Customer Training Material

Lecture 2

Introduction to CFD Methodology

Introduction to ANSYS FLUENT

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

Release 13.0
December 2010

What is CFD?

- **Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations**
  - Conservation of mass
  - Conservation of momentum
  - Conservation of energy
  - Conservation of species
  - Effects of body forces

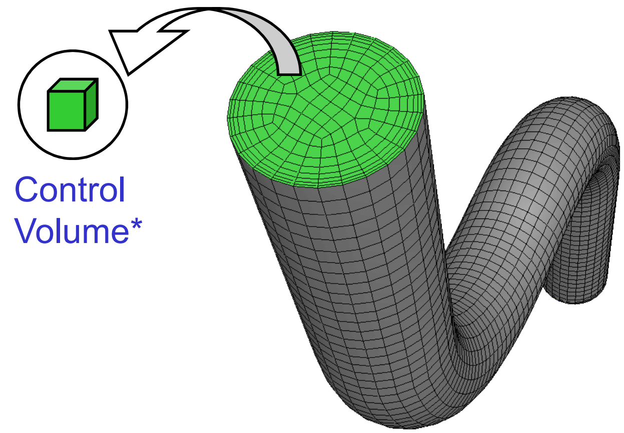
- **The results of CFD analyses are relevant in:**
  - Conceptual studies of new designs
  - Detailed product development
  - Troubleshooting
  - Redesign

- **CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.**

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

How Does CFD Work?

- **ANSYS CFD solvers are based on the finite volume method**
  - Domain is discretised into a finite set of control volumes
  - General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes



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

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi \, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi \mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi \, dV}_{\text{Generation}}$$

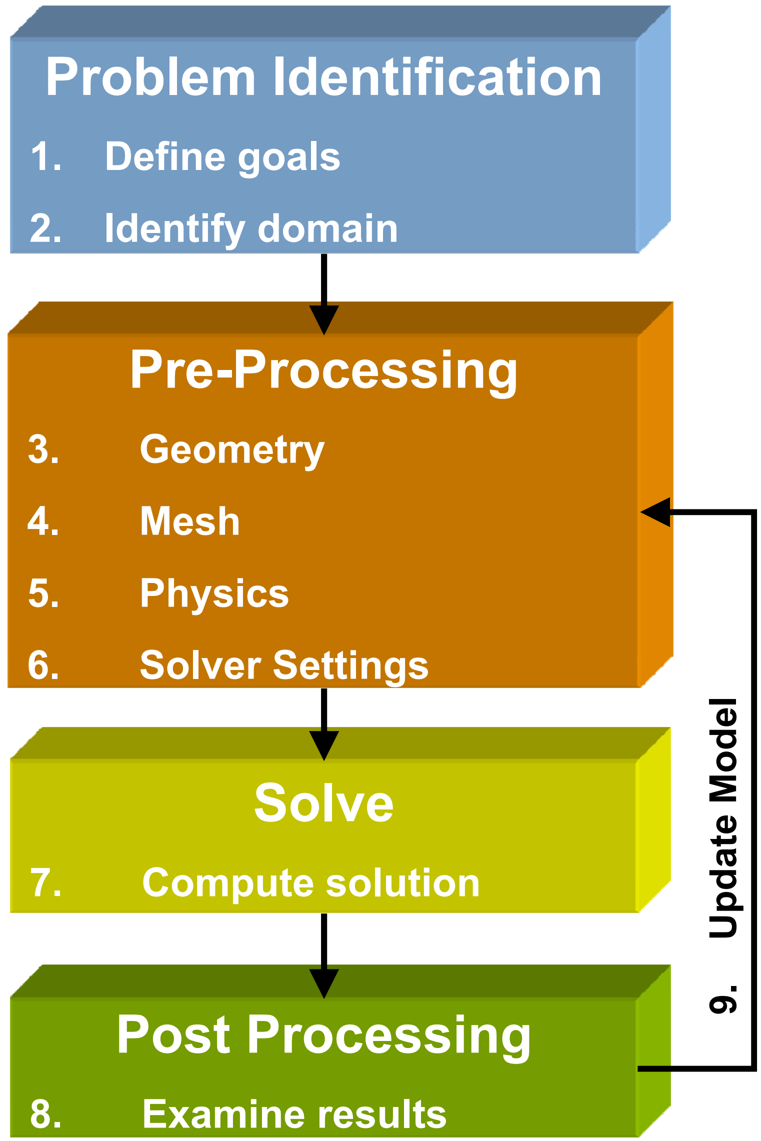
Equation	Variable
Continuity	1
X momentum	*u*
Y momentum	*v*
Z momentum	*w*
Energy	*h*

