



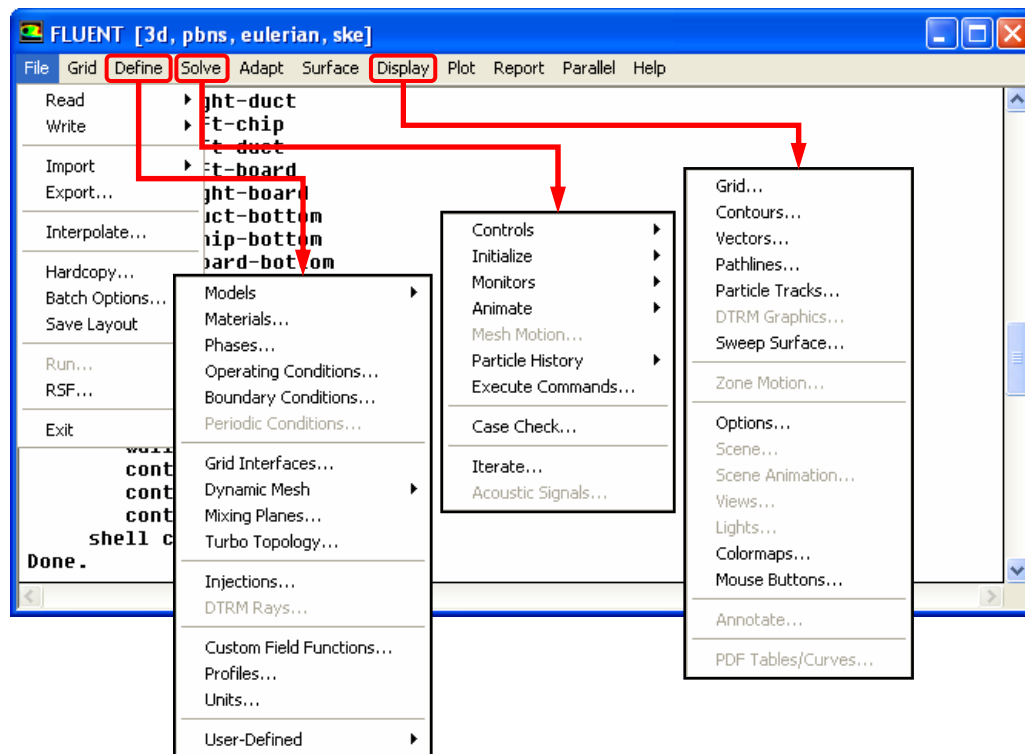
Solver Basics

Introductory FLUENT Training



Solver Execution

- ◆ The menus are arranged such that the order of operation is generally left to right.
 - Read and scale the mesh file.
 - Select physical models.
 - Define material properties.
 - Prescribe operating conditions.
 - Prescribe boundary conditions.
 - Provide an initial solution.
 - Set solver controls.
 - Set up convergence monitors.
 - Compute and monitor solution.
- ◆ Postprocessing
 - Feedback into the solver
 - Engineering analysis



Text User Interface

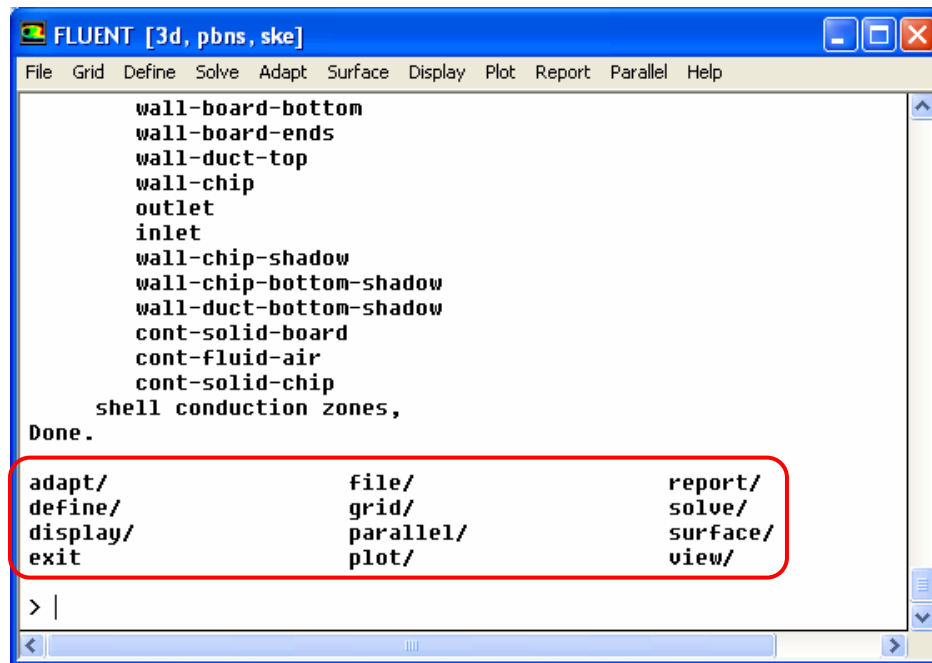
- ◆ All GUI commands have a corresponding TUI command.
 - Advanced commands are only available through TUI.
 - Press the **Enter** key to display the command set at the current level.
 - **q** moves up one level.

- ◆ FLUENT can be run in batch mode or scripted using a journal file.

File → Write → Start Journal...

File → Write → Stop Journal...

File → Read → Journal...



- ◆ A TUI user guide is available on the FLUENT User Services Center.

www.fluentusers.com

Using TUI commands

- ◆ Some common operations are aliased to simple TUI commands:
 - **ls** Lists the files in the working directory
 - **rcd** Reads case and data files
 - **wcd** Writes case and data files
 - **rc/wc** Reads/writes case file
 - **rd/wd** Reads/writes data file
 - **it** Iterate
- ◆ TUI commands in a batch file can be used to automate operations in a non-interactive mode.
- ◆ The TUI commands **file/read-bc** and **file/write-bc** can be used for reading and writing the settings for a FLUENT session to and from a file, respectively.
- ◆ TUI commands can be used to toggle various solver settings.
- ◆ Add the **.gz** extension to case and data files to write in compressed format and to decompress upon reading in FLUENT.

