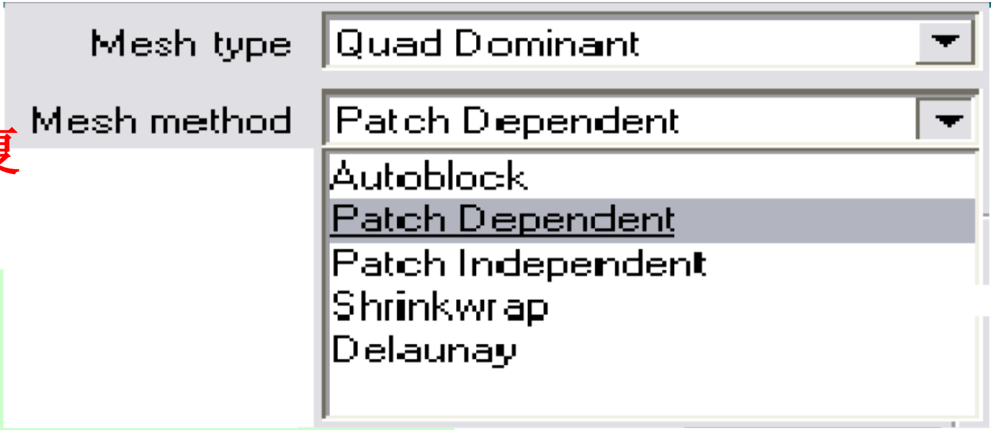
– ***Curve Mesh Setup*** **will override** ***Part Mesh Setup*** **parameters if set afterward**



7. Mesh Methods

Algorithm used to create mesh

- **Patch Dependent(根据面的轮廓线来生成网格，能够较好地捕捉几何特征,创建以四边形为主的高质量网格)**
  - Based on loops of curves surrounding patches
  - Best for capturing surface details and creating quad dominant mesh with good quality
- **Patch Independent (网格生成不严格按照轮廓线，生成网格过程中能够忽略小的几何特征,适用于精度不高的几何模型)**
  - Robust octree algorithm
  - Good for dirty geometry, ignoring small features, gaps, holes
- **Autoblock（自动在每个面上生成二维的 Block然后生成网格）**
  - Based on 2D orthogonal blocks
  - Best for mapped meshing, mesh follows contours of geometry



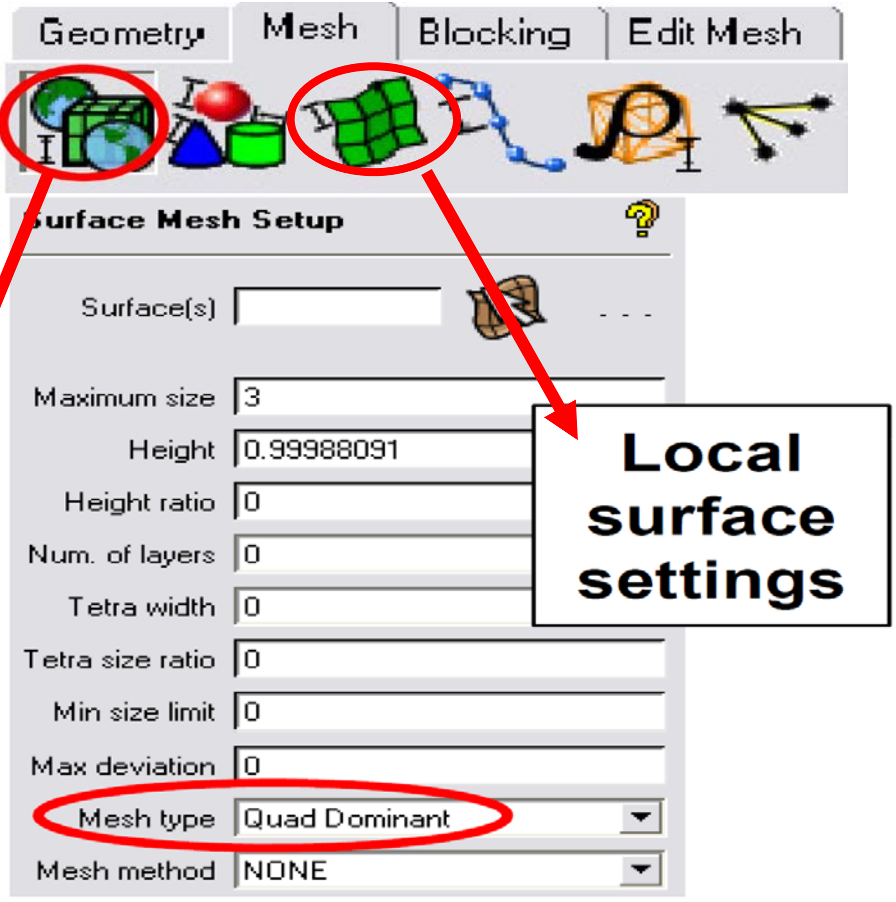
- **Shrinkwrap(一种笛卡尔网格生成方法,会忽略大的几何特征,适用于复杂的几何模型快速生成网格)**
  - **Automatic defeaturing(模型简化)**
  - **Quick Cartesian algorithm**
  - **Allows ignoring of larger features, gaps and holes**

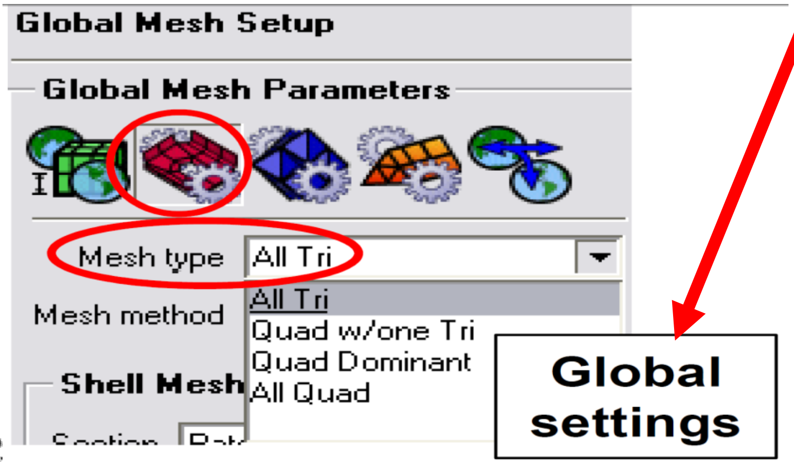
- **Delauney（三角网格生成算法）**
  - **Allows for transition in mesh size**
    - **Coarser towards surface interior**
  - **Tri only**

8. Mesh Types

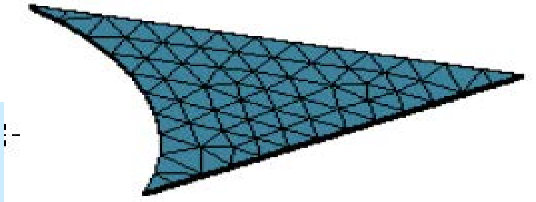
– Set in *Global Mesh Setup > Shell Mesh Parameters* or *Surface Mesh Setup* (local upon selected surface entities

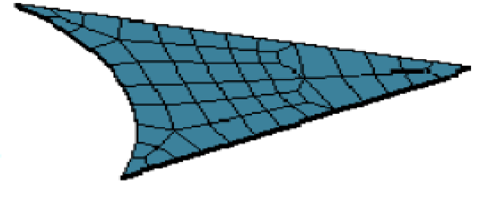
- Global defaults overridden(覆盖) by **local settings**





Mesh Types

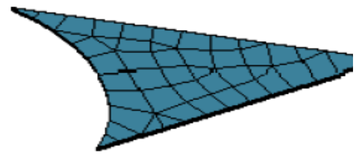




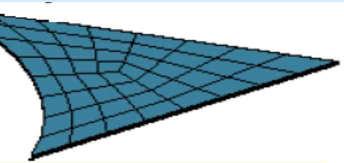
– ***All Tri***

– ***Quad with one Tri*（面上的网格单元大部分为四边形,最多允许有一个三角形网格单元；）**

- **Almost all quad except with one tri per surface**
- **Single tri allows transition between uneven mesh distribution on loop edges**
- **Where pure quad will fail**



All quad, autoblock



– ***Quad Dominant***

- **Allows for several transition triangles**
- **Very useful in surface meshing complicated surfaces where a pure quad mesh may have poor quality**

– ***All Quad***

These mesh types will look different with the different mesh methods

9. Compute Mesh

Once sizes, methods and types are set – ready to compute!

