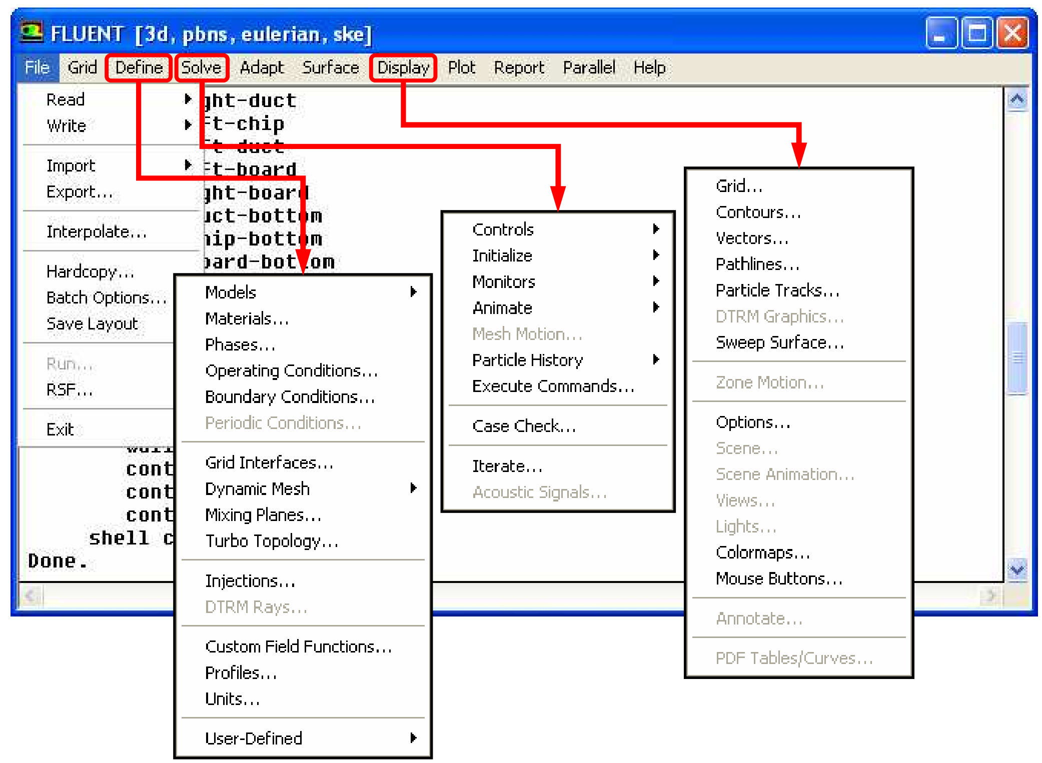
Solver Basics

Introductory FLUENT Training

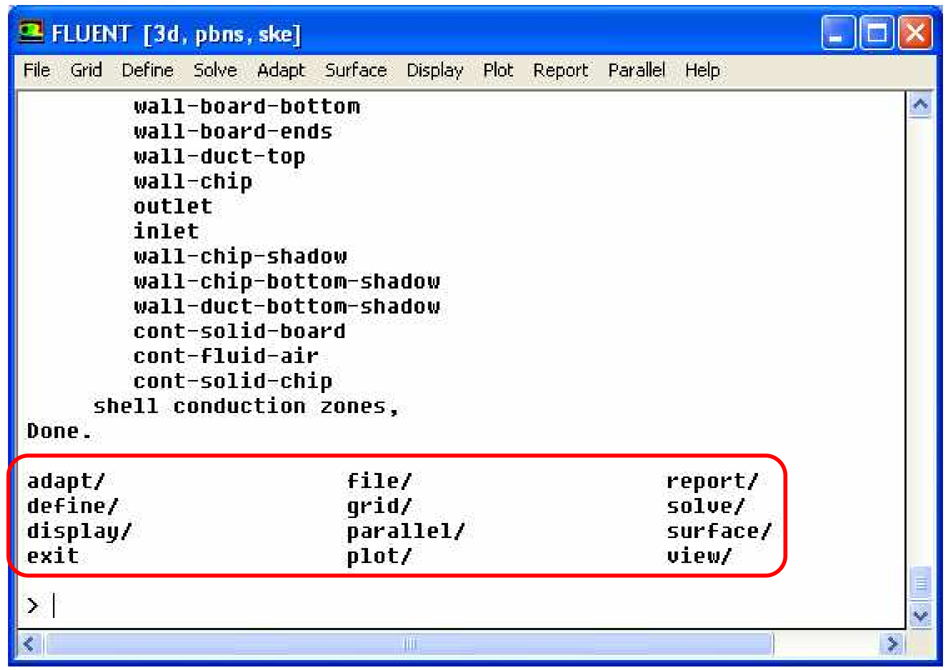
Solver Execution

- 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.

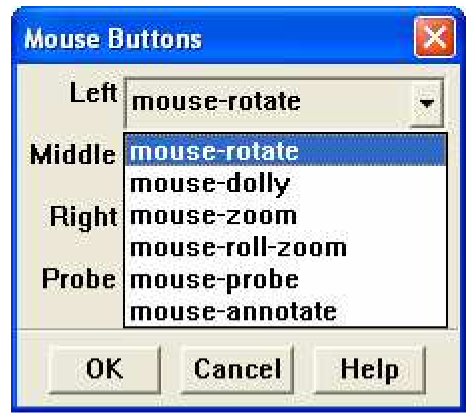
Sample Batch File

```
; 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

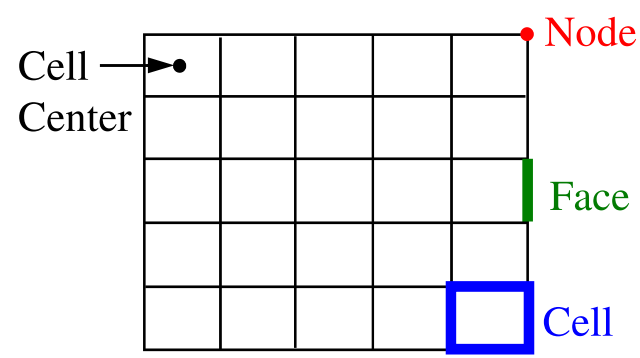
Display → Mouse Buttons...



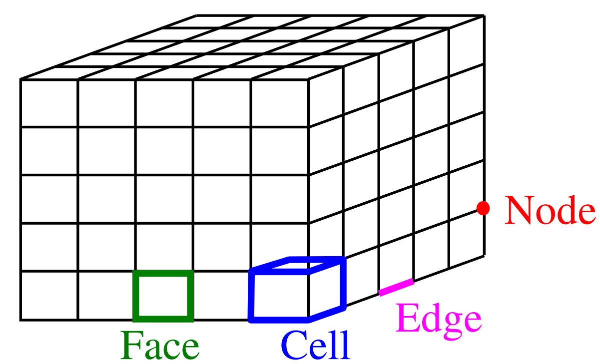
- Retrieve detailed flow field information at point with Probe enabled.
  - Right-click on the graphics display.

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.