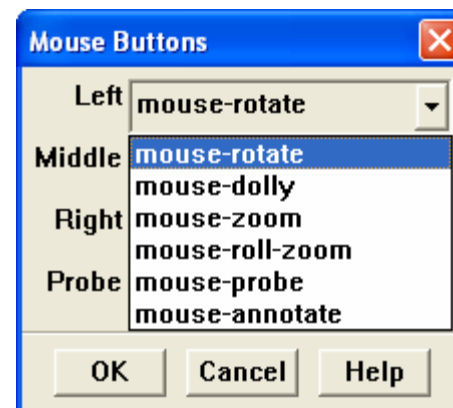
Sample Batch File

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

Mouse Functionality

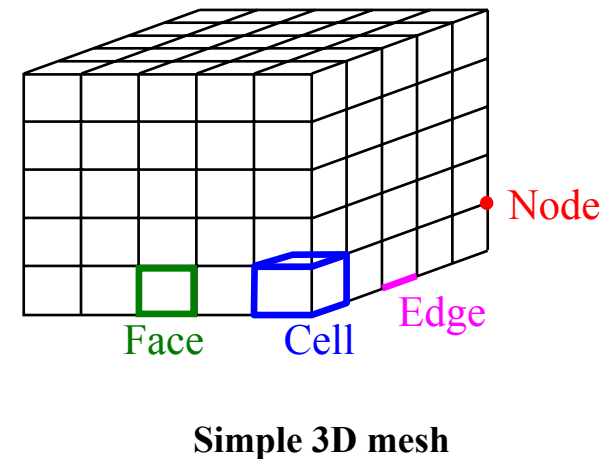
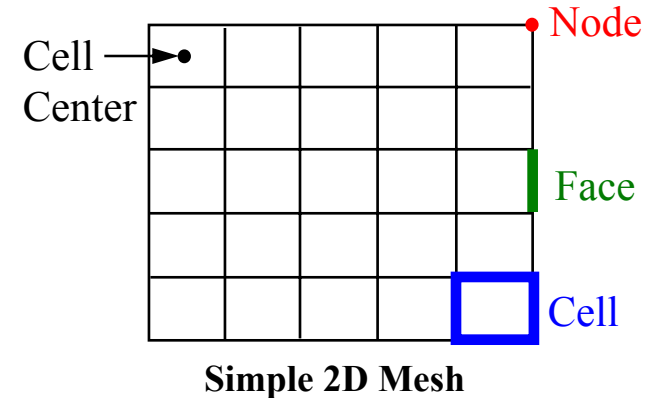
- ◆ Mouse button functionality depends on the chosen solver (2D / 3D) and can be configured in the solver.
- ◆ Default settings
 - 2D Solver
 - Left button translates/pans (dolly)
 - Middle button zooms
 - Right button selects/probes
 - 3D Solver
 - Left button rotates about 2 axes
 - Middle button zooms
 - ◆ Middle click on point in screen centers point in window
 - Right button selects/probes
- ◆ Retrieve detailed flow field information at point with Probe enabled.
 - Right-click on the graphics display.

Display → Mouse Buttons...

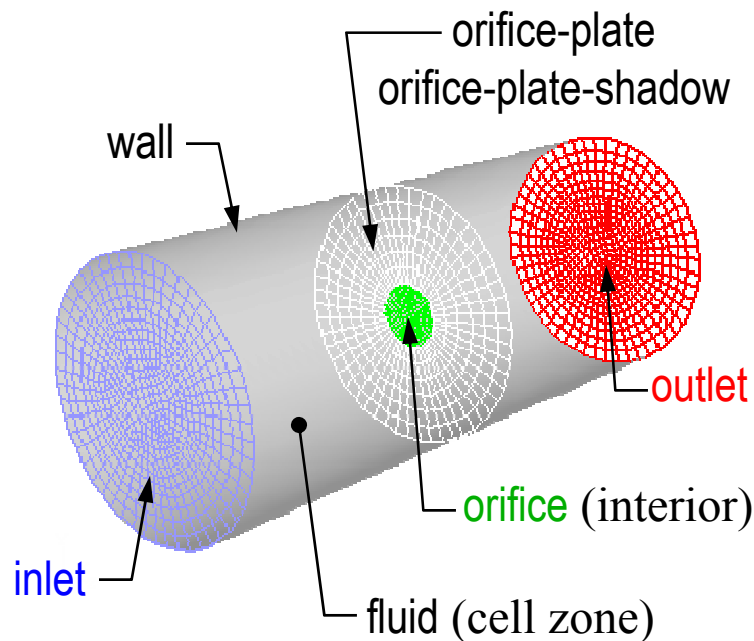


Reading the Mesh – Components

- ◆ Components are defined in the preprocessor and stored in the mesh file.
 - **Cell** – The control volumes into which the domain is discretized.
 - Computational domain is defined by mesh that represents the fluid and solid regions of interest.
 - **Face** – The boundaries of cells
 - **Edge** – Boundary of a face
 - **Node** – Edge intersection / grid point
 - **Zone** – Grouping of nodes, faces, and/or cells.
 - Boundary data is assigned to face zones.
 - Material data and source terms are assigned to cell zones.



Reading the Mesh – Zones



FLUENT [3d, pbns, ske]

File Grid Define Solve Adapt Surface Display Plot Report Parallel Help

```

441 mixed velocity-inlet faces, zone 7.
38238 mixed interior faces, zone 9.
13230 hexahedral cells, zone 2.

Building...
grid,
materials,
interface,
domains,
zones,
  default-interior
inlet
outlet
orifice-plate
orifice
wall
fluid
  creating orifice-plate-s
shell conduction zones,
Done.
  
```

Define → Boundary Conditions...

Boundary Conditions

Zone	Type
default-interior	inlet-vent
fluid	intake-fan
inlet	interface
orifice	mass-flow-inlet
orifice-plate	outflow
orifice-plate-shadow	outlet-vent
outlet	pressure-far-field
wall	pressure-inlet
	pressure-outlet
	symmetry
	velocity-inlet
	wall

ID: 6

Set... Copy... Close Help

default-interior is the zone containing all internal cell faces (not used).

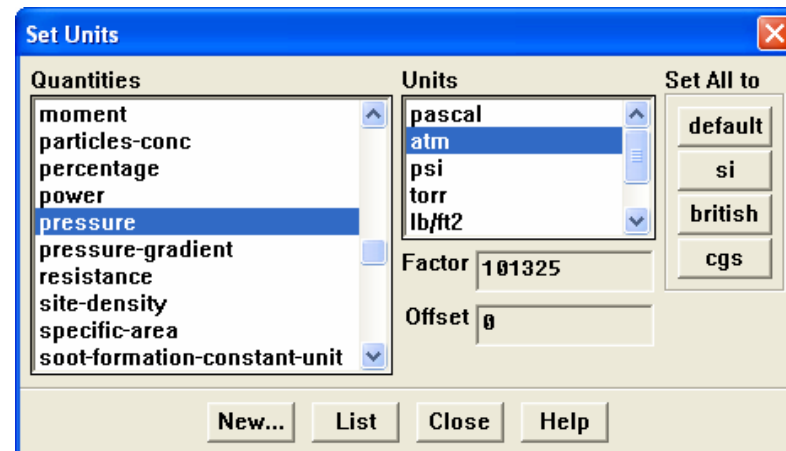
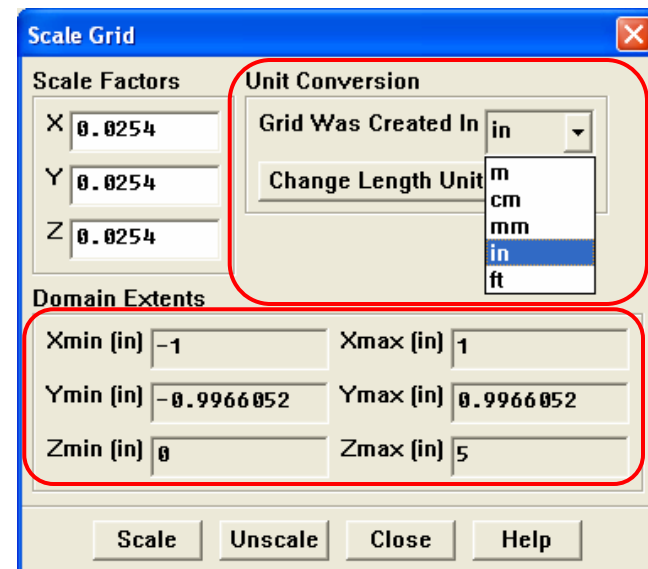
Scaling the Mesh and Selecting Units

- ◆ When FLUENT reads a mesh, all physical dimensions are assumed to be in units of meters.
 - If your model was not built in meters, then it must be scaled appropriately.
 - Verify that the Domain Extents are correct after scaling the mesh.

Grid → Scale...

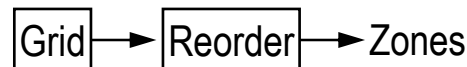
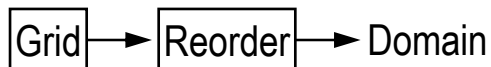
- ◆ Any “mixed” units system can be used if desired.
 - By default, FLUENT uses the SI system of units (specifically, MKS system).
 - Any units can be specified in the Set Units panel.

Define → Units...