- **Select *Mesh > Compute Mesh >Surface Mesh Only***

– Most of the time can just select *Compute* at bottom of panel which will create shell mesh for entire model *(Input* = *All*)

– Overwrite Surface Preset/Default Mesh Type/Method
- **To quickly override global and local settings**
- **Avoid going back to other *Mesh Setup* menus to change parameters**

Input

① Can mesh ***All*** (default – entire model)

② **Visible** – only visibly displayed surfaces/geometry

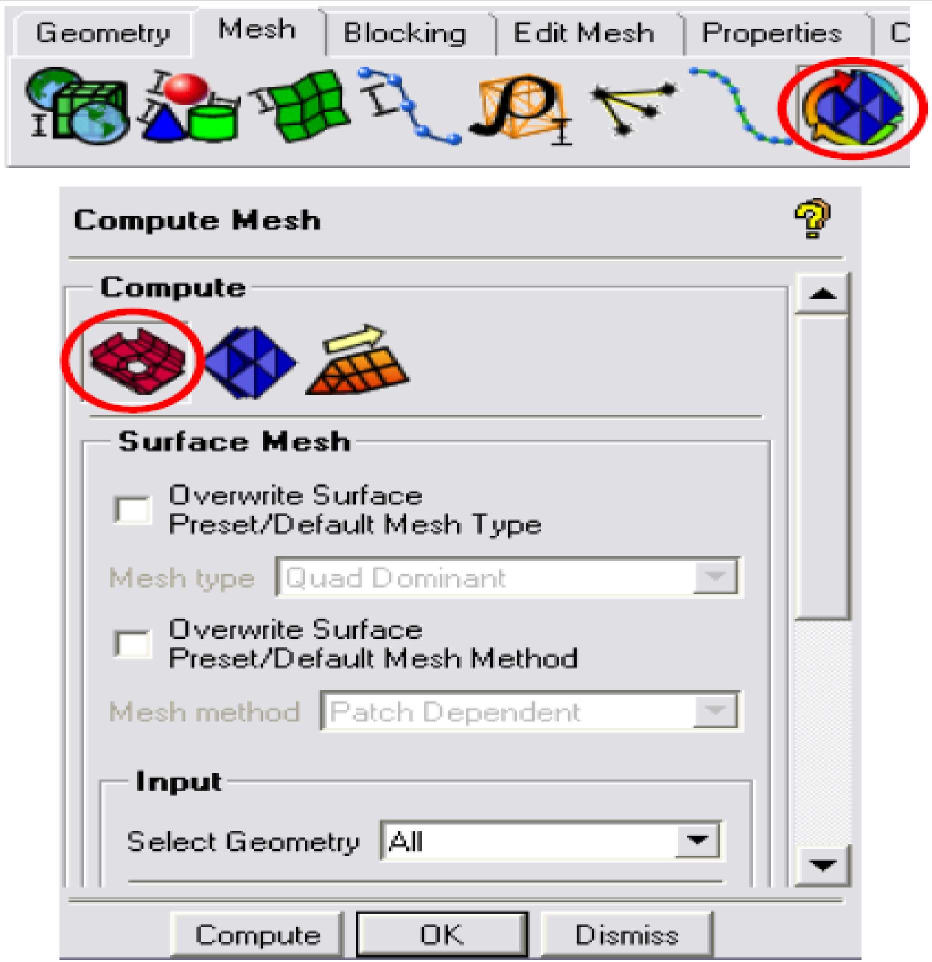
③ **Part by Part**

– Parts meshed separately

– Mesh will be non-conformal between parts

④ **From Screen**

– Select entities to mesh from screen


Geometry
Mesh
Blocking
Edit Mesh
Properties
C
Compute Mesh
Compute
Surface Mesh
Overwrite Surface Preset/Default Mesh Type
Mesh type
Quad Dominant
Overwrite Surface Preset/Default Mesh Method
Mesh method
Patch Dependent
Input
Select Geometry
All
Compute
OK
Dismiss

12.5.3 Introduction to Volume Meshing with ICEM

To automatically create 3D elements to fill volumetric domain

① Generally termed “unstructured”（非结构网格）

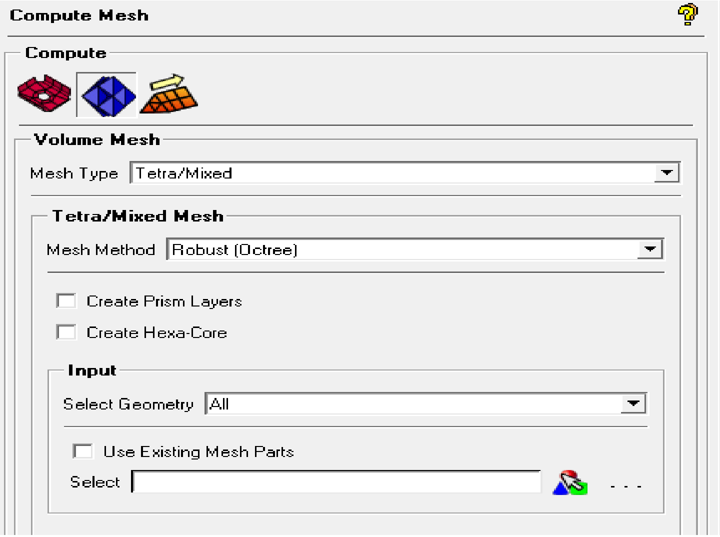
② Full 3D analysis

- Where 2D approximations don’t tell the full story

③ Internal/External flow simulation

④ Structural solid modeling
⑤ Thermal stress

$\Big\}$ Finite element analysis



1. Standard procedures

① Start from just geometry(从已有几何体开始）

- Octree tetra(八叉树四面体)
  - Robust
  - Walk over features
- Cartesian（笛卡尔方法）
  - Fastest

② Start from existing shell mesh

- Delauney/T-grid
  - Quick
- Advancing Front
  - Smoother gradients, size transition（梯度光顺，尺度过渡）
- Hex Core（六面体核心网格）
- Hex Dominant（全六面体）

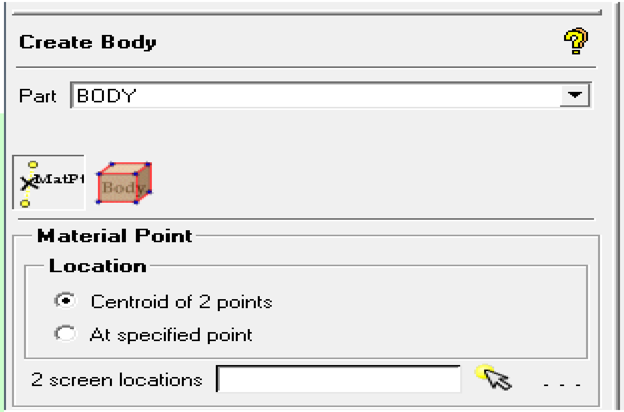
③ Start from both geometry and shell mesh