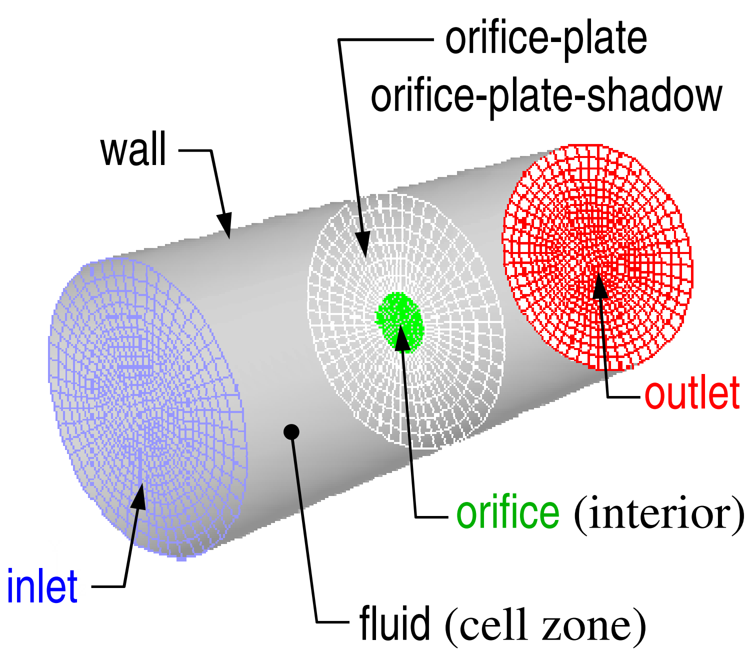


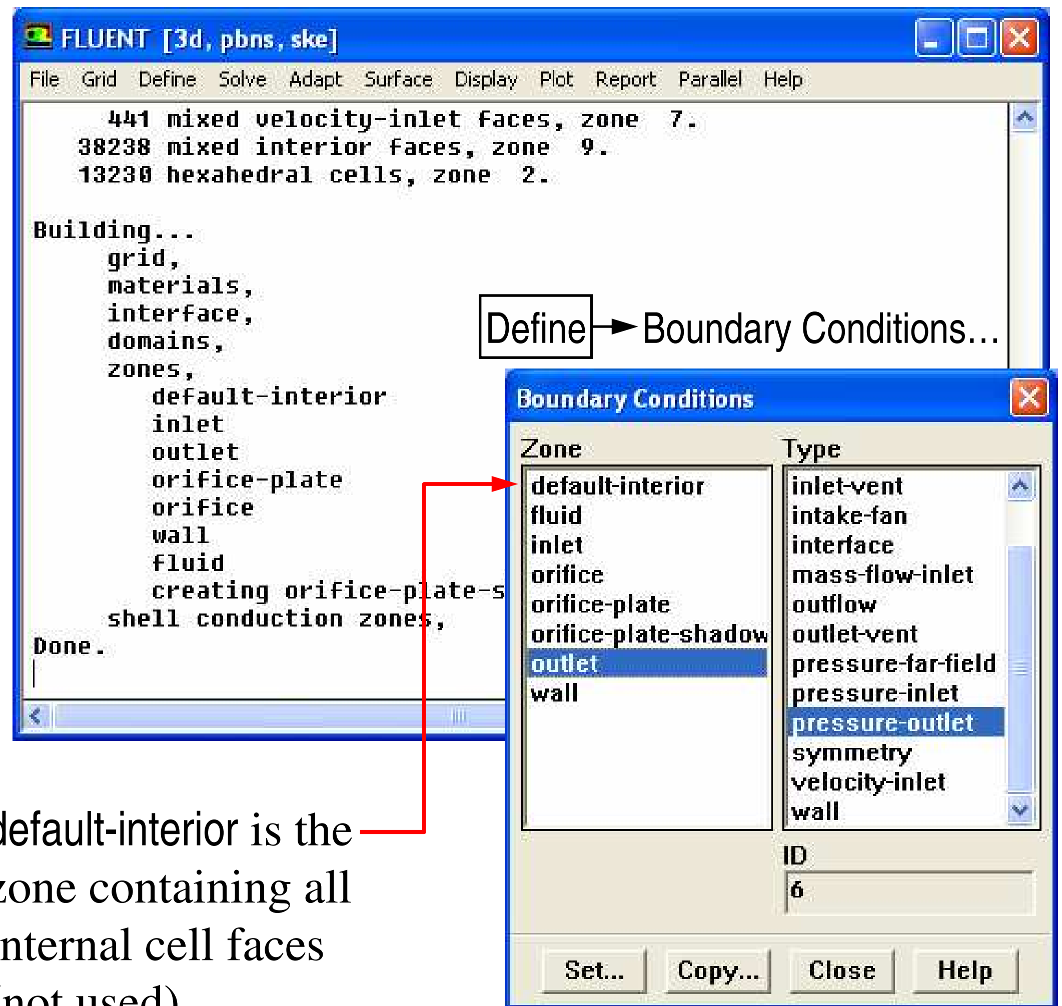
Simple 2D Mesh



Simple 3D mesh

Reading the Mesh – Zones



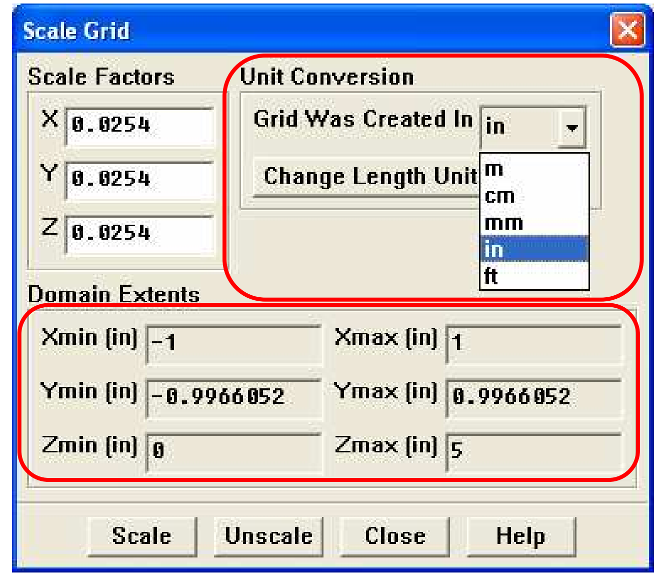


default-interior is the zone containing all internal cell faces (not used).

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.

    Grid → Reorder → Domain   Grid → Reorder → Zones

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

    Grid → Reorder → Print Bandwidth

- 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

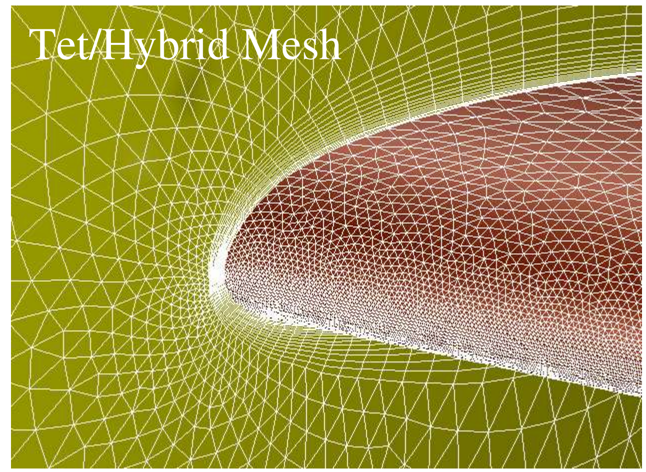
- A tetrahedral or hybrid grid can be converted in FLUENT 6.3 to polyhedra directly without going through a preprocessor.
  - Advantages
    - Improved mesh quality.
    - Can reduce cell count significantly.
    - User has control of the conversion process.
  - Disadvantages:
    - Cannot be adapted or converted again.
    - Cannot use tools such as smooth, swap, merge and extrude to modify the mesh.