  - Partial differential equations are discretised into a system of algebraic equations
  - All algebraic equations are then solved numerically to render the solution field

* FLUENT control volumes are cell-centered (i.e. they correspond directly with the mesh) while CFX control volumes are node-centered

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

CFD Modeling Overview



© 2010 ANSYS, Inc. All rights reserved.

1. Define Your Modeling Goals

Problem Identification

1. *Define goals*
2. Identify domain

- **What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?**
  - What are your modeling options?
    - What physical models will need to be included in your analysis
    - What simplifying assumptions do you **have to make**?
    - What simplifying assumptions **can you make** (i.e. symmetry, periodicity)?
- **What degree of accuracy is required?**
- **How quickly do you need the results?**
- **Is CFD an appropriate tool?**

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

2. Identify the Domain You Will Model

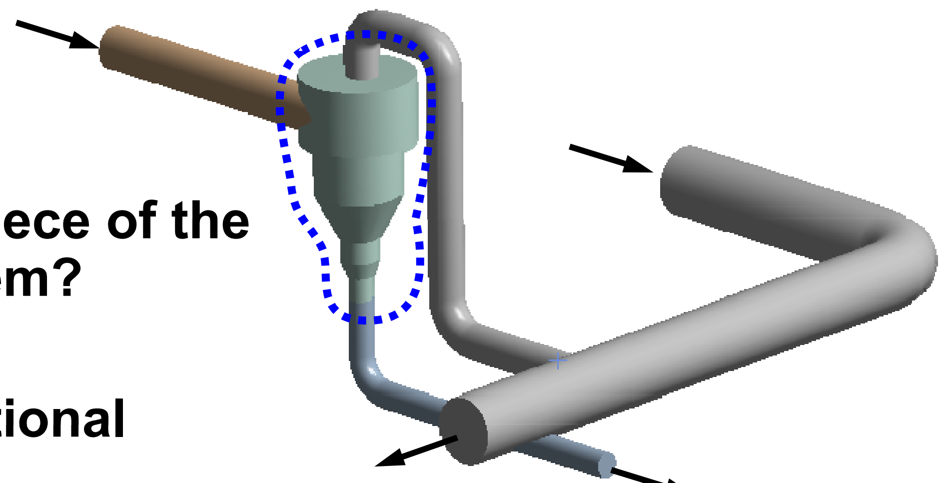
Problem Identification

1. *Define goals*
2. Identify domain

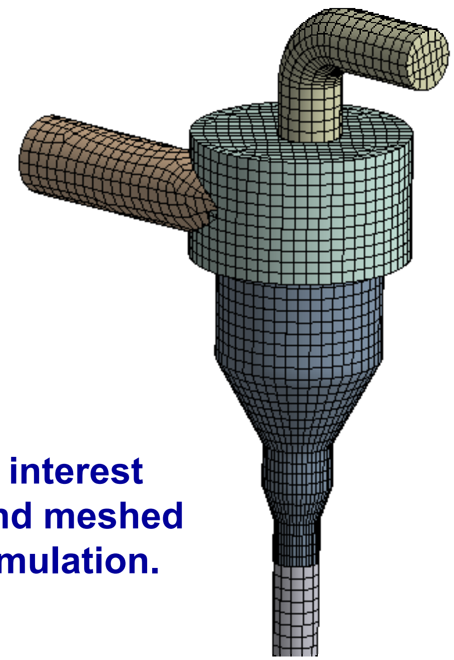
- **How will you isolate a piece of the complete physical system?**

- **Where will the computational domain begin and end?**
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?