- Potions of model already meshed
  - Inflation layers
- “Prism” sizes

2. Define Volumetric Domain(设置体区域)

– Optional
- Recommended for complex geometries（复杂几何体）
- Multiple volumes（多个几何体）

– *Geometry -> Create Body– Material Point*

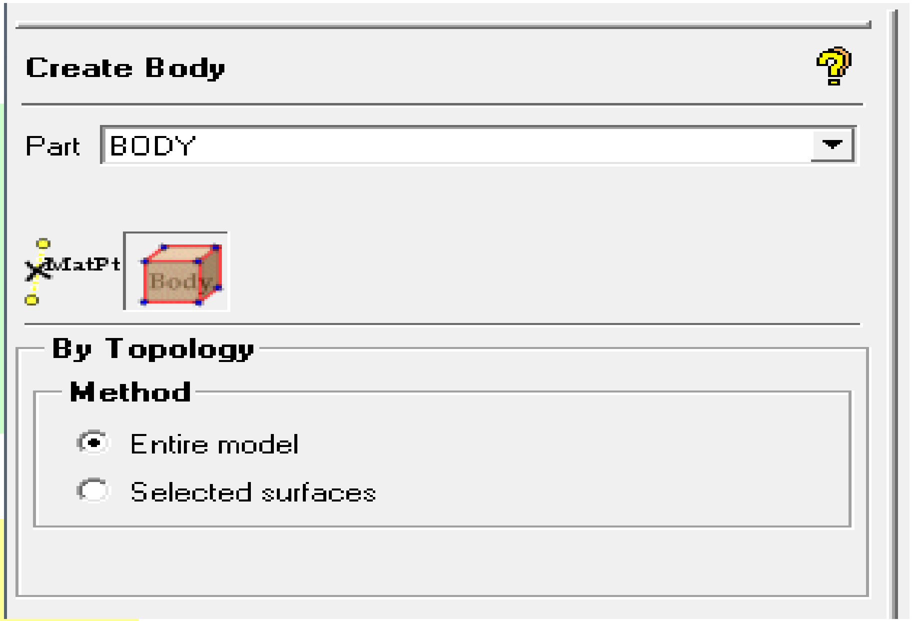


Material Point

- Centroid of 2 points
  – Select any two locations whose mid-point is within volume
  – Preferred（首选方法）
- At specified point
  – Define volume region by "point" within volume

By Topology

- **Defines volume region by set of closed surfaces（通过设置封闭曲面定义体）**
- **Must first *Build Diagnostic Topology*（必须先建立诊断拓扑）**



- ***Entire model***
  - **Automatically define all volumes**
- ***Selected surfaces***
  - **User selects surfaces that form a closed volume（用户从封闭体选择面）**

3. Volume Meshing General Procedure

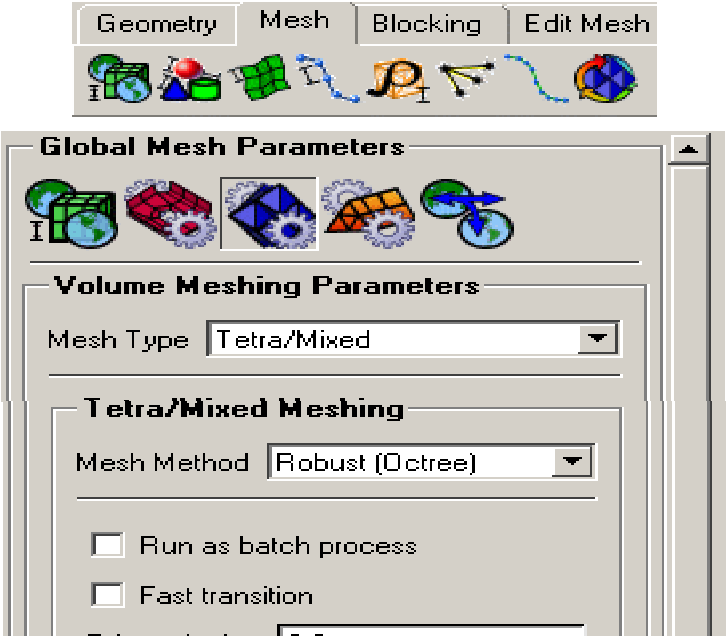
① First decide volume mesh parameters

– *Global Mesh Setup > Volume Meshing Parameters*

– Select *Mesh Type*

– Select *Mesh Method* for selected *Type*

– Set options for specific *Methods*



② Set mesh sizes

- Globally
  - As in *Shell Meshing*
- Locally
  - *Part/Surface/Curve Mesh Setup*
  - As in Shell Meshing
  - For *From geometry* only
    - Octree（八叉树）
    - Cartesian（笛卡尔）

③ Load/create surface mesh

–As in shell meshing section

–For Delauney, Advancing Front, T-grid, Hex-Dominant

• Either of these types run from geometry will automatically create surface mesh using global and local Shell Mesh settings without any user input/editing(以上方法均通过全局和局部网格设置，从几何开始创建网格，无需用户干预)

• If in doubt, run Shell Mesh first, then from existing mesh(先生成面网格，再生成体网格)

④ Define volumetric region

– Typically for octree on complex models(在复杂模型上进行Octree划分)

– Multiple volumes

⑤ Compute Prism (optional)

– As separate process(可作为分开过程)

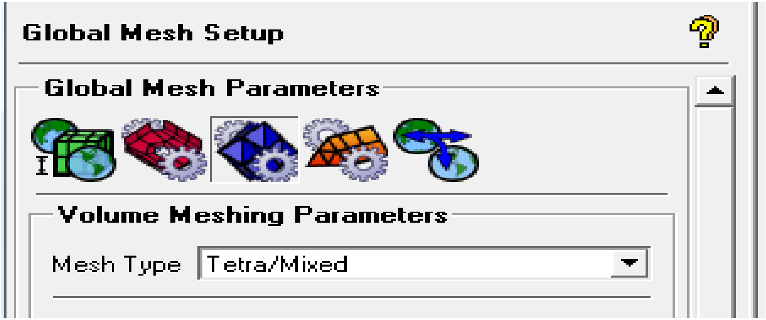
– Also option to run automatically following tetra creation(也可随四面体网格生成自动划分)

⑥ Compute Mesh

– *Mesh> Compute Mesh > Volume Mesh*

4.Mesh Types

① Tetra/mixed（四面体/混合）



– Most used

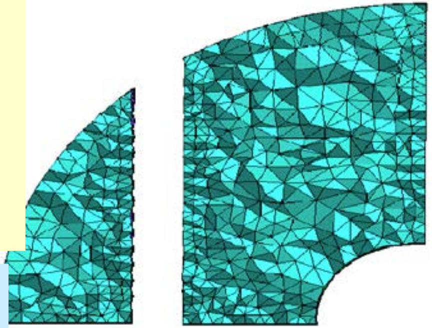
– Tetra

– With hex core（带有六面体核心的网格）

- Hexa (cartesian) filling majority volume（六面体网格填充主要区域）

- Tetra (from delauney algorithm) used to fill between surface or top of inflation layers and hex core（四面体填充表面（或棱柱层）和六面体网格核心之间的区域）

- Pyramids to make conformal between tetra tri and hex quad faces（四面体和六面体之间采用金字塔形状过渡）