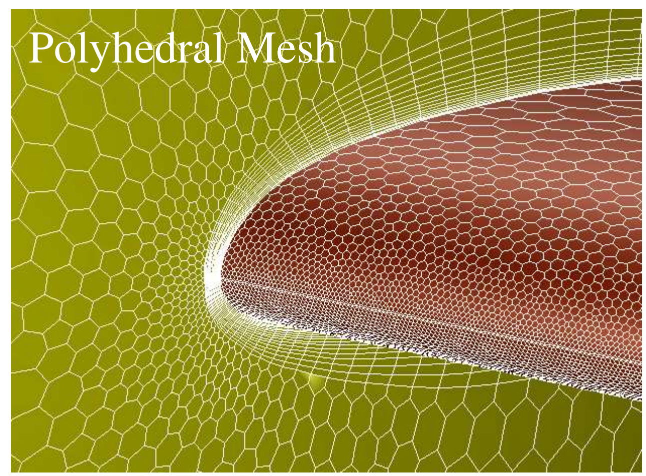
Grid → Polyhedra → Convert Domain

- 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

Grid → Polyhedra → Convert Skewed Cells

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.

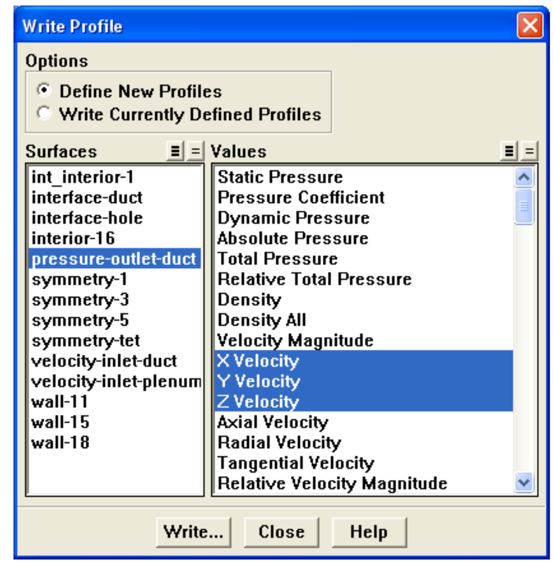
- Profile files are data files which contain point data for selected variables on particular face zones, and can be both written and read in a FLUENT session.

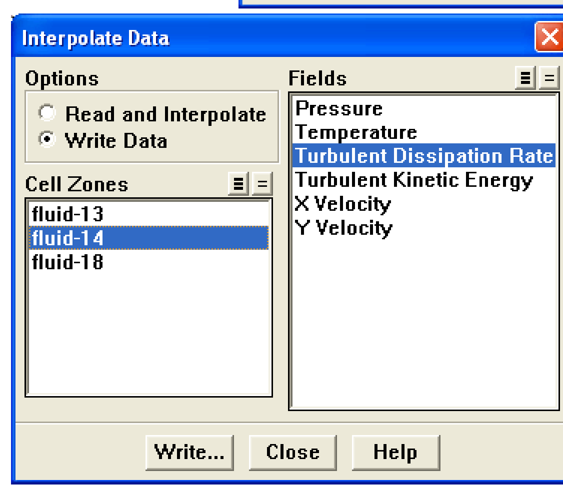
  File → Write → Profile...

  File → Read → Profile...

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

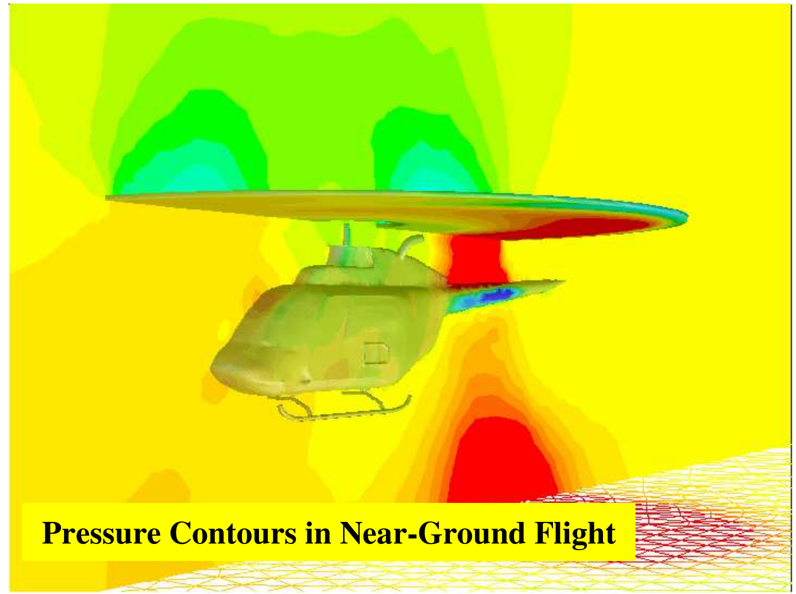
  File → Interpolate...

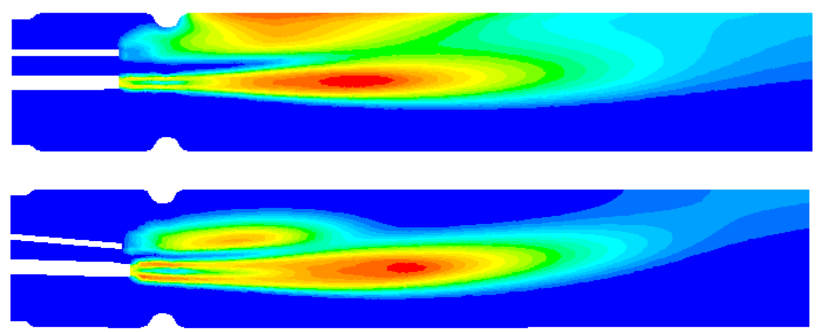




Models Available in FLUENT 6

- Fluid flow and heat transfer
  - Momentum, continuity, energy equations
  - Radiation
- Turbulence
  - RANS-based models (Spalart-Allmaras, k–ε, k–ω, 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