- **Can it be simplified or approximated as a 2D or axisymmetric problem?**



Domain of Interest as Part of a Larger System (not modeled)



Domain of interest isolated and meshed for CFD simulation.

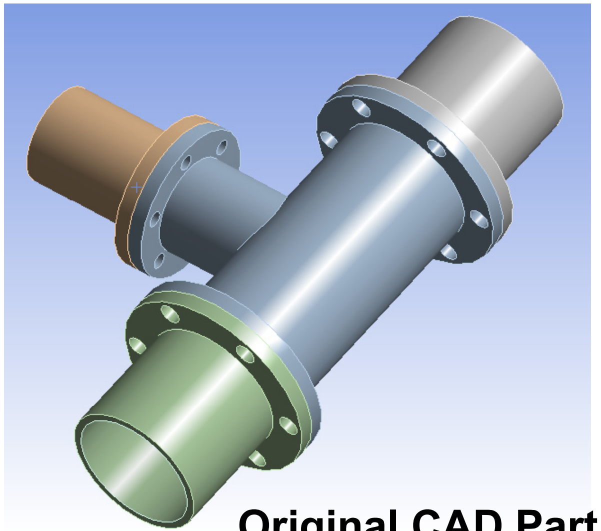
© 2010 ANSYS, Inc. All rights reserved.

3. Create a Solid Model of the Domain

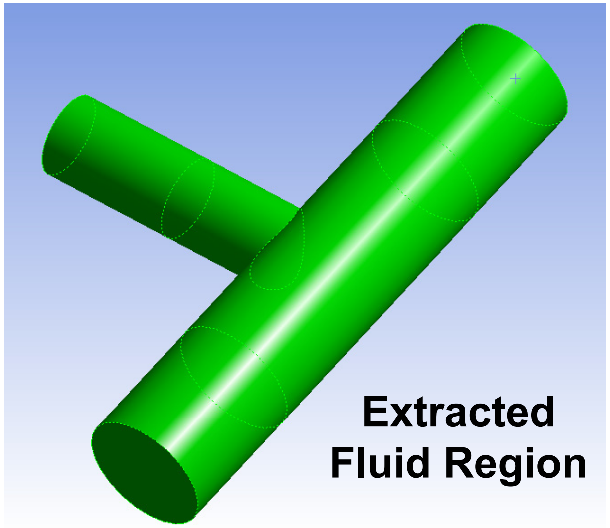
Pre-Processing

3. *Geometry*
4. Mesh
5. Physics
6. Solver Settings

- **How will you obtain a model of the *fluid* region?**
  - Make use of existing CAD models?
  - Extract the fluid region from a solid part?
  - Create from scratch?
- **Can you simplify the geometry?**
  - Remove unnecessary features that would complicate meshing (fillets, bolts…)?
  - Make use of symmetry or periodicity?
    - Are both the solution and boundary conditions symmetric / periodic?
- **Do you need to split the model so that boundary conditions or domains can be created?**



Original CAD Part



Extracted Fluid Region

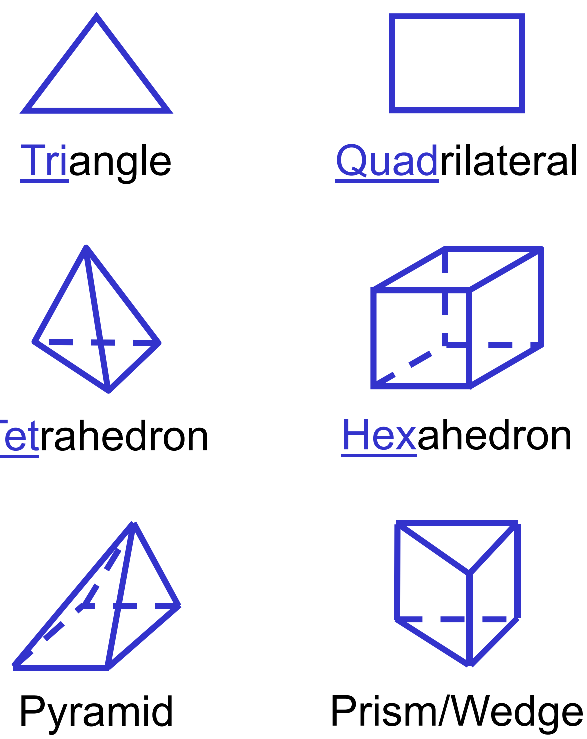
© 2010 ANSYS, Inc. All rights reserved.