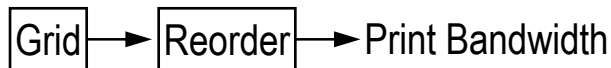


Reordering and Modifying the Grid

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



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

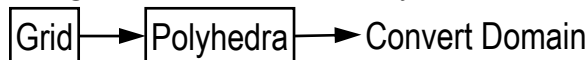


- ◆ The face/cell zones can also be modified by the following operations in the Grid menu:
 - Separation and merge of zones
 - Fusing of cell zones with merge of duplicate faces and nodes
 - Translate, rotate, reflect face or cell zones
 - Extrusion of face zones to extend the domain
 - Replace a cell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

Polyhedral Mesh Conversion

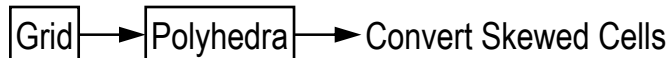
- ◆ A tetrahedral or hybrid grid can be converted in FLUENT 6.3 to polyhedra directly without going through a preprocessor.

- Advantages
 - Improved mesh quality.
 - Can reduce cell count significantly.
 - User has control of the conversion process.
- Disadvantages:
 - Cannot be adapted or converted again.
 - Cannot use tools such as smooth, swap, merge and extrude to modify the mesh.

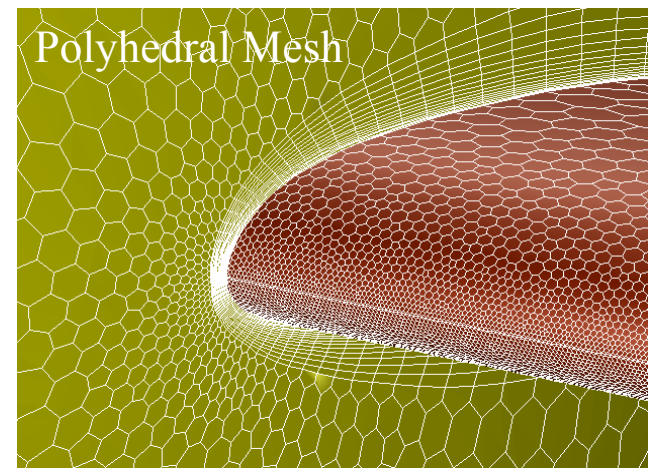
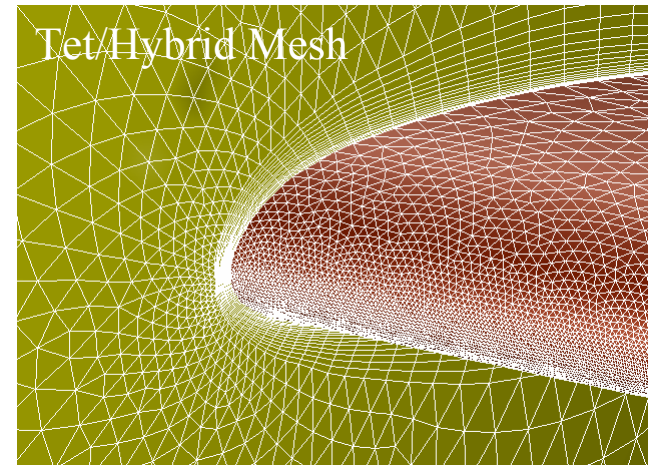


- ◆ Two conversion options are available in the Grid menu:

- Convert all cells in the domain (except hex cells) to polyhedra
 - Cannot convert meshes with hanging nodes
 - HexCore mesh can be converted using the **tpoly** standalone utility.
- Convert only highly skewed cells to polyhedra

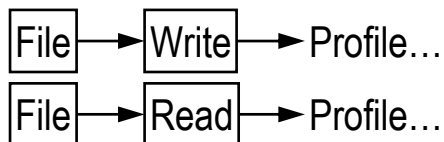


Converting Mesh for DPW3-W1 Wing

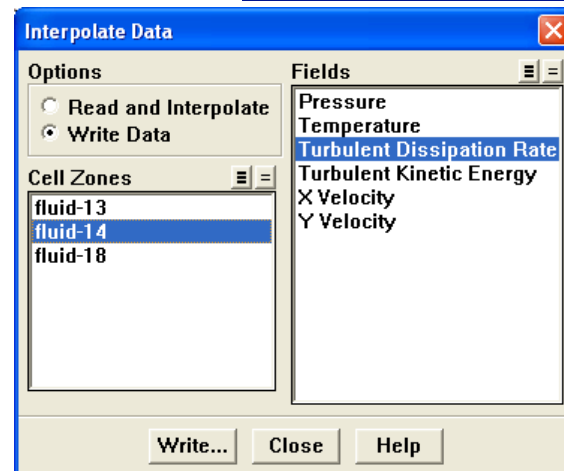
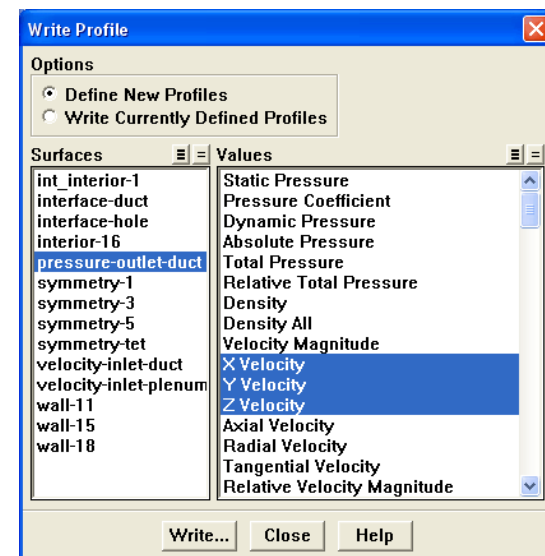


Profile Data and Solution Data Interpolation

- ◆ FLUENT allows interpolation of selected variable data on both face zones and cell zones by using profile files and data interpolation files, respectively.
 - For example, a velocity profile from experimental data or previous FLUENT run at an inlet, or a solution interpolated from coarse mesh to fine mesh.
- ◆ Profile files are data files which contain point data for selected variables on particular face zones, and can be both written and read in a FLUENT session.

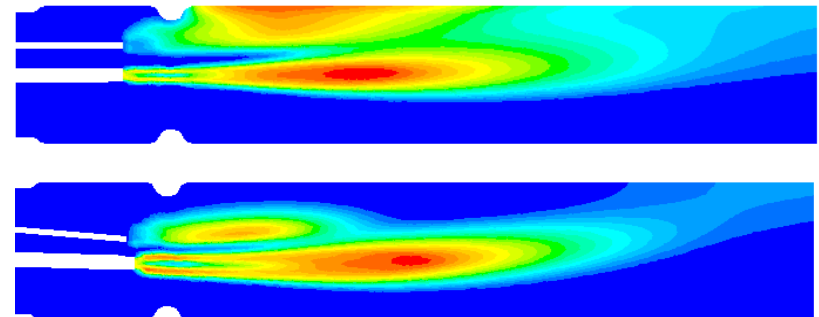
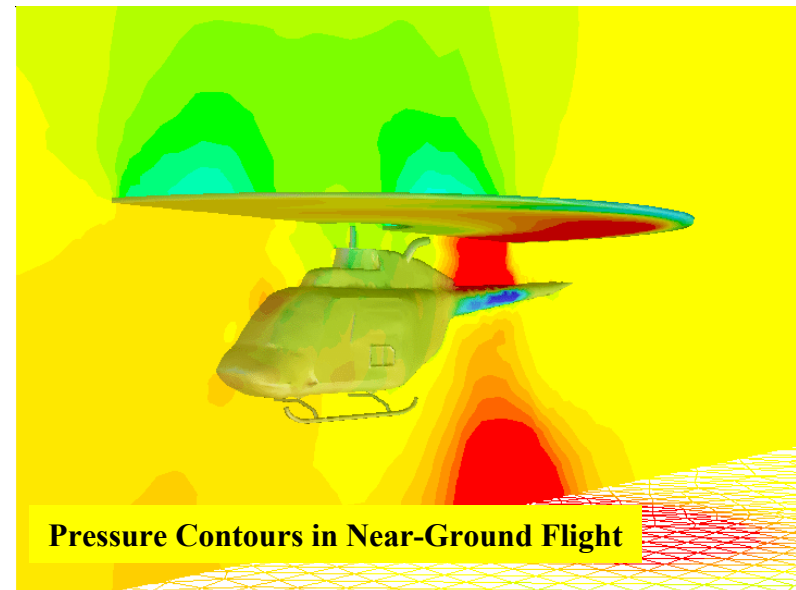


- ◆ Similarly, Interpolation data files contain discrete data for selected field variables on particular cell zones to be written and read into FLUENT.