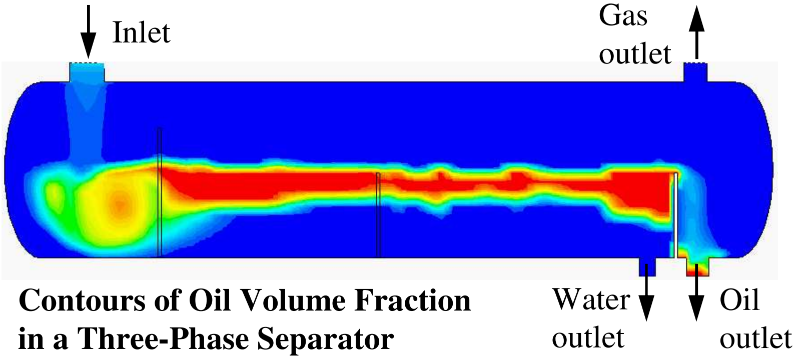




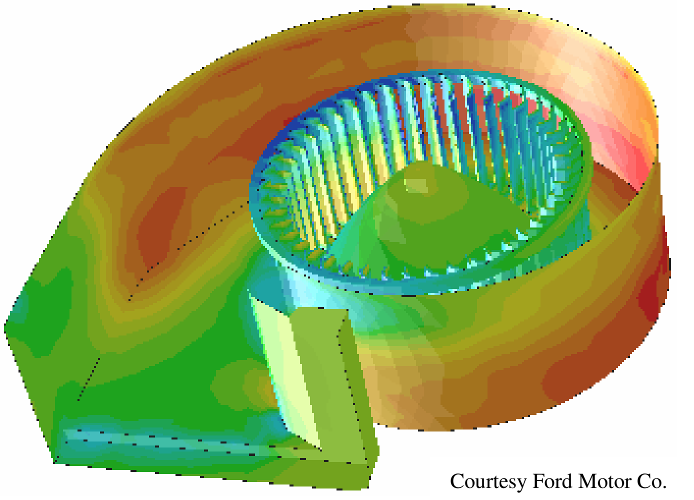
Temperature Contours for Kiln Burner Retrofit

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



Contours of Oil Volume Fraction in a Three-Phase Separator

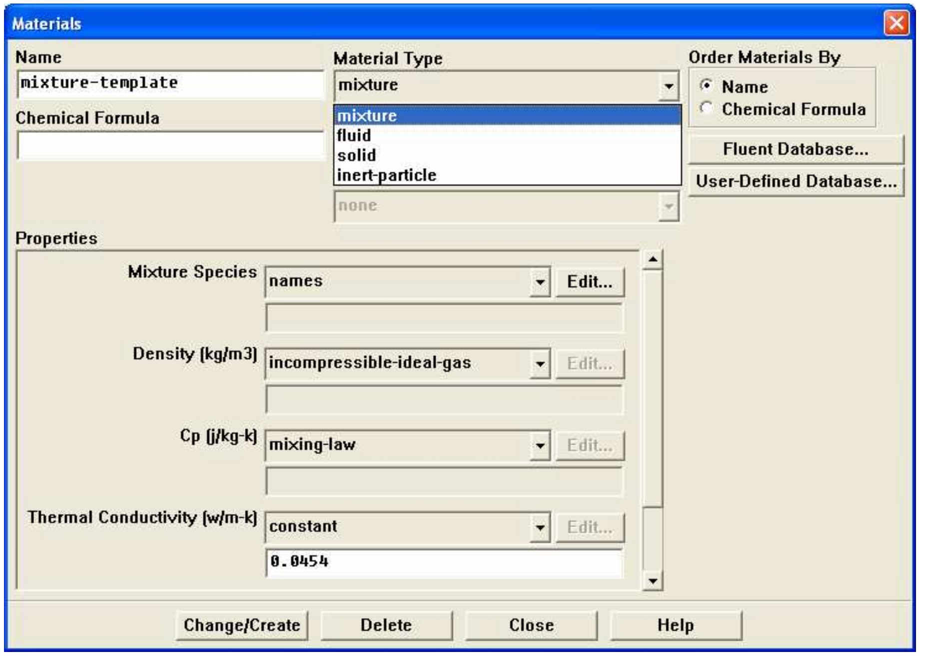


Pressure Contours in a Squirrel Cage Blower

Setting Material Properties

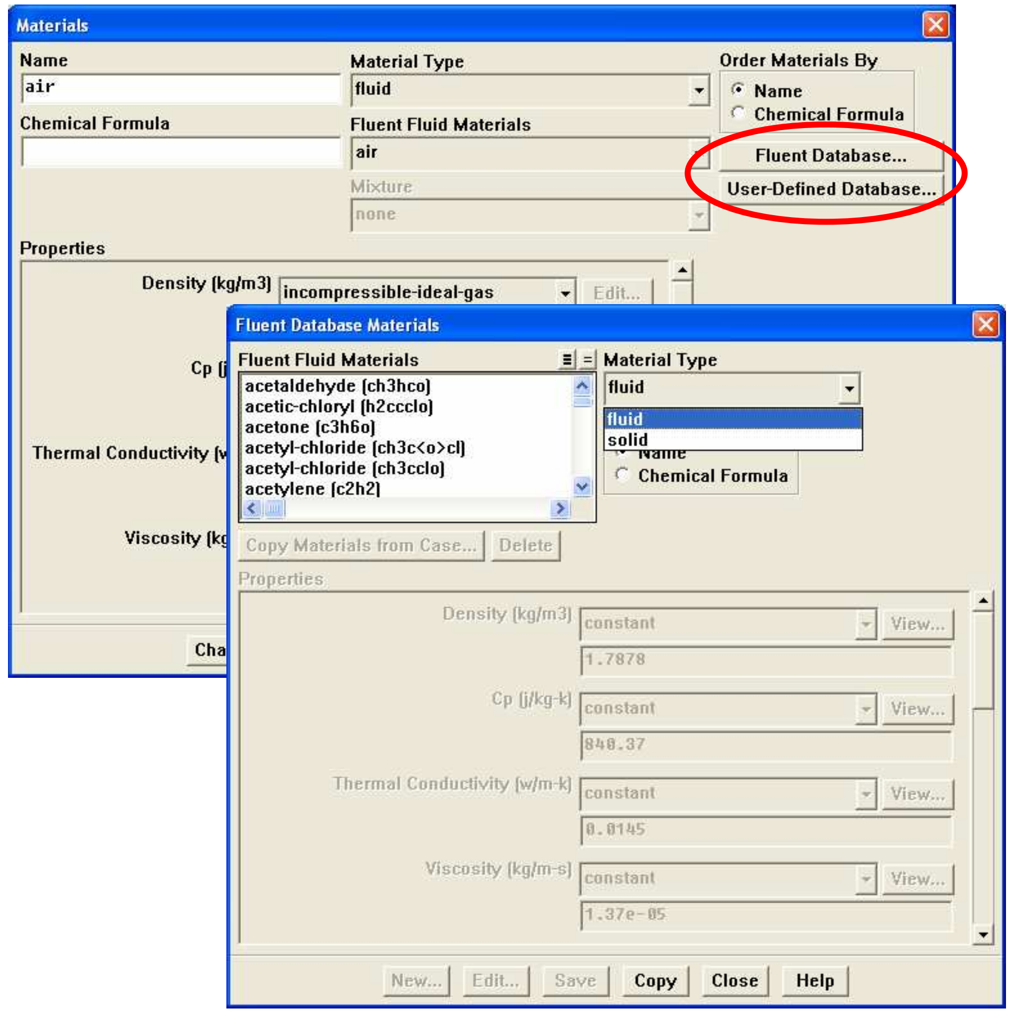
- Material properties can be set and modified in FLUENT, for fluids, solids, mixtures and particles.
- FLUENT provides a standard database of materials and the ability to create a customized user defined database.
- Physical models may require inclusion of additional materials and dictate which properties 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

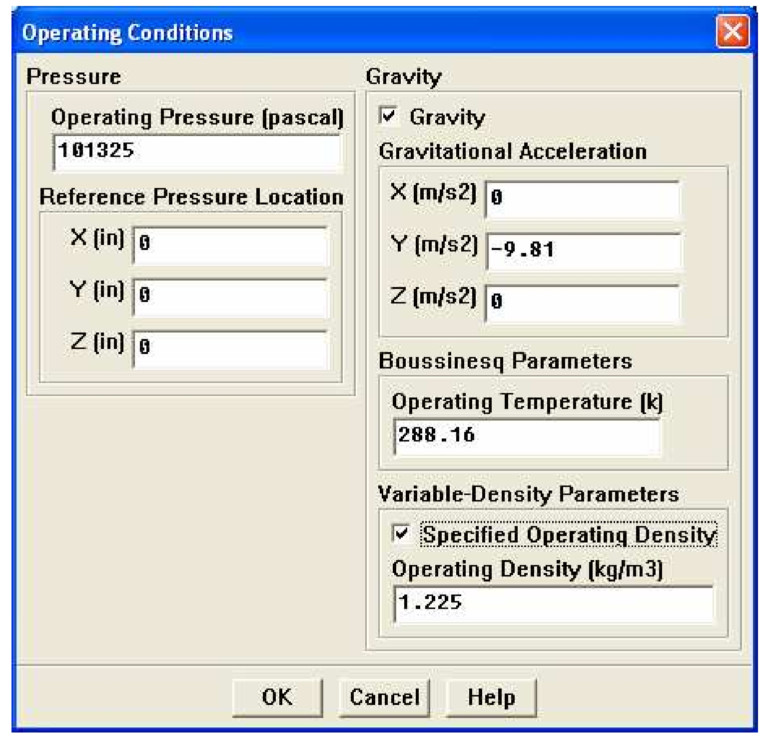
- FLUENT materials database
  - Provides access to a number of pre-defined fluid, solid and mixture materials.