4. Design and Create the Mesh

Pre-Processing

3. **Geometry**
4. ***Meshing***
5. **Physics**
6. **Solver Settings**

A mesh divides a geometry into many elements. These are used by the CFD solver to construct control volumes



- **What degree of mesh resolution is required in each region of the domain?**
  - The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients
  - Can you predict regions of high gradients?
  - Will you use adaption to add resolution?

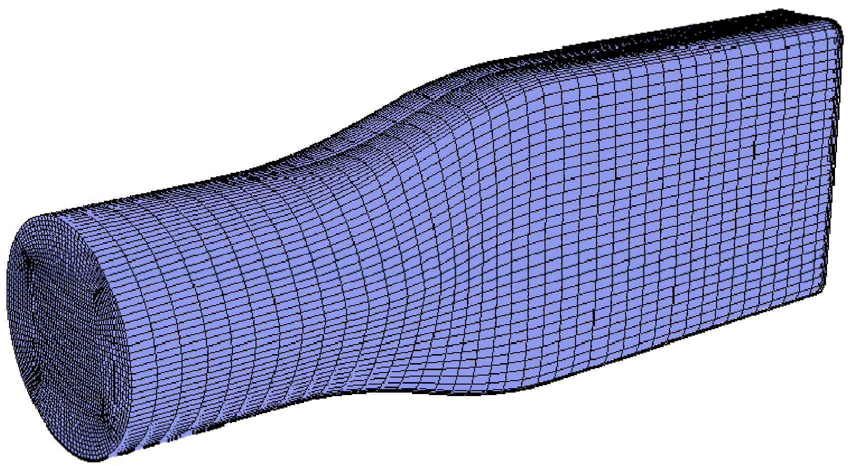
- **Do you have sufficient computer resources?**
  - How many cells/nodes are required?
  - How many physical models will be used?

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

Tri/Tet vs. Quad/Hex Meshes

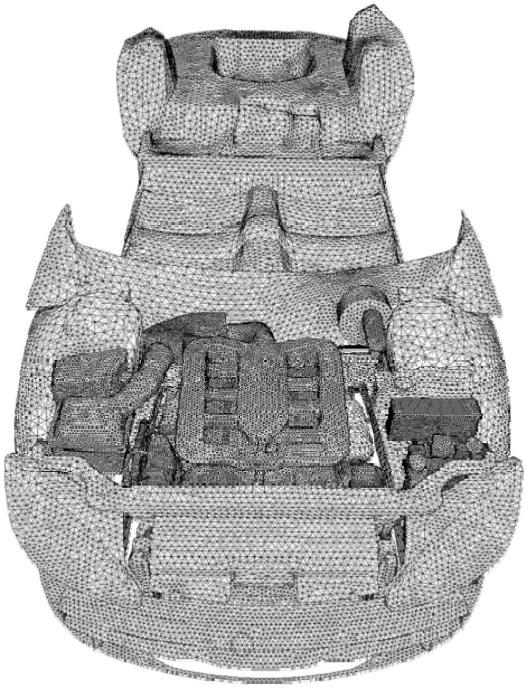
For flow-aligned geometries:

- Quad/hex meshes can provide higher-quality solutions with fewer cells/nodes than a comparable tri/tet mesh
- Quad/hex meshes show reduced numerical diffusion when the mesh is aligned with the flow.
- It does require more effort to generate a quad/hex mesh



© 2010 ANSYS, Inc. All rights reserved.