Models Available in FLUENT 6

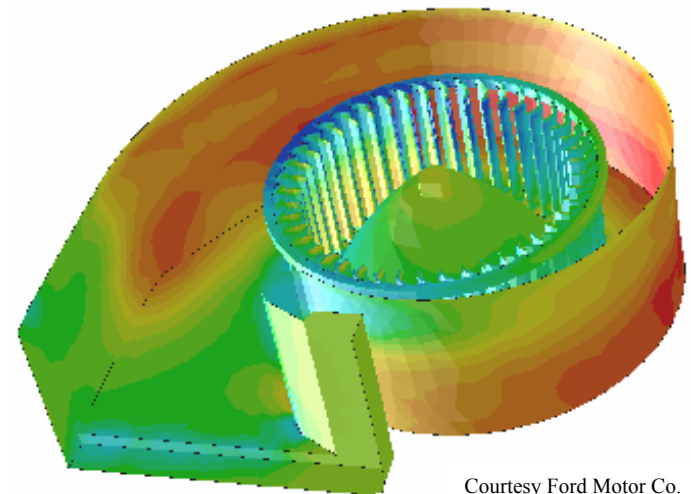
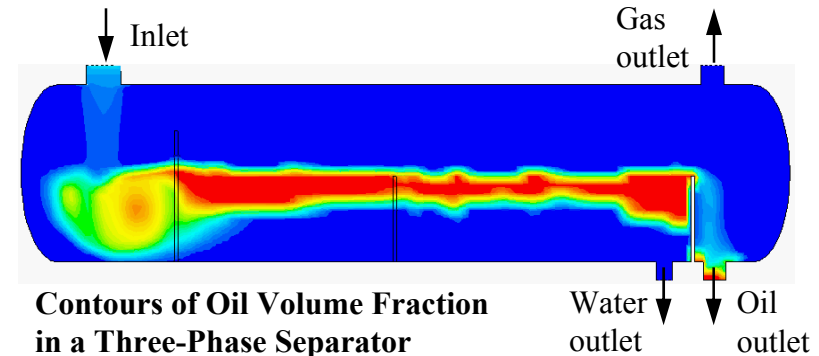
- ◆ Fluid flow and heat transfer
 - Momentum, continuity, energy equations
 - Radiation
- ◆ Turbulence
 - RANS-based models (Spalart-Allmaras, $k-\epsilon$, $k-\omega$, Reynolds stress)
 - Large-eddy simulation (LES) and detached eddy simulation (DES)
- ◆ Species transport
- ◆ Volumetric reactions
 - Arrhenius finite-rate chemistry
 - Turbulent fast chemistry
 - Eddy Dissipation, non-Premixed, premixed, partially premixed
 - Turbulent finite-rate chemistry
 - EDC, laminar flamelet, composition PDF transport
 - Surface Reactions



Temperature Contours for Kiln Burner Retrofit

Models Available in FLUENT 6

- ◆ **Multiphase flows**
 - Discrete Phase Model (DPM)
 - Volume of Fluid (VOF) model for immiscible fluids
 - Mixtures
 - Eulerian-Eulerian and Eulerian-granular
 - Liquid/Solid and cavitation phase change
- ◆ **Moving and deforming mesh**
 - Moving zones
 - Single and multiple reference frames (MRF)
 - Mixing plane model
 - Sliding mesh model
 - Moving and deforming (dynamic) mesh (MDM)
- ◆ **User-defined scalar transport equations**



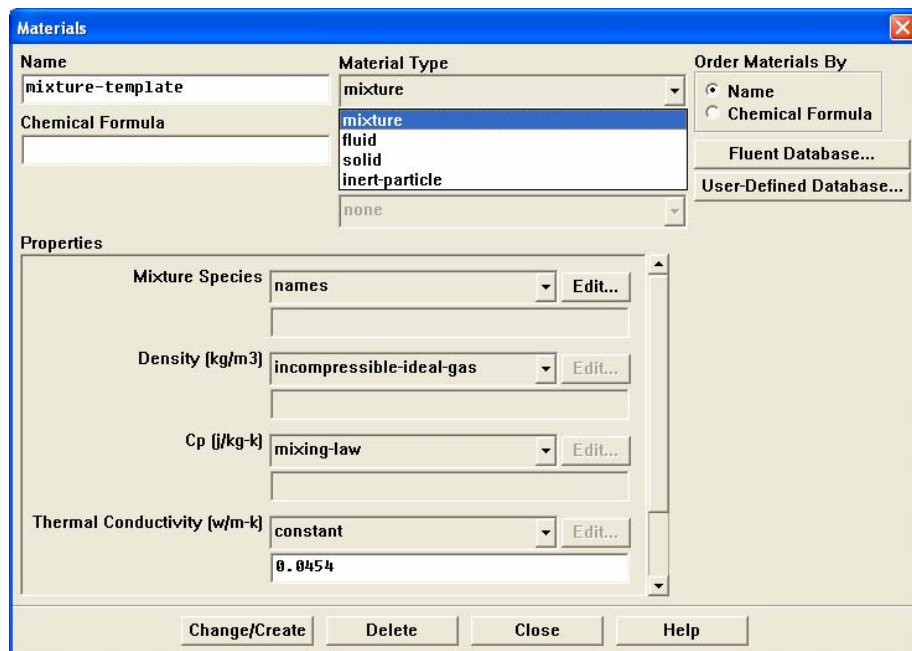
Courtesy Ford Motor Co.

Pressure Contours in a Squirrel Cage Blower

Setting Material Properties

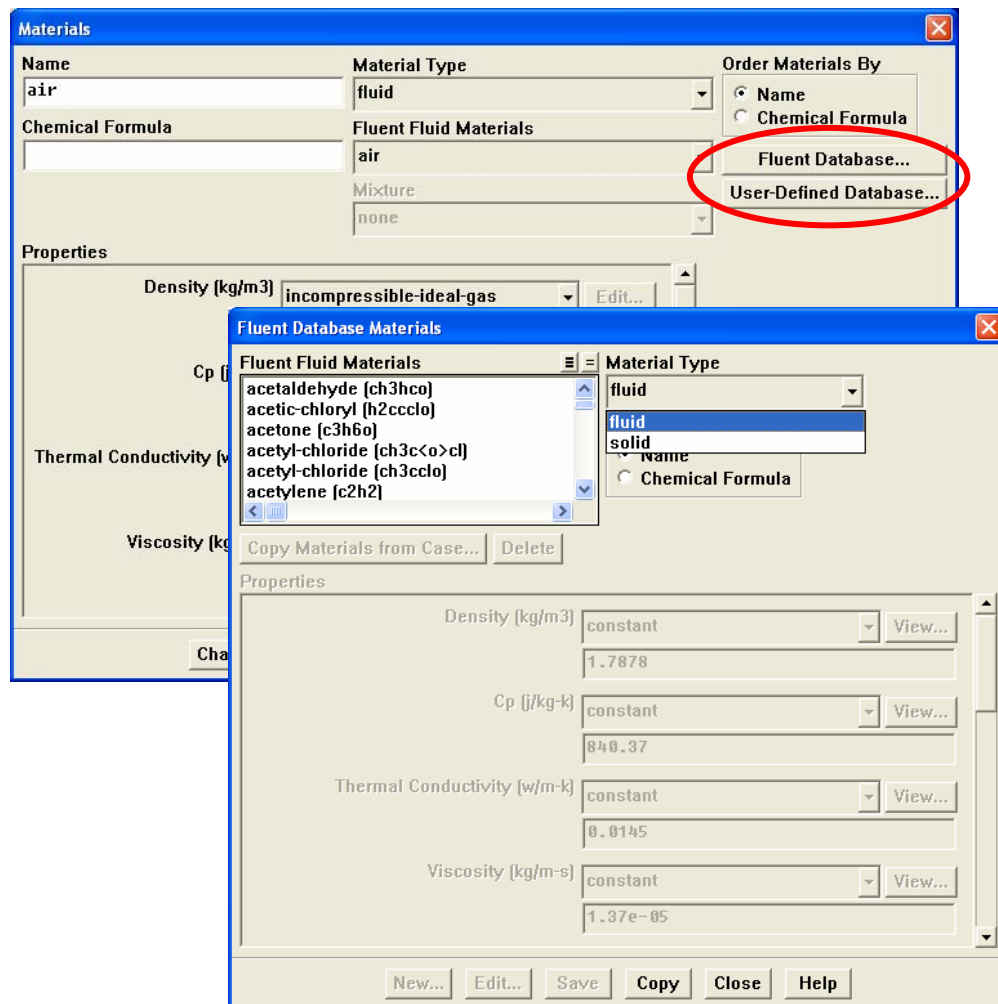
- ◆ Material properties can be set and modified in FLUENT, for fluids, solids, mixtures and particles.
- ◆ FLUENT provides a standard database of materials and the ability to create a customized user defined database.
- ◆ Physical models may require inclusion of additional materials and dictate which properties need to be defined.
 - Additional materials and mixture properties in gas phase combustion
 - Emissivity and absorbtivity for radiation
 - Thermal conductivity for heat transfer
 - Diffusivity for mass transfer
- ◆ Material properties can be customized, directly as functions of temperature, or using a user defined function as a function of other variables.