  - Materials can be copied to the case file and edited if required.
- Custom material database:
  - Create a new custom database of material properties and reaction mechanisms from materials 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.



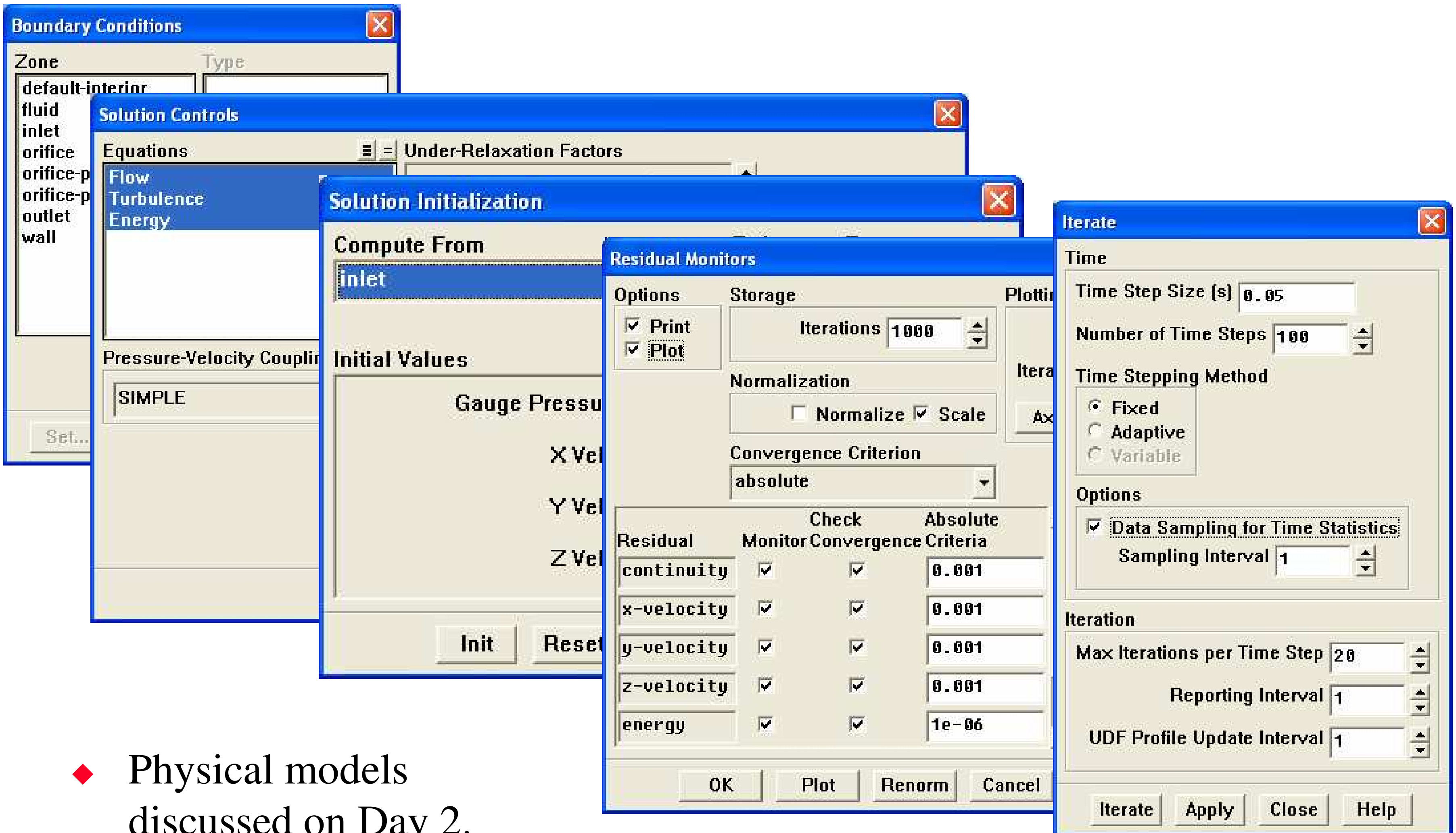
Setting Operating Conditions

- 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.

Define → Operating Conditions...



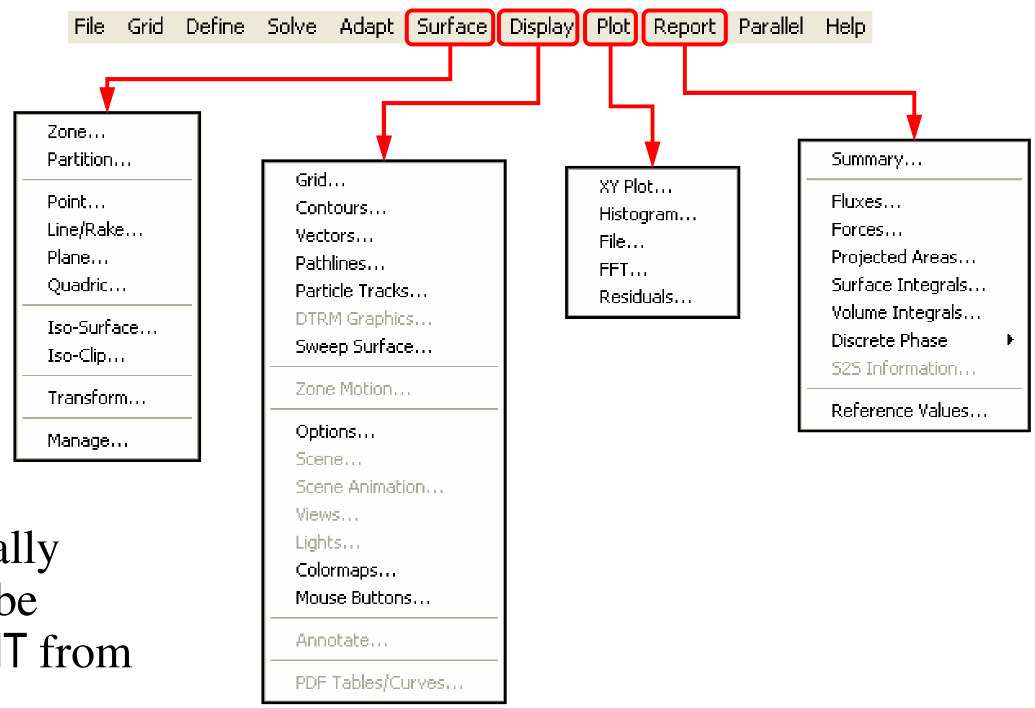
Solver Execution – Other Lectures



- 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

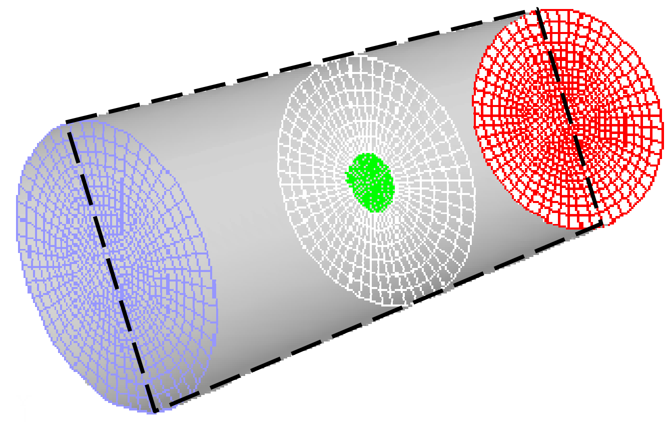


- Post processing functions typically operate on surfaces, which can be automatically created by FLUENT from existing zones or by the user.