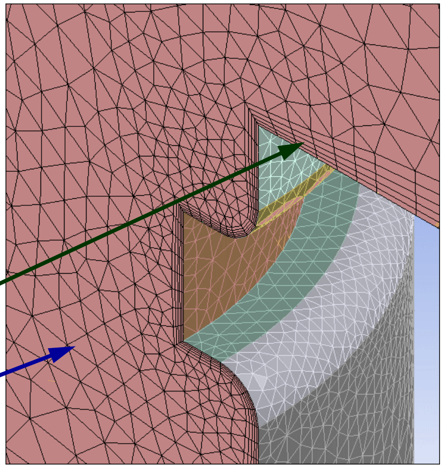
Tri/Tet vs. Quad/Hex Meshes



For complex geometries:

- It would be impractical to generate a structured (flow-aligned) hex mesh.
- You can save meshing effort by using a tri/tet mesh or hybrid mesh
- Quick to generate

- Hybrid meshes typically combine tri/tet elements with other elements in selected regions
  - For example, use wedge/ prism elements to resolve boundary layers.
  - More efficient and accurate than tri/tet alone.

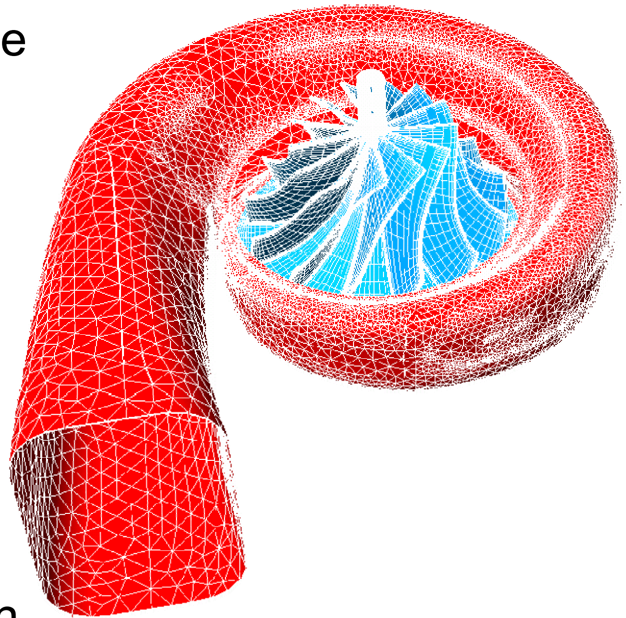


© 2010 ANSYS, Inc. All rights reserved.

Non-Conformal Meshes

Non conformal meshes:

- Usually when meshing, where two volumes meet, the mesh should exactly match (conformal mesh).
- This will be the case if, in ANSYS DesignModeler all the bodies are combined to form **1 part.**
- If you have **multiple parts** the mesh will not match, and in FLUENT you **MUST** set up a non-conformal interface to pair the surfaces.
- Typical scenarios for using non-conformals are when meshing very complex geometries and for sliding mesh applications.



Compressor and Scroll
The compressor and scroll are joined through a non conformal interface. This serves to connect the hex and tet meshes and also allows a change in reference frame

© 2010 ANSYS, Inc. All rights reserved.

Set Up the Physics and Solver Settings

Pre-Processing

3. **Geometry**
4. **Mesh**
5. ***Physics***
6. ***Solver Settings***

For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time.