Define → Materials...



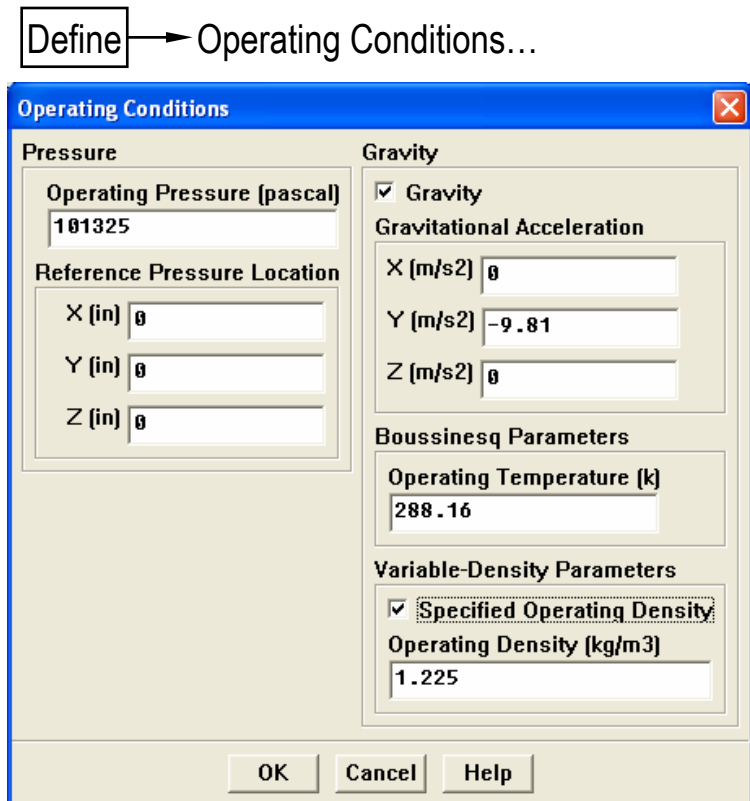
Materials Databases

- ◆ FLUENT materials database
 - Provides access to a number of pre-defined fluid, solid and mixture materials.
 - Materials can be copied to the case file and edited if required.
- ◆ Custom material database:
 - Create a new custom database of material properties and reaction mechanisms from materials in an existing case file for reuse in different cases
 - Custom databases can be created, accessed and modified from the standard materials panel in FLUENT.



Setting Operating Conditions

- ◆ Operating conditions set the reference values for FLUENT calculations:
 - The Operating Pressure with a Reference Pressure Location sets the reference value or baseline for calculating gauge pressure.
 - Set operating pressure close to the mean pressure in the domain.
 - The Gravity vector is used to calculate gravitational forces.
 - The Operating Temperature sets the reference value of temperature to calculate buoyancy forces with the Boussinesq model.
 - The Specified Operating Density sets the reference value for flows with widely varying density.



Solver Execution – Other Lectures

The image displays five overlapping ANSYS FLUENT dialog boxes, illustrating the setup for solver execution:

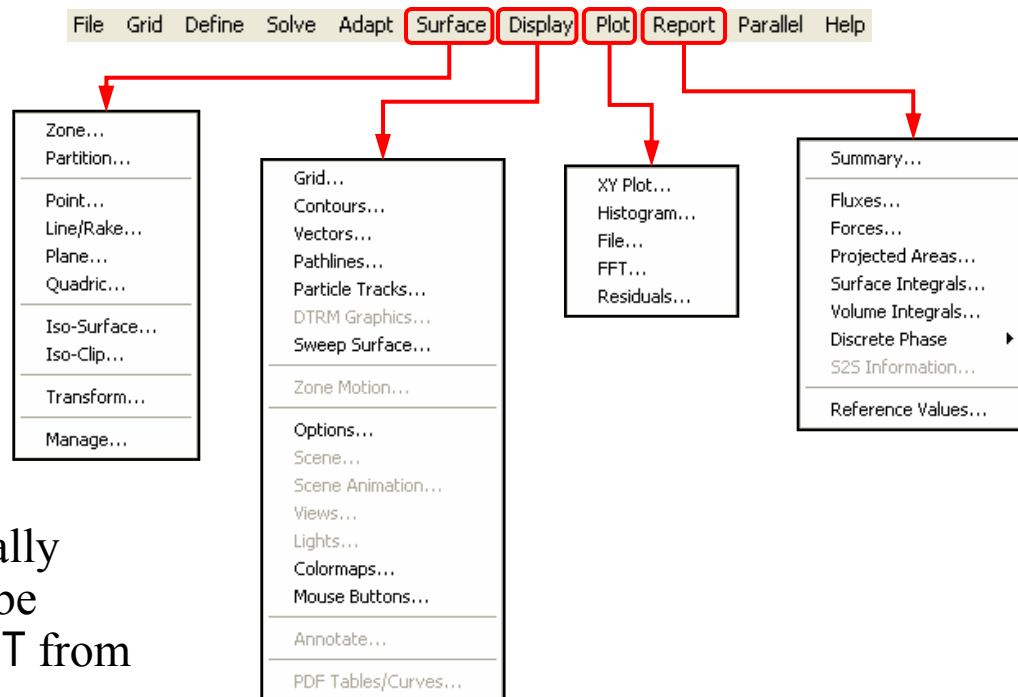
- Boundary Conditions:** Shows a list of zones (default-interior, fluid, inlet, orifice, orifice-p, orifice-p, outlet, wall) and a 'Type' column.
- Solution Controls:** Shows 'Equations' (Flow, Turbulence, Energy) and 'Under-Relaxation Factors'.
- Solution Initialization:** Shows 'Compute From' (inlet) and 'Initial Values' (Gauge Pressure, X Vel, Y Vel, Z Vel).
- Residual Monitors:** Shows 'Options' (Print, Plot), 'Storage' (Iterations: 1000), 'Normalization' (Normalize, Scale), 'Convergence Criterion' (absolute), and a table of residual monitors.
- Iterate:** Shows 'Time' (Time Step Size: 0.05, Number of Time Steps: 100, Time Stepping Method: Fixed), 'Options' (Data Sampling for Time Statistics), and 'Iteration' (Max Iterations per Time Step: 20, Reporting Interval: 1, UDF Profile Update Interval: 1).