- 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

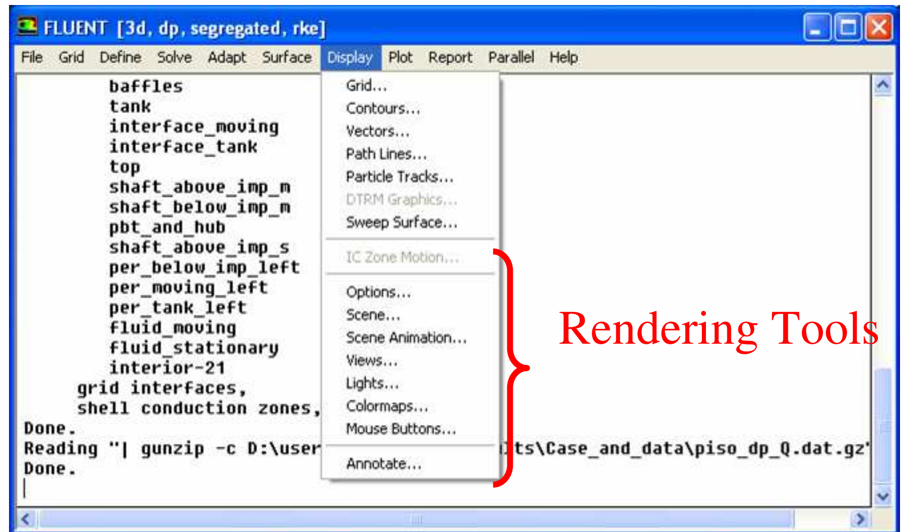
- Fluent allows you to select portions of the CFD domain, called **surfaces,** to be used for visualizing and plotting the flow field.
- There are a variety of ways to create surfaces:
  - Zone surfaces (surfaces automatically created by solver from zones)
  - Plane surfaces (specifying a specific plane 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.



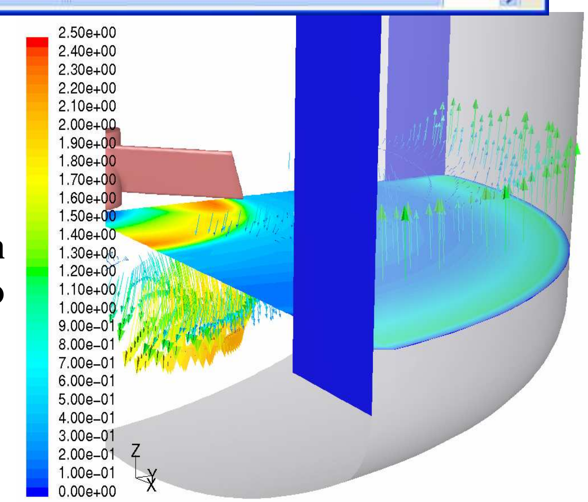
Example: an Iso-Surface of a constant grid coordinate can be created for viewing data within a plane.

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)



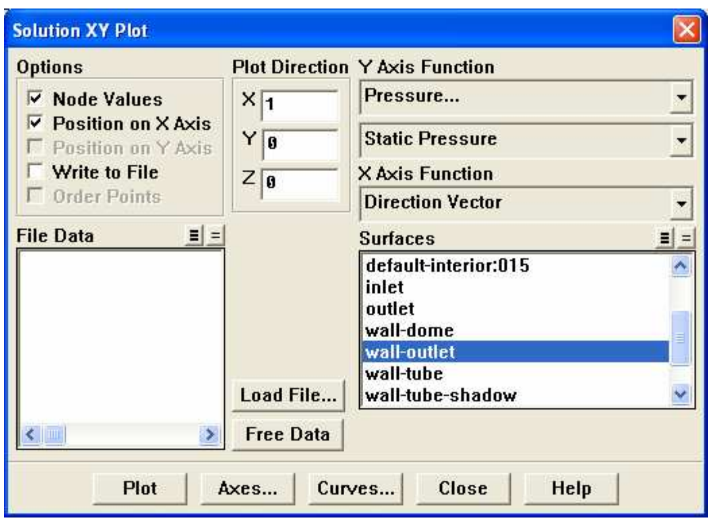
Example of scene composition: Overlay of contour and vector plot with transparent walls to show internal details.

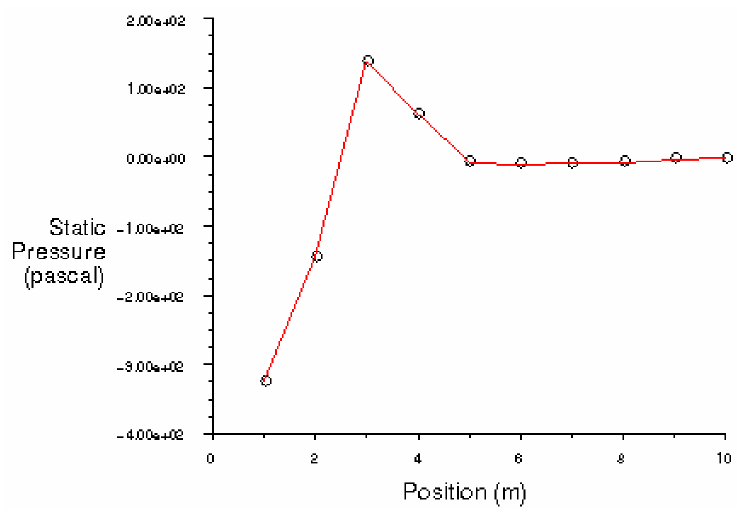


Plots

- 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...