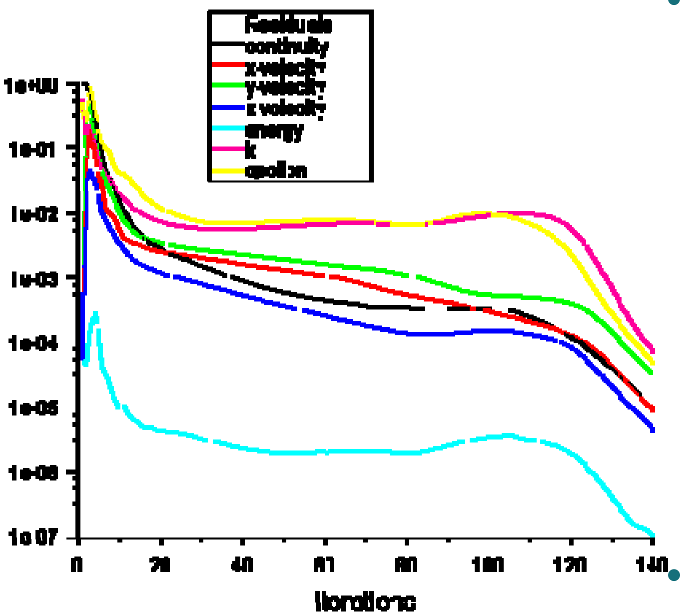
- **For a given problem, you will need to:**
  - Define material properties
    - Fluid
    - Solid
    - Mixture
  - Select appropriate physical models
    - Turbulence, combustion, multiphase, etc.
  - Prescribe operating conditions
  - Prescribe boundary conditions at all boundary zones
  - Provide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

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

Compute the Solution

Solve

7. Compute solution



A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

- **The discretised conservation equations are solved iteratively until convergence.**

- **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
    - Imbalances measure global conservation
  - Quantities of interest (e.g. drag, pressure drop) have reached steady values.
    - Monitor points track quantities of interest.

- **The accuracy of a *converged* solution is dependent upon:**
  - Appropriateness and accuracy of physical models.
  - Mesh resolution and independence
  - Numerical errors

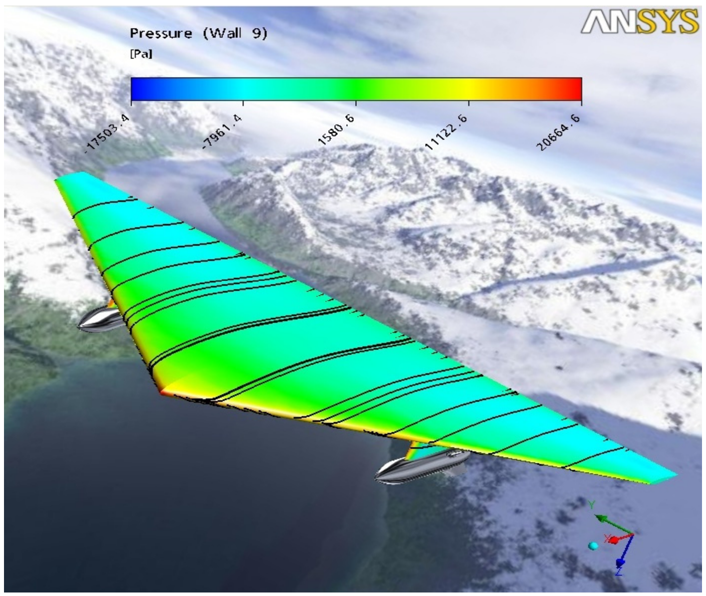
ANSYS, Inc. Proprietary
© 2010 ANSYS, Inc. All rights reserved.

Examine the Results

Post Processing

8. *Examine results*

9. Update Model

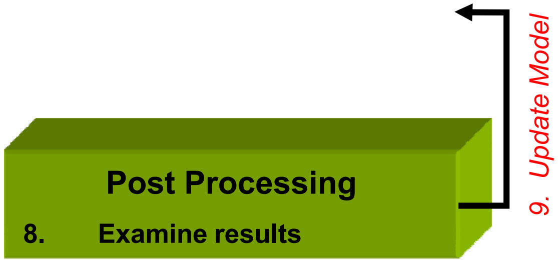


Examine results to ensure property conservation and correct physical behavior. High residuals may be caused by just a few poor quality cells.