◆ Physical models discussed on Day 2.

Post Processing

- ◆ Many post processing tools are available in FLUENT:

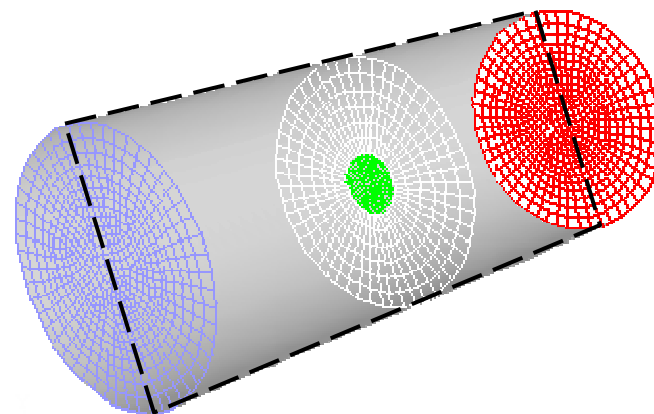
- Surface creation
- Display types
- Rendering options
- Plots of solution data
- Flux reports and Integral calculations



- ◆ Post processing functions typically operate on surfaces, which can be automatically created by FLUENT from existing zones or by the user.
- ◆ For more information, please refer to the web-based lecture, “Post-Processing in FLUENT” on www.LearningCFD.com or the FLUENT User Services Center.

Surface Creation

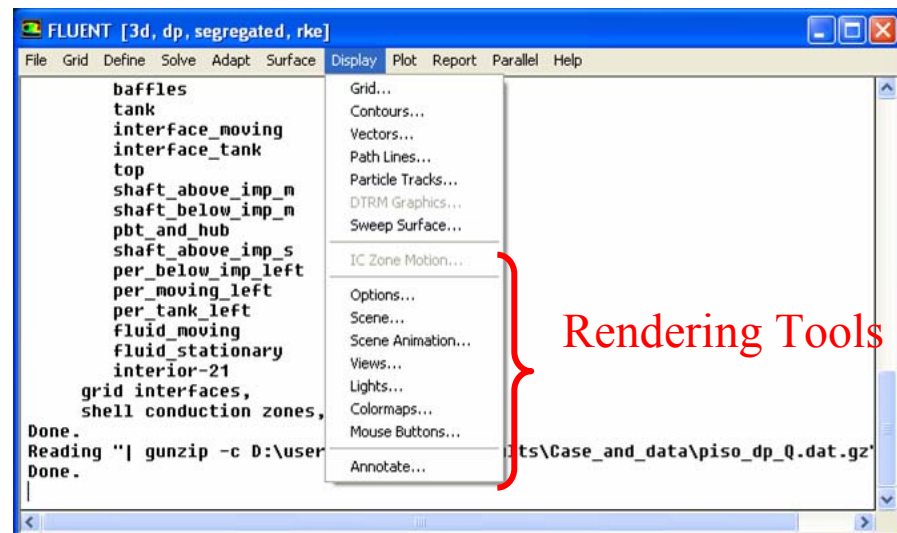
- ◆ Fluent allows you to select portions of the CFD domain, called **surfaces**, to be used for visualizing and plotting the flow field.
- ◆ There are a variety of ways to create surfaces:
 - Zone surfaces (surfaces automatically created by solver from zones)
 - Plane surfaces (specifying a specific plane in the domain)
 - Iso-surfaces (surfaces that have constant value for a specified variable)
 - Clipping Surfaces (iso-surfaces trimmed within specified range of values)
 - Point surfaces (specifying a particular location in the domain)
 - Line and Rake Surfaces (used for display of particle path lines)
- ◆ Surfaces can be renamed, deleted or moved and used to write out profile files.



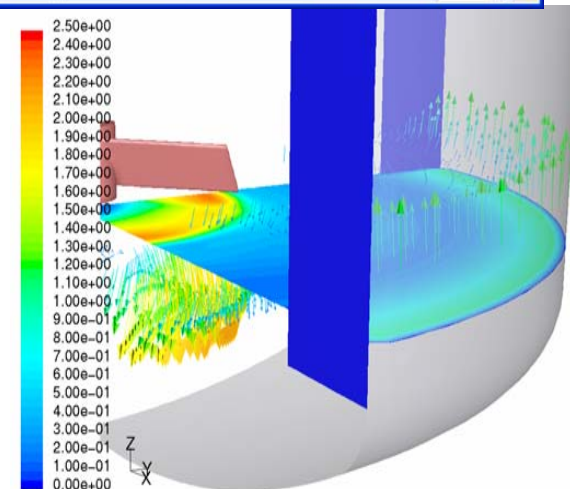
Example: an Iso-Surface of a constant grid coordinate can be created for viewing data within a plane.

Post-Processing Rendering Options

- ◆ The rendering options in FLUENT allow control of the look-and-feel of the post-processing plots, including:
 - Views and display options
 - Colormaps for contour/vector plots
 - Shading on surfaces using Lights
 - Annotation of plots
 - Surface manipulation
 - Scene Composition using plot overlays, different colors, shading, transparency
 - Scene animation (fly throughs)



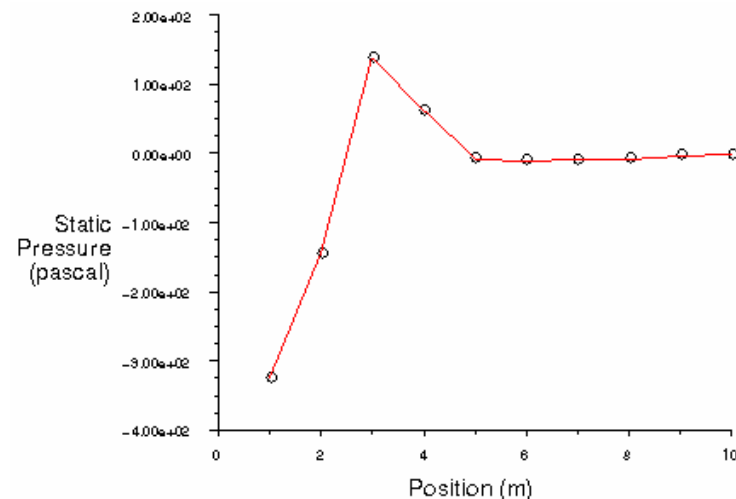
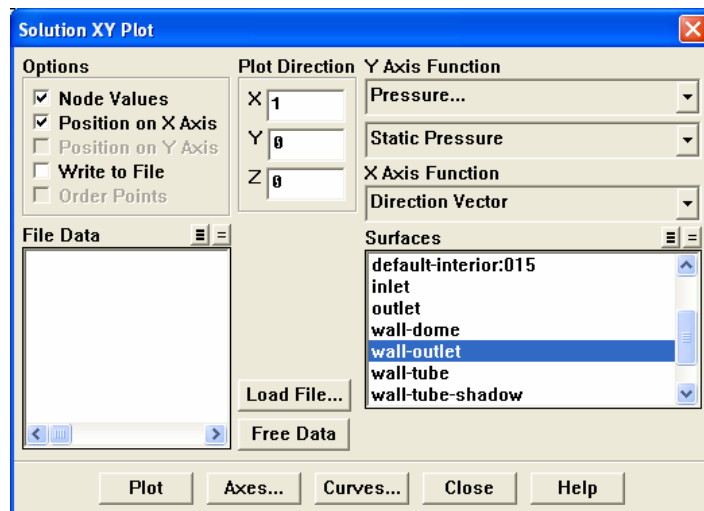
**Example of scene composition:
Overlay of contour
and vector plot with
transparent walls to
show internal
details.**