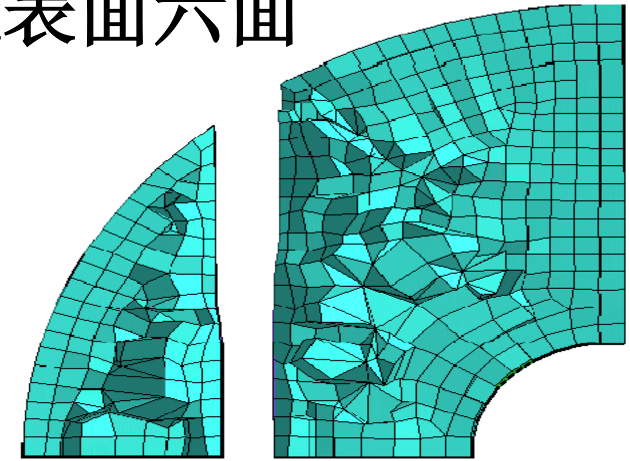
Pure tetra

② Hexa-Dominant（六面体为主）

– From existing quad mesh（从已存在的四边形网格开始）

– Good quality hex near surface（近表面六面体的网格质量较好）

– Somewhat poor in interior（内部网格质量较差）



③ Cartesian（自动生成的六面体非结构网格）

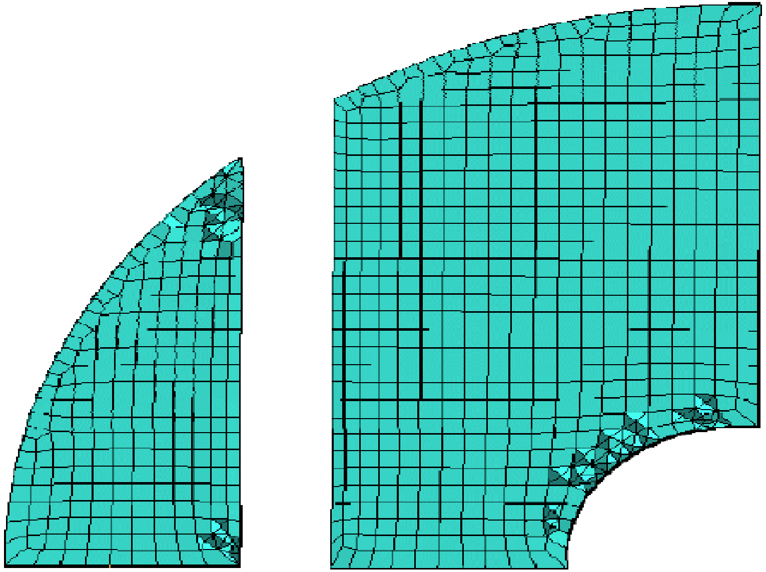
– Automatic pure hexa(纯六面体)

– Rectilinear mesh(直线网格)

– Staircase or(阶梯梯度)

– Body fitted(体适应)

– Fastest method for creating volume mesh



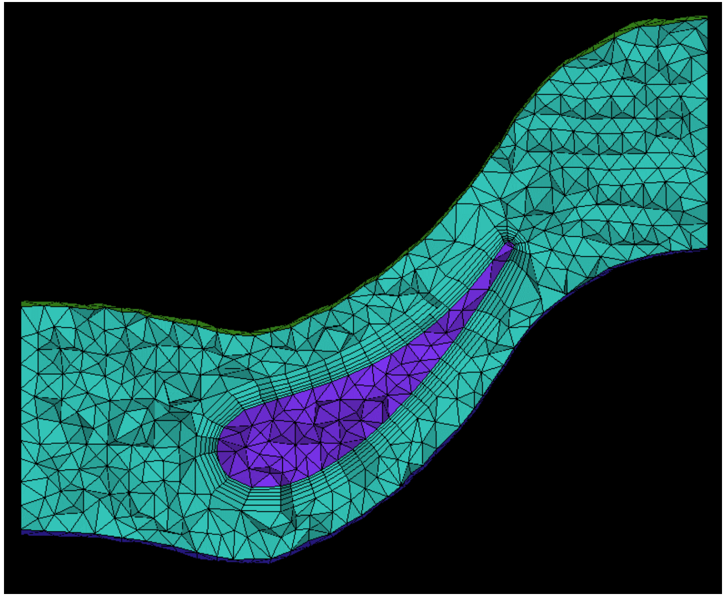
12.5.4. Prism Meshing

The velocity and temperature gradients normal to a wall is typically much larger than the gradients parallel to the wall. Use inflation layers to correctly capture the velocity and temperature gradients near no-slip walls.

Inflation layers:

(1) To simulate the boundary layer effects;

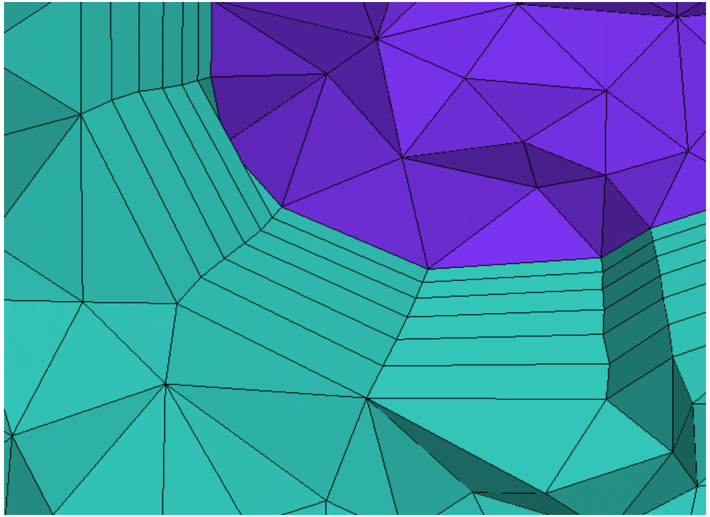
(2) Mesh orthogonal to surface with faces perpendicular to boundary layer flow direction.



Procedure

1) **Set *Global Prism Parameters***

2) **Select *Parts* to grow layers from:**
   - Typically wall boundaries and holes



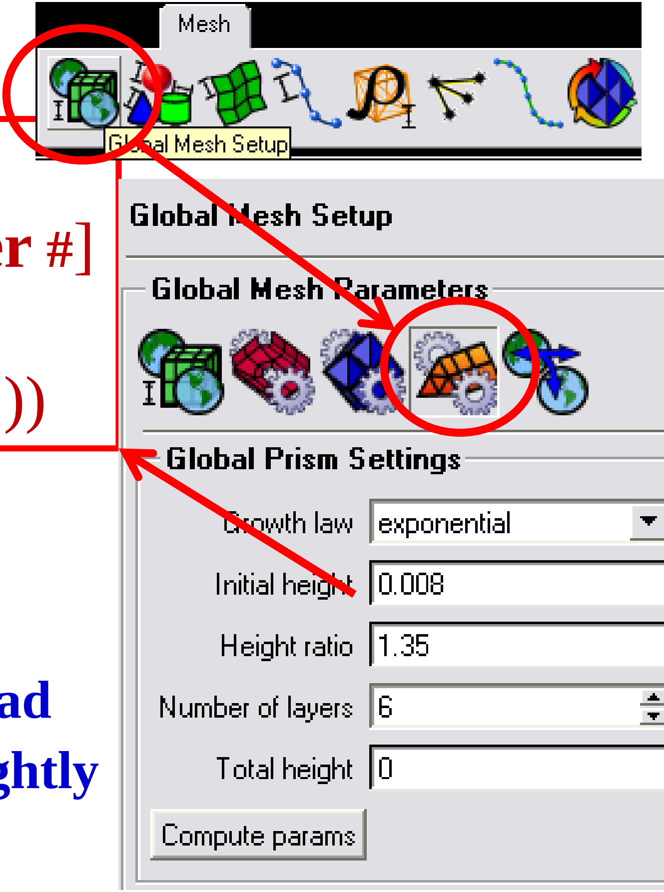
3) **Set Local Parameters for each part**
   - Local overrides global
   - Zero or blank will defer to global settings (零或空白将依照全局设置)

4) **Run mesher**
   - From existing mesh
     - -Extrude into tetra/hexa mesh
     - -Extrude from surface tri mesh, then fill volumes
   - Run automatically during ***Volume Mesh*** creation

Prism -Global Parameters

Growth law

- *exponential*: height = $h(r)^{(n-1)}$ [n is layer #]
- *linear*: height = $h(1+(n-1)(r-1))$
- *wb-exponential*: height = $h*exp((r-1)(n-1))$