- **Examine the results to review solution and extract useful data**
  - Visualization Tools can be used to answer such questions as:
    - What is the overall flow pattern?
    - Is there separation?
    - Where do shocks, shear layers, etc. form?
    - Are key flow features being resolved?
  - Numerical Reporting Tools can be used to calculate quantitative results:
    - Forces and Moments
    - Average heat transfer coefficients
    - Surface and Volume integrated quantities
    - Flux Balances

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

Consider Revisions to the Model



- **Are the physical models appropriate?**
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?

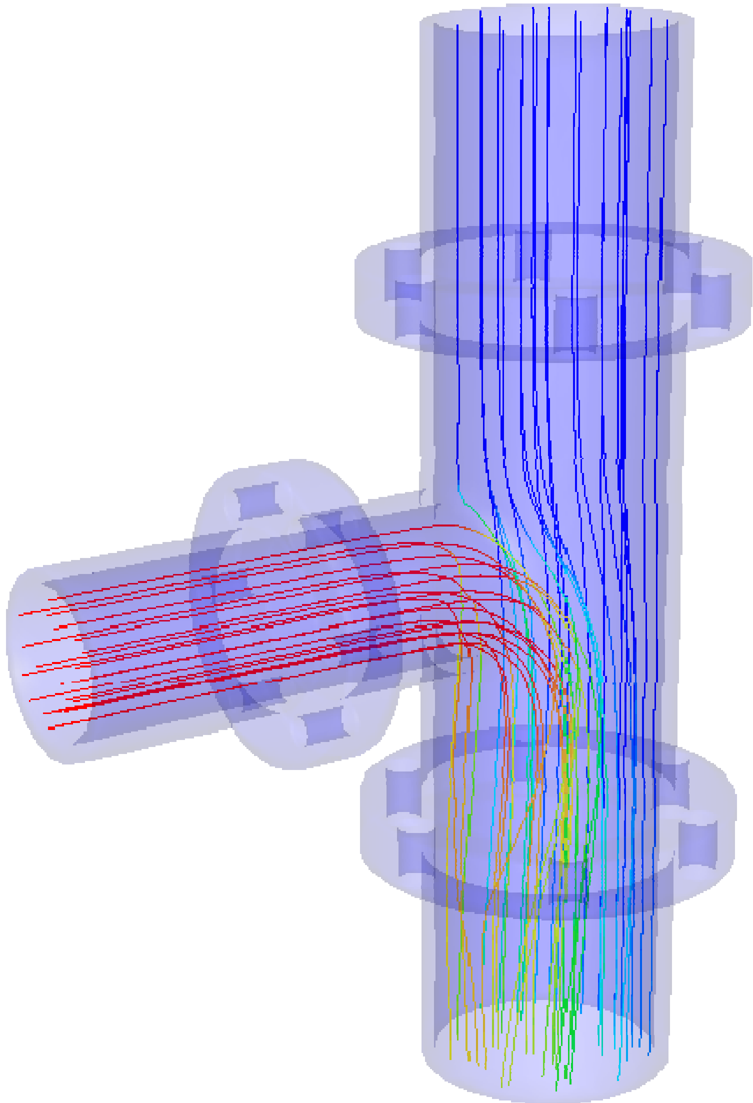
- **Are the boundary conditions correct?**
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?

- **Is the mesh adequate?**
  - Can the mesh be refined to improve results?
  - Does the solution change significantly with a refined mesh, or is the solution mesh independent?
  - Does the mesh resolution of the geometry need to be improved?

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

Demonstration of FLUENT Software

- **Start FLUENT (assume the mesh has already been generated).**
  - Set up a simple problem.
  - Solve the flow field.
  - Postprocess the results.



© 2010 ANSYS, Inc. All rights reserved.

Using our Training Machines

- Log in to your workstation
  - Login name: `training`
  - Password: `training`

- Directories
  - Workshop mesh/case/data files can be found in `c:\Users\Training_Materials\originals\course_name`
  - We recommend that you save your work into a central working folder: `c:\Users\Training\your_name`

- To start FLUENT and/or Workbench, use the desktop icons.

- It is recommended that you restart FLUENT and/or Workbench for each tutorial to avoid mixing solver settings from different workshops.

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