Plots

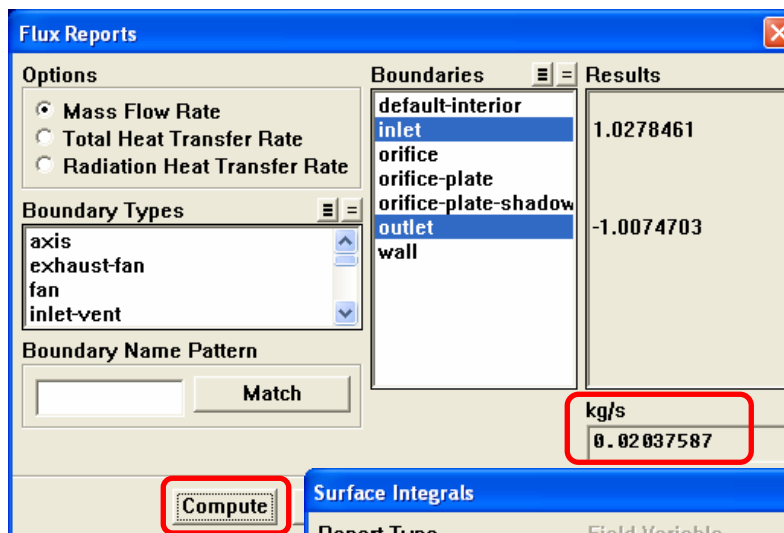
- ◆ FLUENT provides tools to generate data plots of the solution:
 - XY plots of solution variables
 - Histograms to illustrate frequency of distribution
 - Fast Fourier Transforms (FFT)
 - Residuals
- ◆ You can modify the colors, titles, legend, axis and curve attributes to customize your plots.
- ◆ Other data files (experimental, computational) can also be read in to compare results.

Plot → XY Plot...



Reports

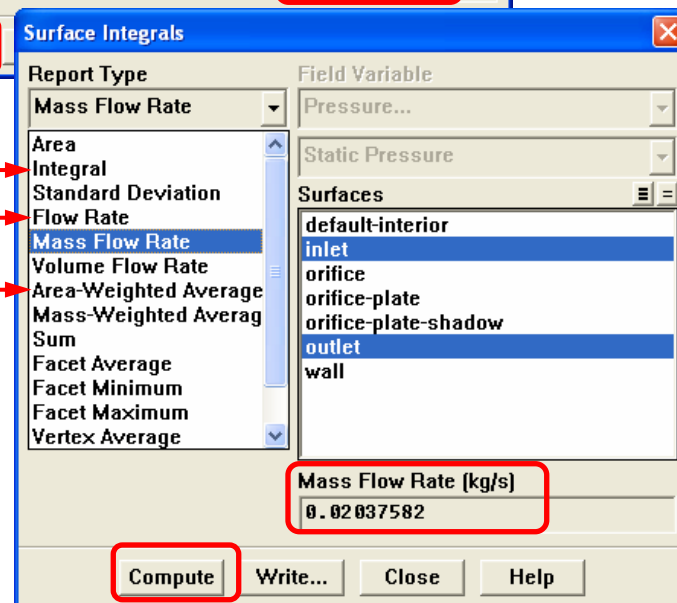
- ◆ Flux reports
 - Net flux is calculated.
 - Total Heat Transfer Rate includes radiation.
- ◆ Surface integrals
 - slightly less accurate on user-generated surfaces due to interpolation error.
- ◆ Volume integrals



$$\int \phi \, dA = \sum_{i=1}^n \phi_i |A_i|$$

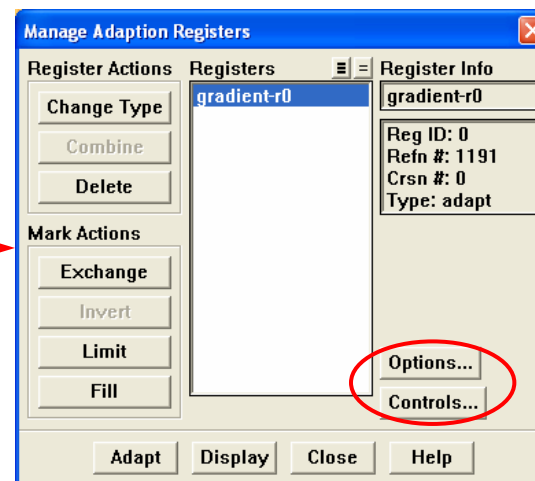
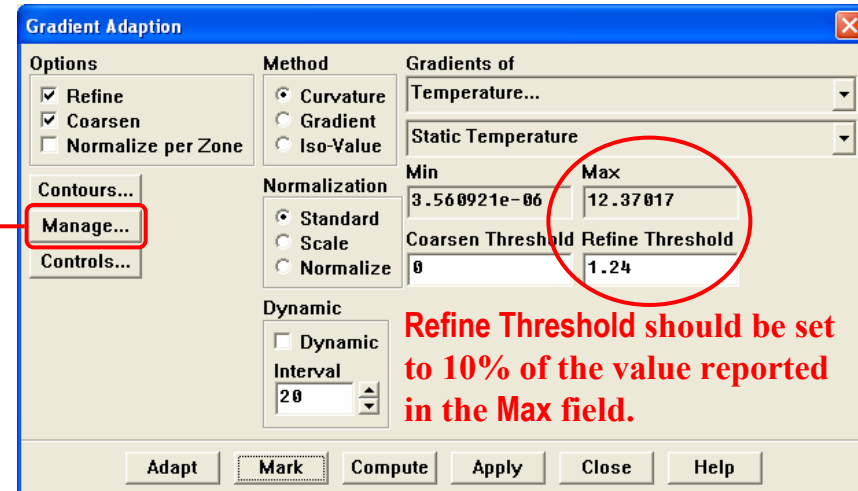
$$\frac{1}{A} \int \phi \rho \mathbf{V} \cdot d\mathbf{A} = \sum_{i=1}^n \phi_i \rho_i \mathbf{V}_i \cdot \mathbf{A}_i$$

$$\frac{1}{A} \int \phi \, dA = \frac{1}{A} \sum_{i=1}^n \phi_i |A_i|$$



Grid Adaption

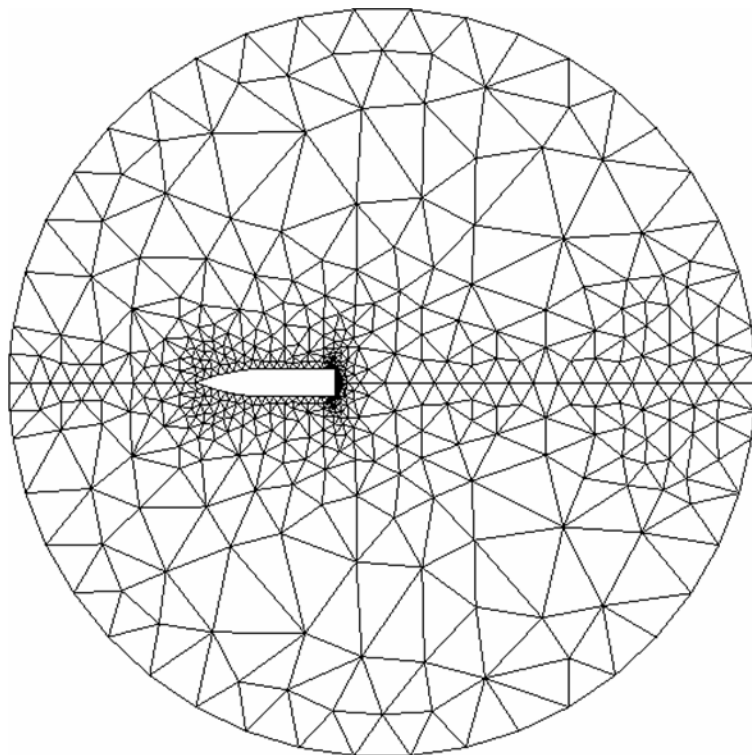
- ◆ Grid adaption refers to refinement and/or coarsening cells where needed to resolve the flow field without pre-processor.
- ◆ Adaption proceeds in three steps:
 - Mark cells satisfying the adaption criteria and store them in a “register.”
 - Display and modify the register.
 - Click Adapt to adapt the cells listed in the register.
- ◆ Registers can be defined based on:
 - Gradients or isovalues of all variables
 - All cells on a boundary
 - All cells in a region with a defined shape
 - Cell volumes or volume changes
 - y^+ in cells adjacent to walls
- ◆ To assist adaption process, you can:
 - Combine adaption registers
 - Draw contours of adaption function
 - Display cells marked for adaption
 - Limit adaption based on cell size and number of cells