Initial height **of first layer:** ***h in above eq.***

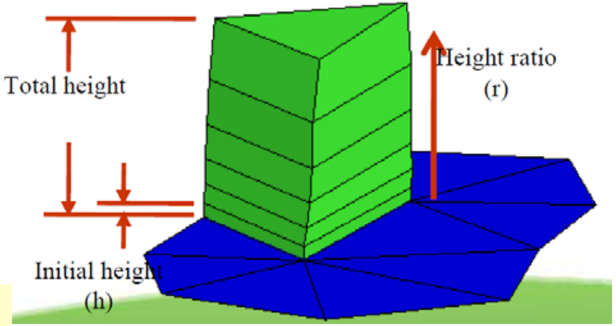
-Auto calculated if not specified

- **Based on factor of edge length of base triangle/quad**
- **Height determined so that top layer volume is slightly less than that of tetra/hex just above it**

Number of layers: ***n***

Height ratio: ***r***

Total height: **usually left blank**



Usually specify 3 of the above 4 parameters

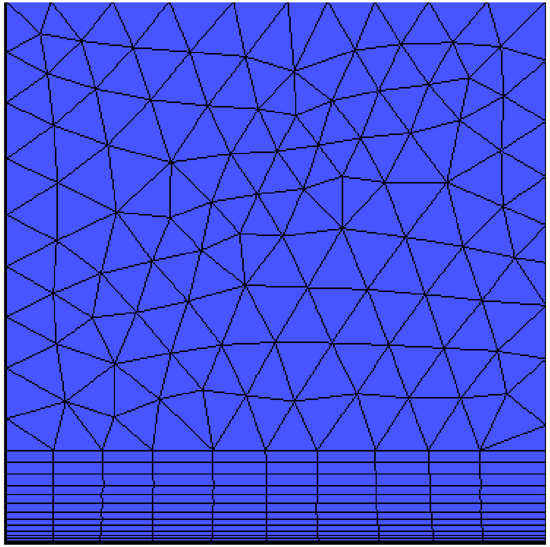
- ***Compute params*(参数组) will calculate the remaining parameter**

- **Or specify only *Height ratio* and *Number of layers* for auto calculation of initial height**

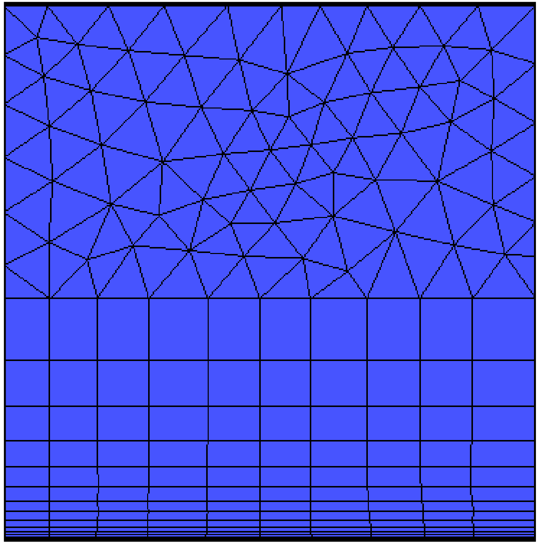
- **Individual surface/curve height/ratio/layers will override these global defaults if set**

Growth Law Comparison

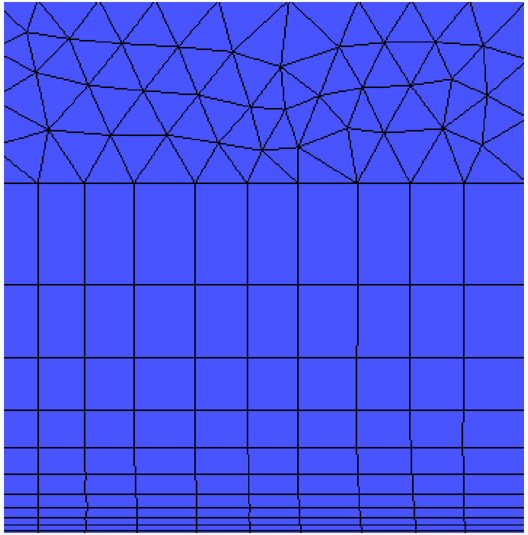
The growth rate of *wb-exponential* is greater than *exponential*
The growth rate of *exponential* is greater than *linear*



Linear

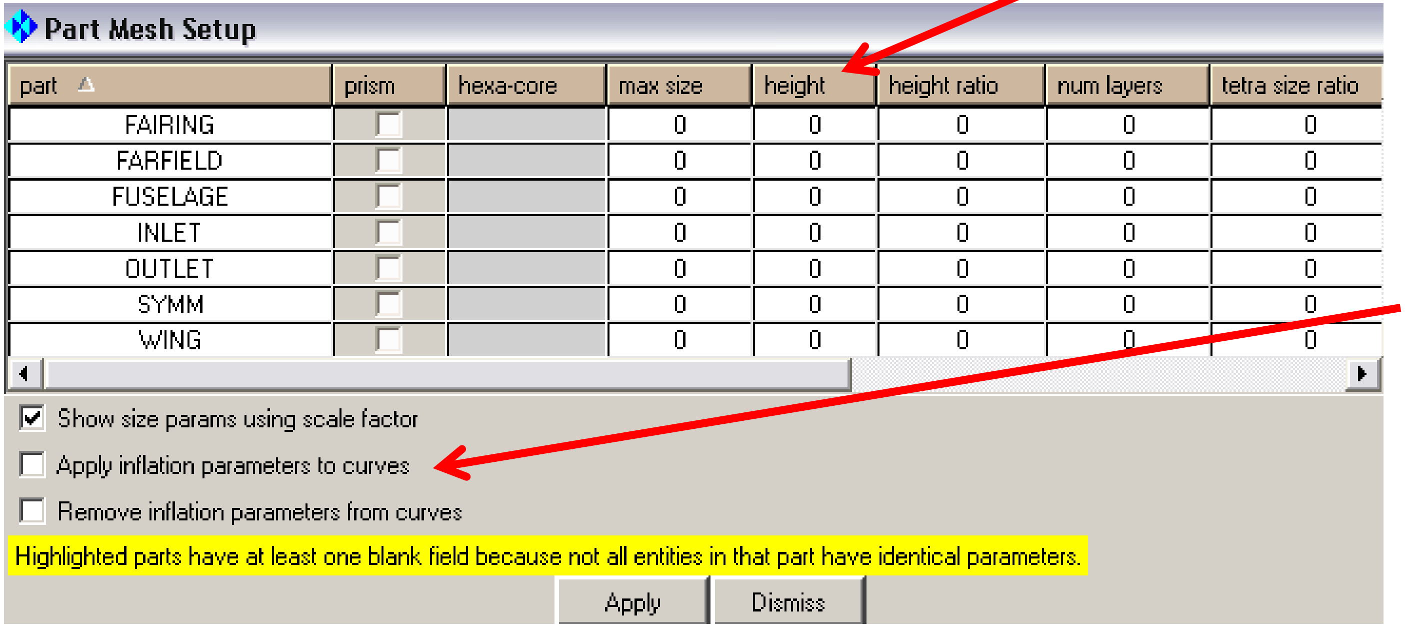


Exponential



Wb-exponential

• Setting Prism Parameters on Parts


Part Mesh Setup

part	prism	hexa-core	max size	height	height ratio	num layers	tetra size ratio
FAIRING	☐		0	0	0	0	0
FARFIELD	☐		0	0	0	0	0
FUSELAGE	☐		0	0	0	0	0
INLET	☐		0	0	0	0	0
OUTLET	☐		0	0	0	0	0
SYMM	☐		0	0	0	0	0
WING	☐		0	0	0	0	0