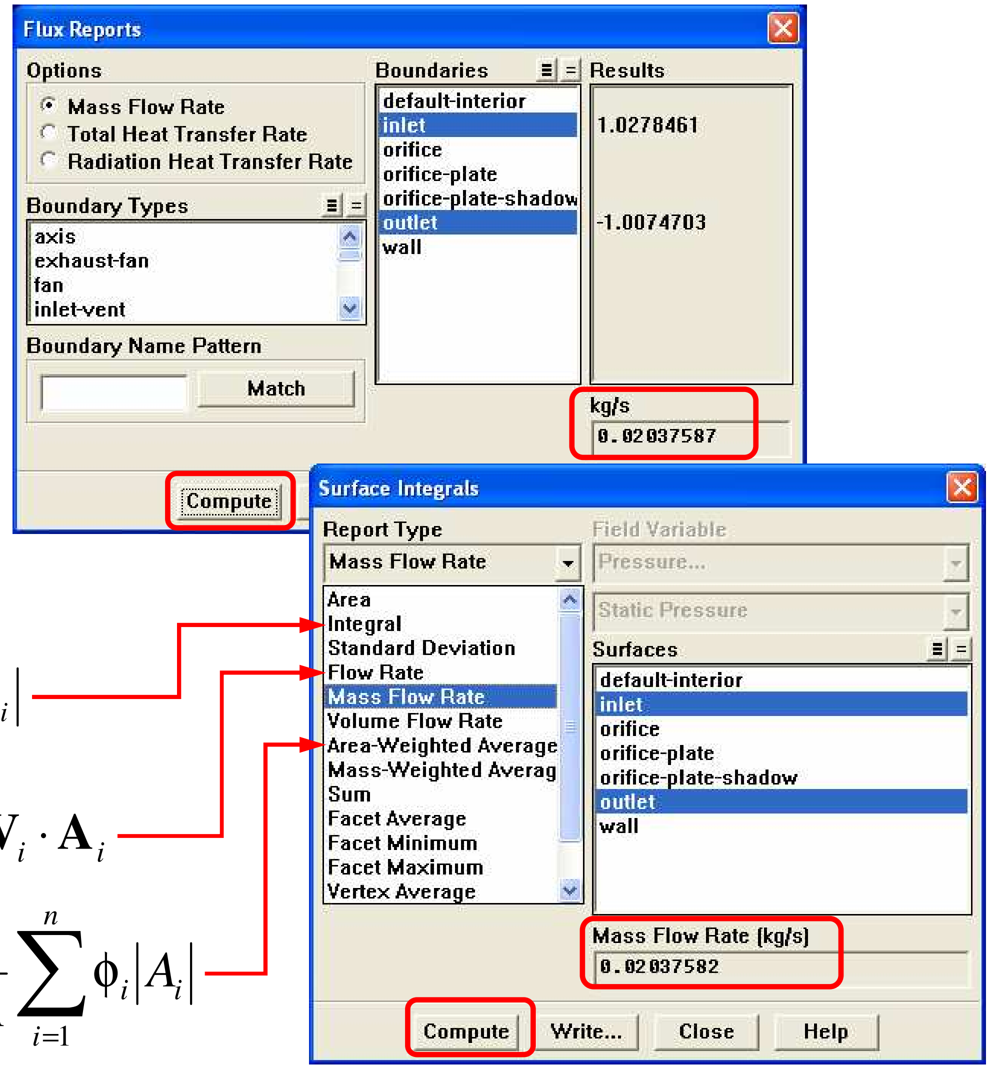
Reports

- 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

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

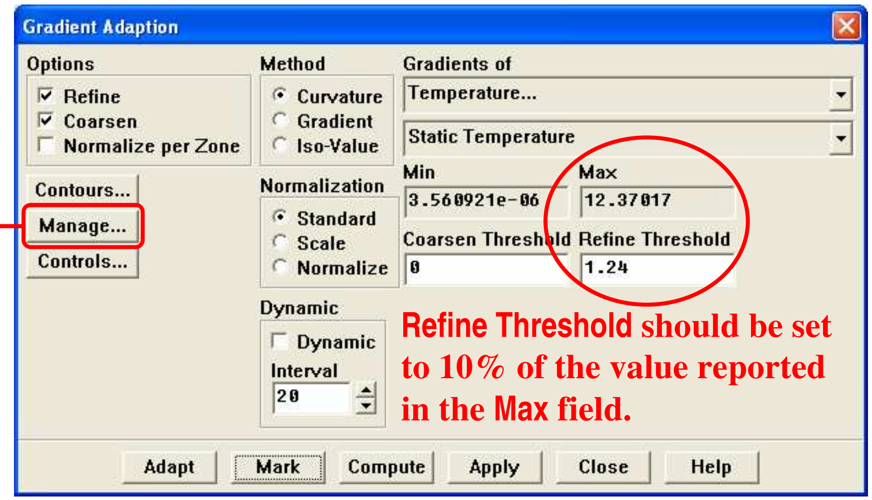
$$\frac{1}{A}\int \phi \rho \mathbf{V} \cdot d\mathbf{A} = \sum_{i=1}^{n} \phi_i \rho_i \mathbf{V}_i \cdot \mathbf{A}_i$$

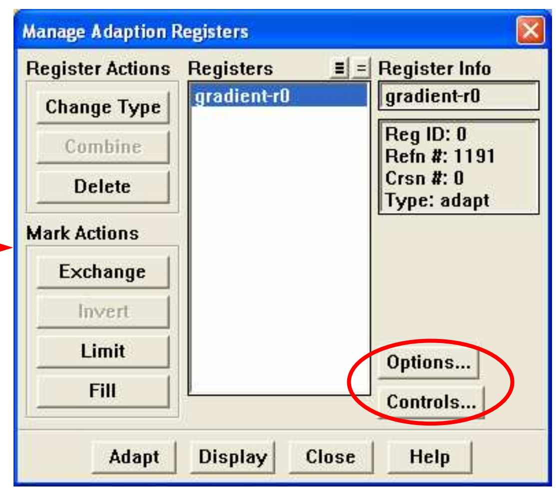
$$\frac{1}{A}\int \phi \, dA = \frac{1}{A}\sum_{i=1}^{n} \phi_i |A_i|$$



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

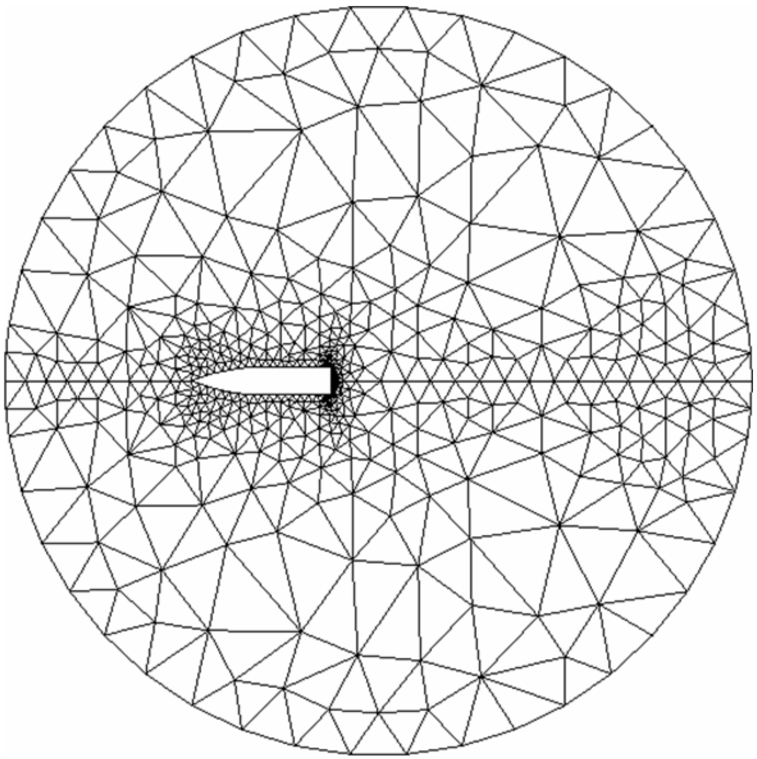




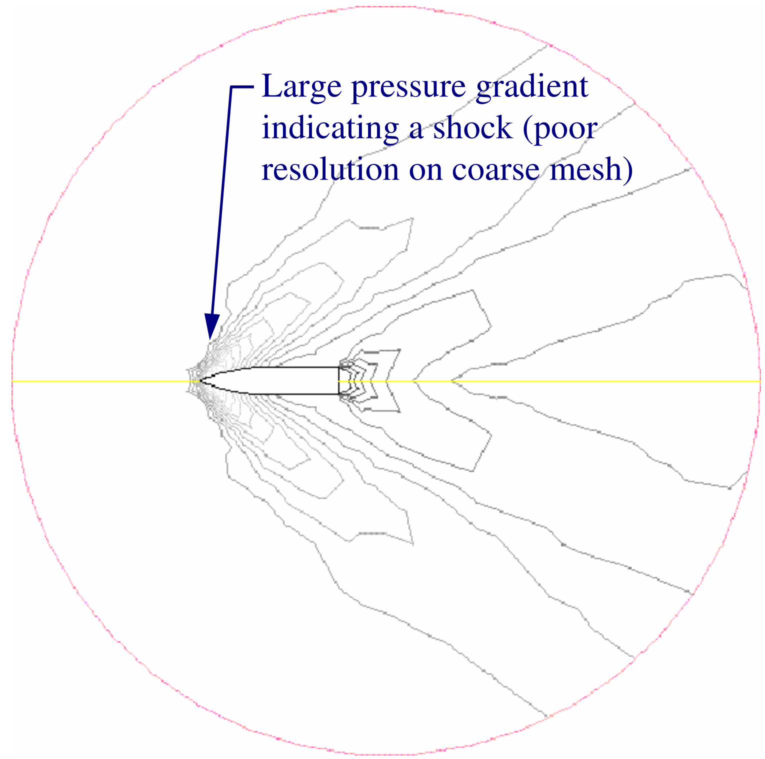
Always save case and data files prior to adapting the mesh!