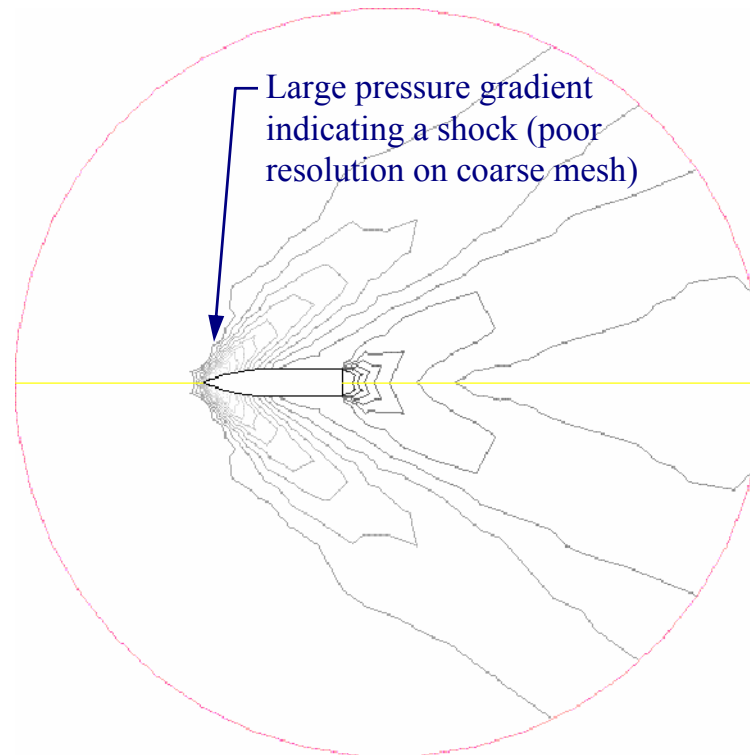
Always save case and data files prior to adapting the mesh!

Adaption Example – 2D Planar Shell

- ◆ Adapt grid in regions of large pressure gradient to better resolve the sudden pressure rise across the shock.



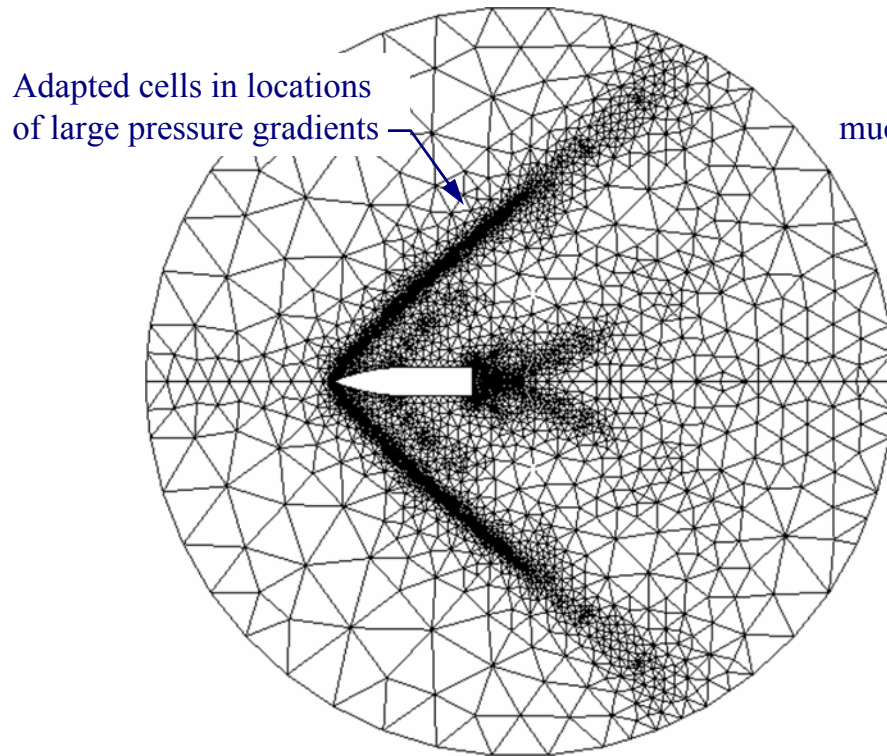
Initial Mesh



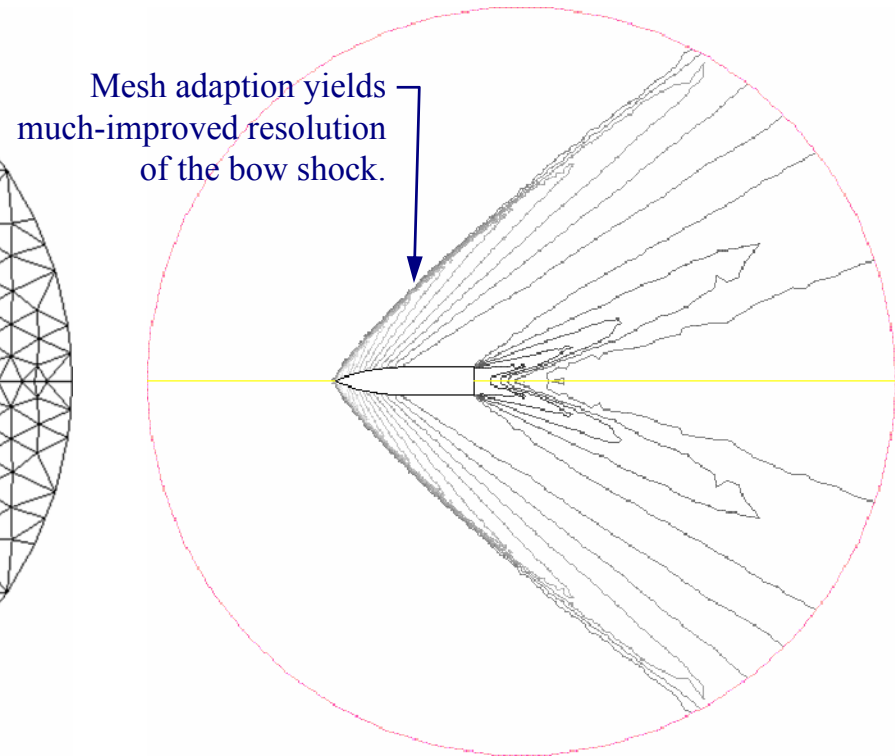
Pressure Contours on Initial Mesh

2D Planar Shell – Adapted Mesh and Solution

- ◆ Solution-based mesh adaption allows better resolution of the bow shock and expansion wave.



Adapted Mesh



Pressure Contours on Adapted Mesh

Parallel Processing

- ◆ Parallel processing can be used to run FLUENT on multiple processors to decrease turnaround time and increase simulation efficiency.
 - Critical for cases involving large mesh and/or complex physics.
- ◆ FLUENT is fully parallelized and capable on running across most hardware and software configurations, such as compute clusters or multi-processor machines.
- ◆ Parallel FLUENT can be launched either using a text command, or from within the FLUENT GUI and run in both batch and interactive modes.
 - **fluent 3d -t2** will launch a parallel FLUENT session using two CPUs.
- ◆ The grid can be partitioned for parallel processing automatically or manually.
 - Non-conformal meshes, sliding mesh interfaces and shell conduction zones require partitioning in serial.
- ◆ A web-based lecture, [Introduction to Parallel Processing](#), is available on the FLUENT User Services Center.