☑ Show size params using scale factor

☐ Apply inflation parameters to curves

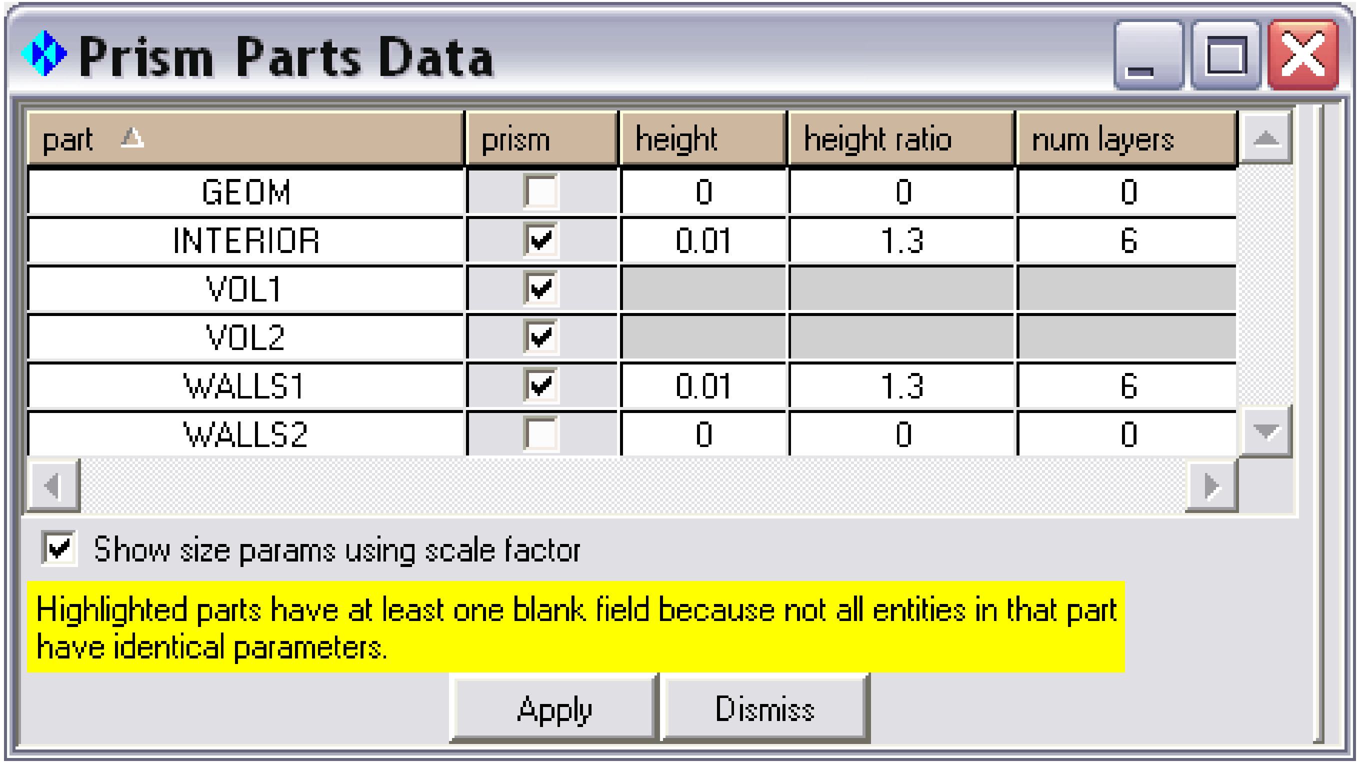
☐ Remove inflation parameters from curves

Highlighted parts have at least one blank field because not all entities in that part have identical parameters.

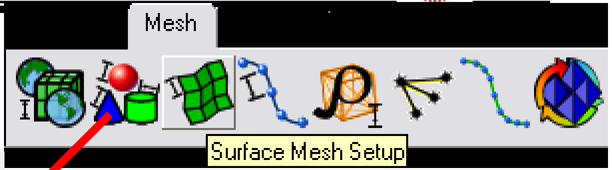
Apply Dismiss

If *Apply inflation parameters to curves* is toggled on, they will also be set on each curve within each part

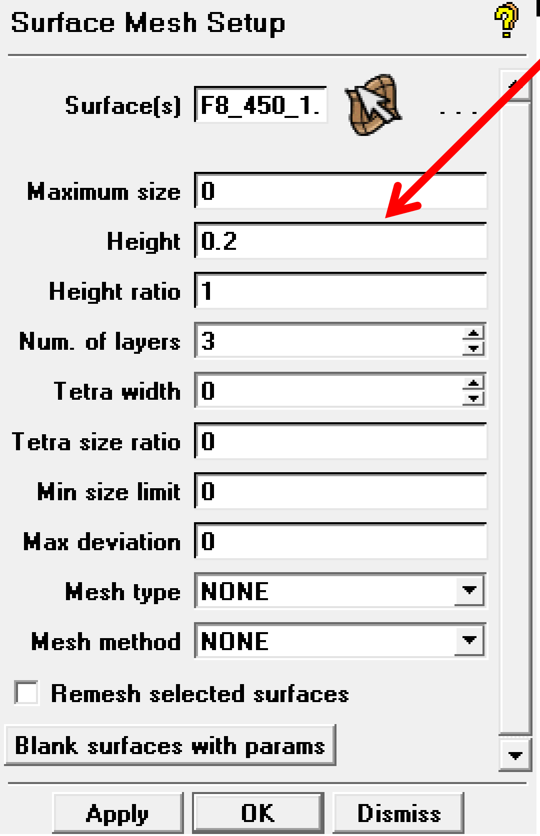
- **Setting Prism Parameters on Volume Parts**



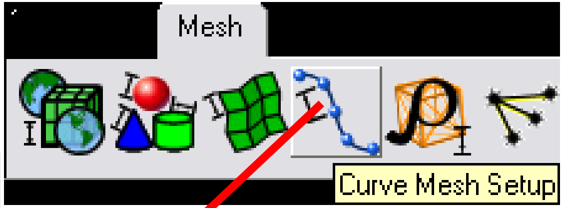
- **Setting Prism Parameters on Surfaces**



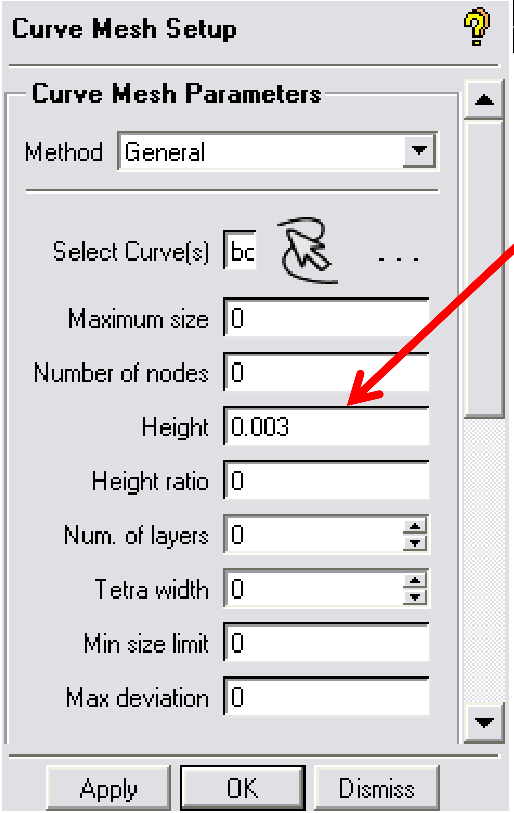
Mesh >
Surface Mesh Setup



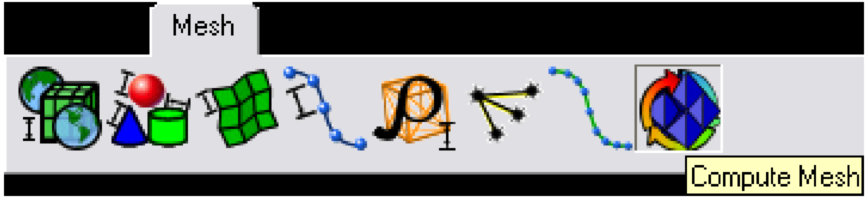
• Setting Prism Parameters on Curves



Mesh > Curve Mesh Setup

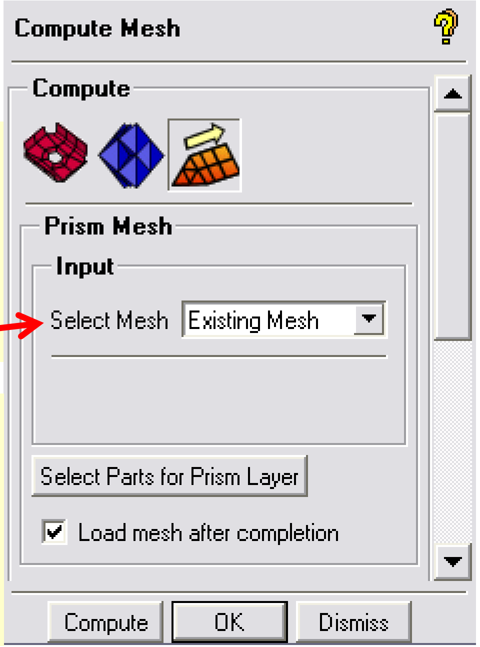


Run Prism



- **Can run separately**

Mesh > Compute Mesh > Prism Mesh



- **The *Select Parts for Prism Layer* button pops up (弹出)the same menu as the *Part Mesh Setup*, except non-prism related columns aren't displayed**