Adaption Example – 2D Planar Shell

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



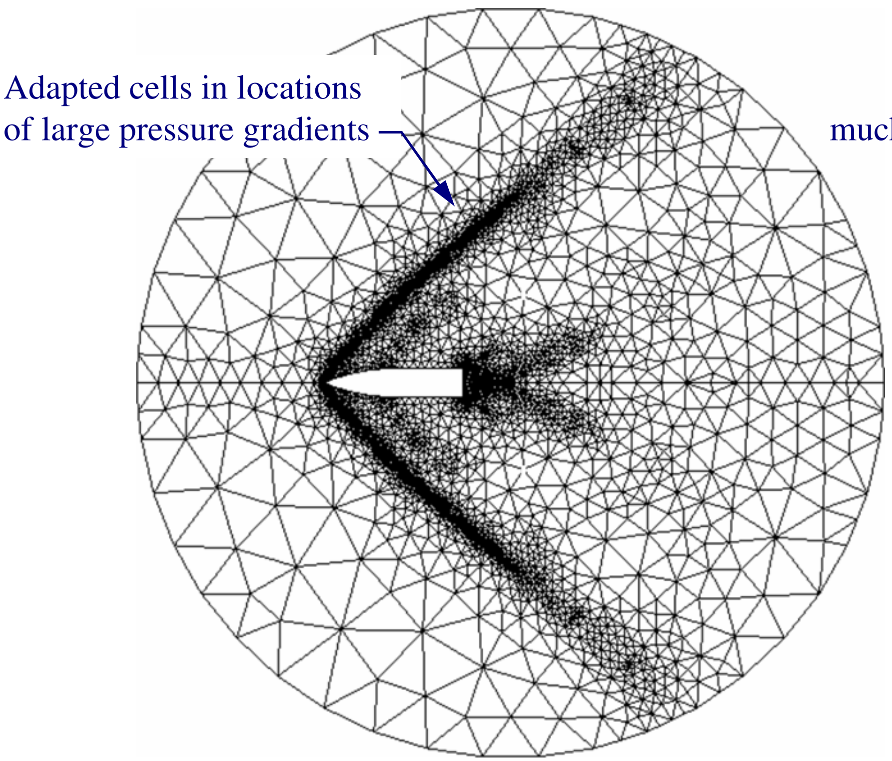
Initial Mesh



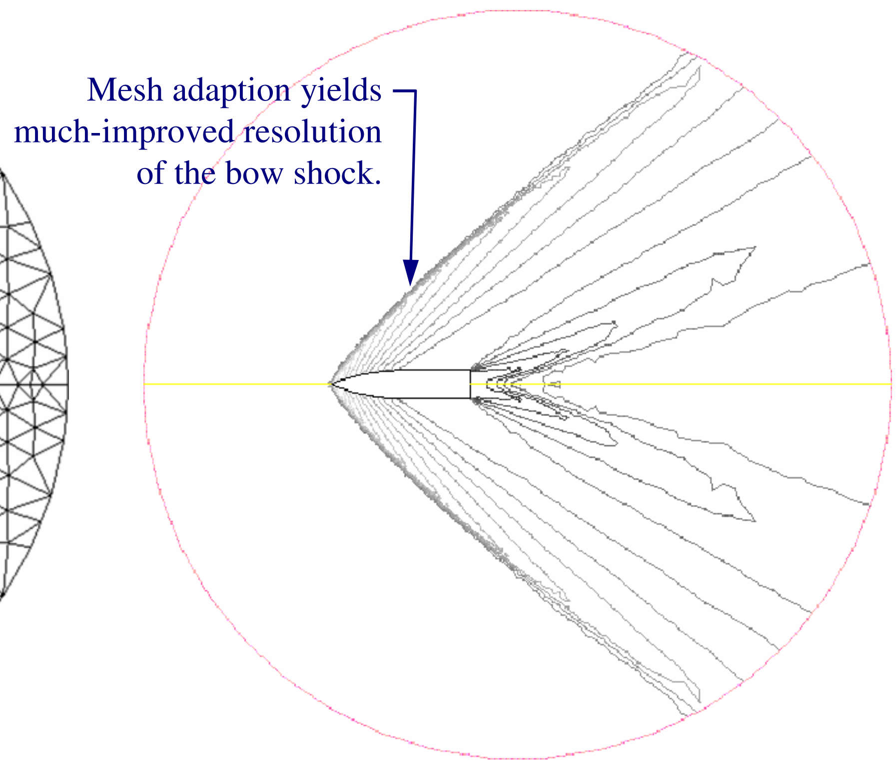
Pressure Contours on Initial Mesh

2D Planar Shell – Adapted Mesh and Solution

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

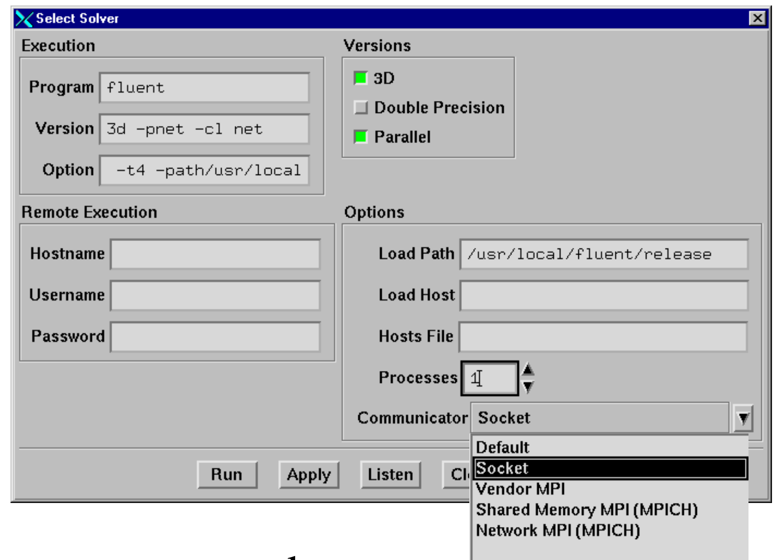


Adapted Mesh



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.