- **Input**
  - *-Existing Mesh*
  - *-From File*

12.5.4 Examples to generate structural grid

Auto-meshing algorithms for unstructural grid may make it easier but the grids they create may lack key qualities.

Reasons for taking the time to create structured grids in CFD

① **High Degree of Quality & Control**

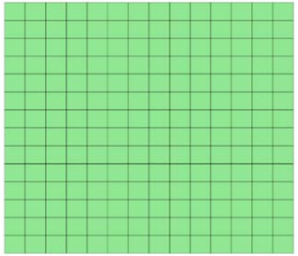
② **Better Alignment: Better Convergence**

③ **Less Memory and Time Required**

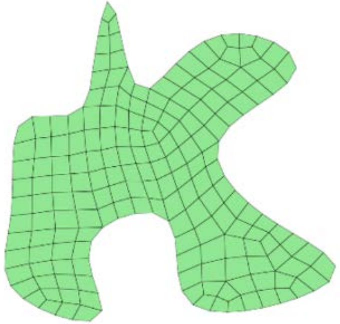
④ **The Data Locality Issue**

⑤ **It Has Available Solution Algorithms**

⑥ **Definable Normals**



Structured mesh



Unstructured mesh

The structured mesh generation procedure:

1.Create/Import geometry.

2.Initialize blocking with respect to geometry dimension

3.Generate block structure using the split, merge, O-grid definition.

4.Associate vertices to points, edges to curve and block faces to geometry face.

5.Check block structure quality to ensure the block model meets specified quality threshold.

6.Determine edge meshing parameters and using spacing 1 or spacing 2 for increasing mesh density in specific zone.

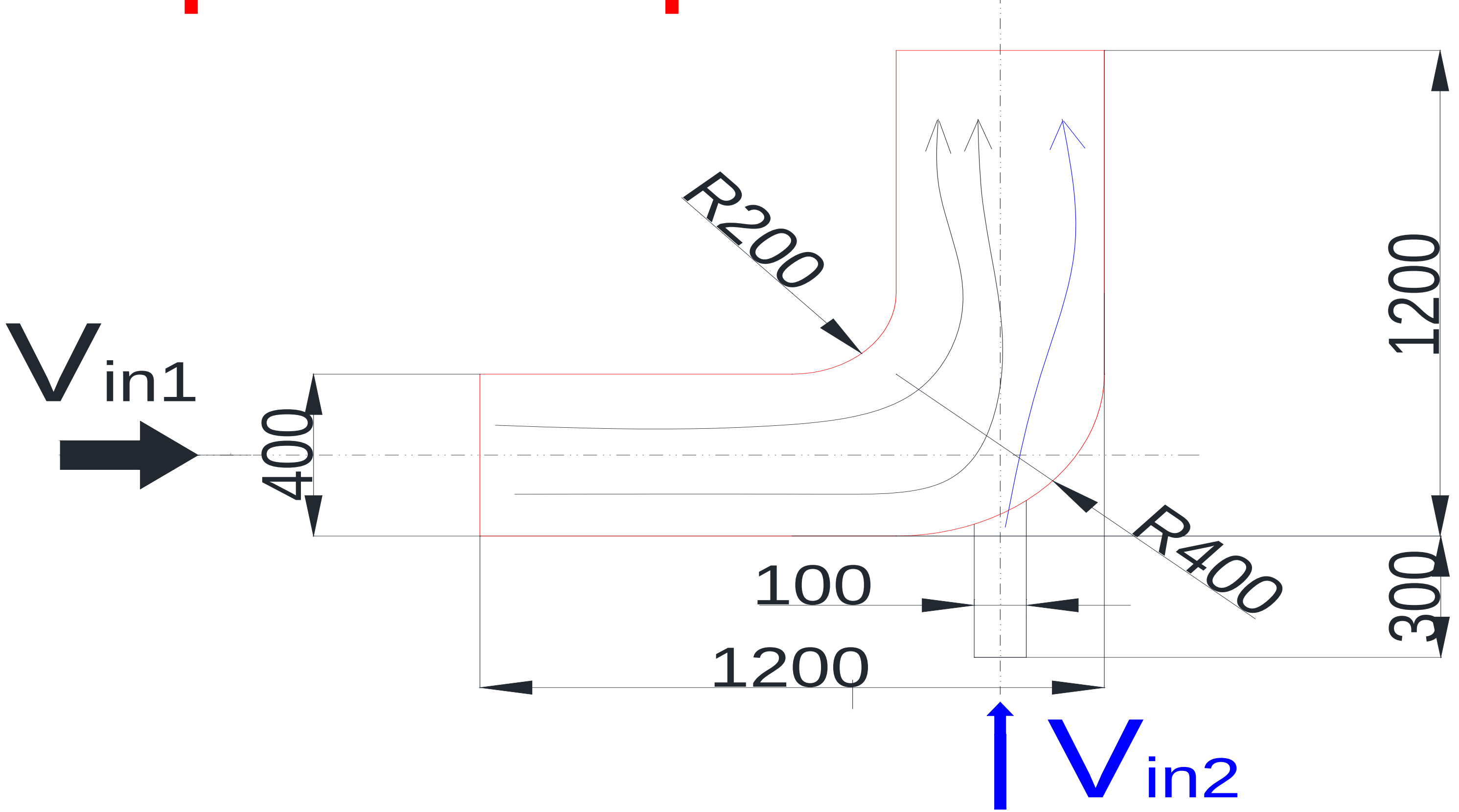
7.Using pre-mesh to update mesh.

8.Check the cell quality of the mesh once its generated.

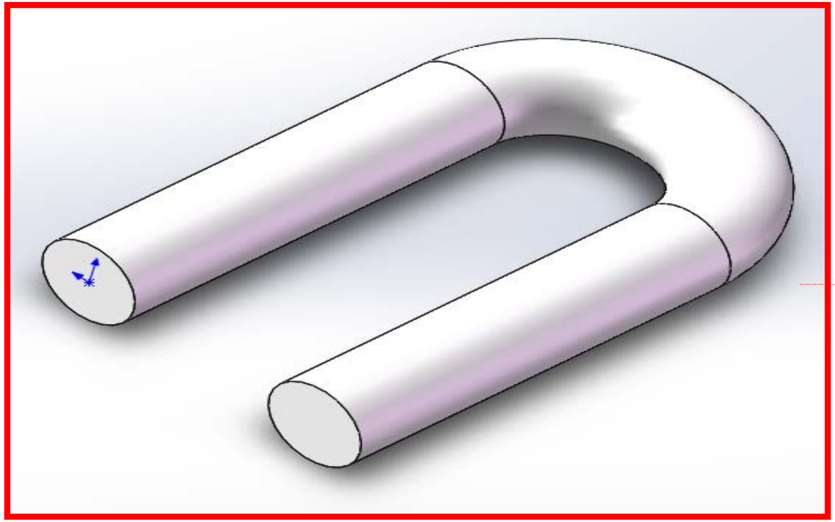
9.Convert structure mesh to substructure mesh by right click on the re-compute mesh.

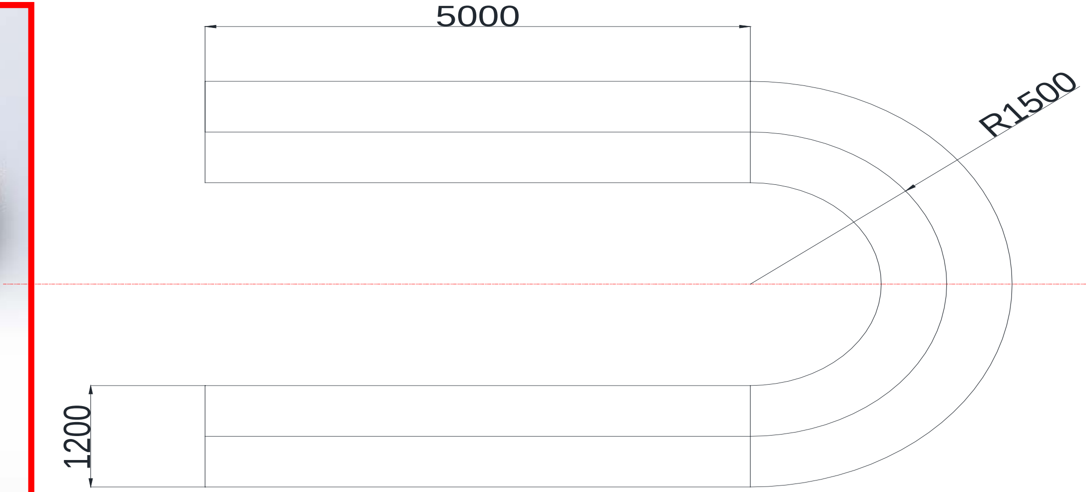
10.Write output files to desired solver like fluent.

Example 1: 2D Pipe Junction

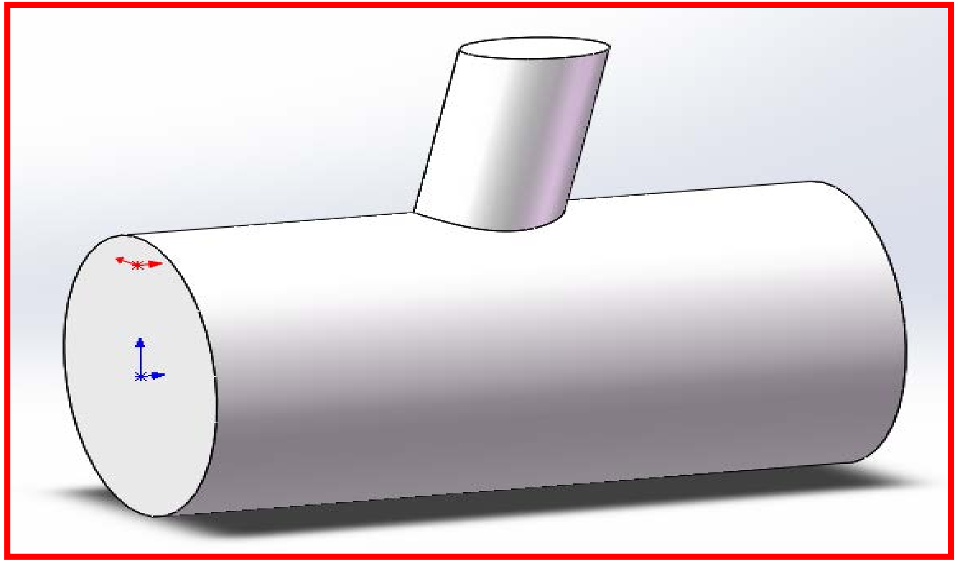


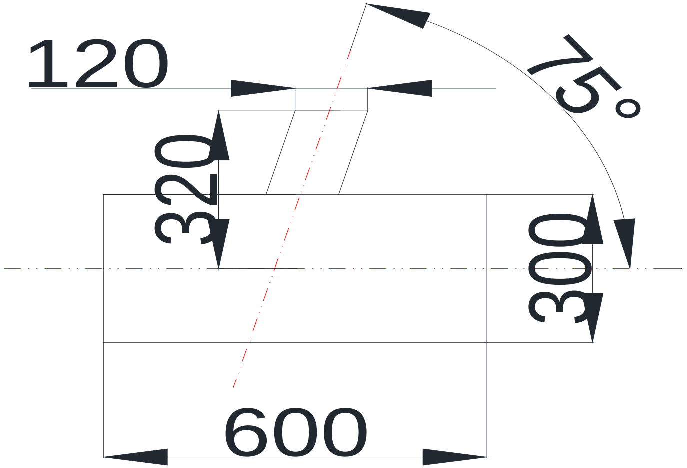
Example 2：Flow in a U turn



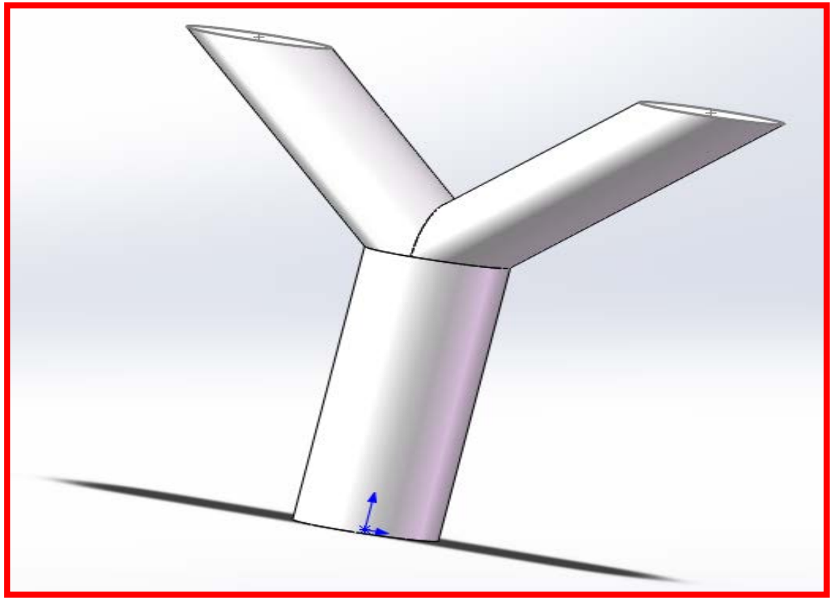


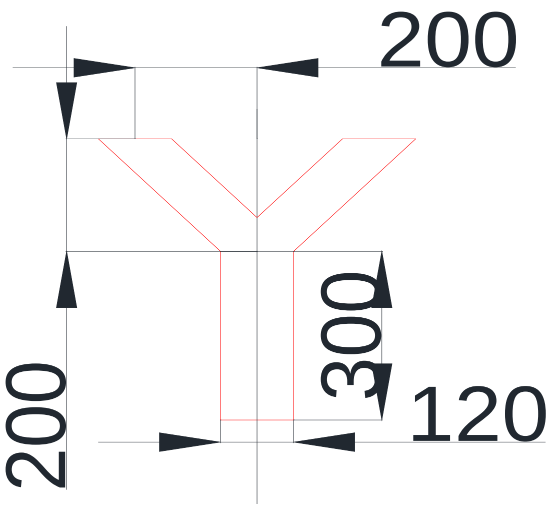
Example 3: Three pipe junction





Example 4 Flow in a “Y” tube





Next Class:

Fundamental:

Prof. Ren Qinlong(任秦龙)

Intermediate:

Prof. Chen Li(陈黎)

Thanks very much!

谢谢各位!

同舟共济
渡彼岸!