Customer Training Material

# Lecture 3

## Basic Overview of Using the FLUENT User Interface

# Introduction to ANSYS FLUENT





Release 13.0
December 2010

# Parallel Processing



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

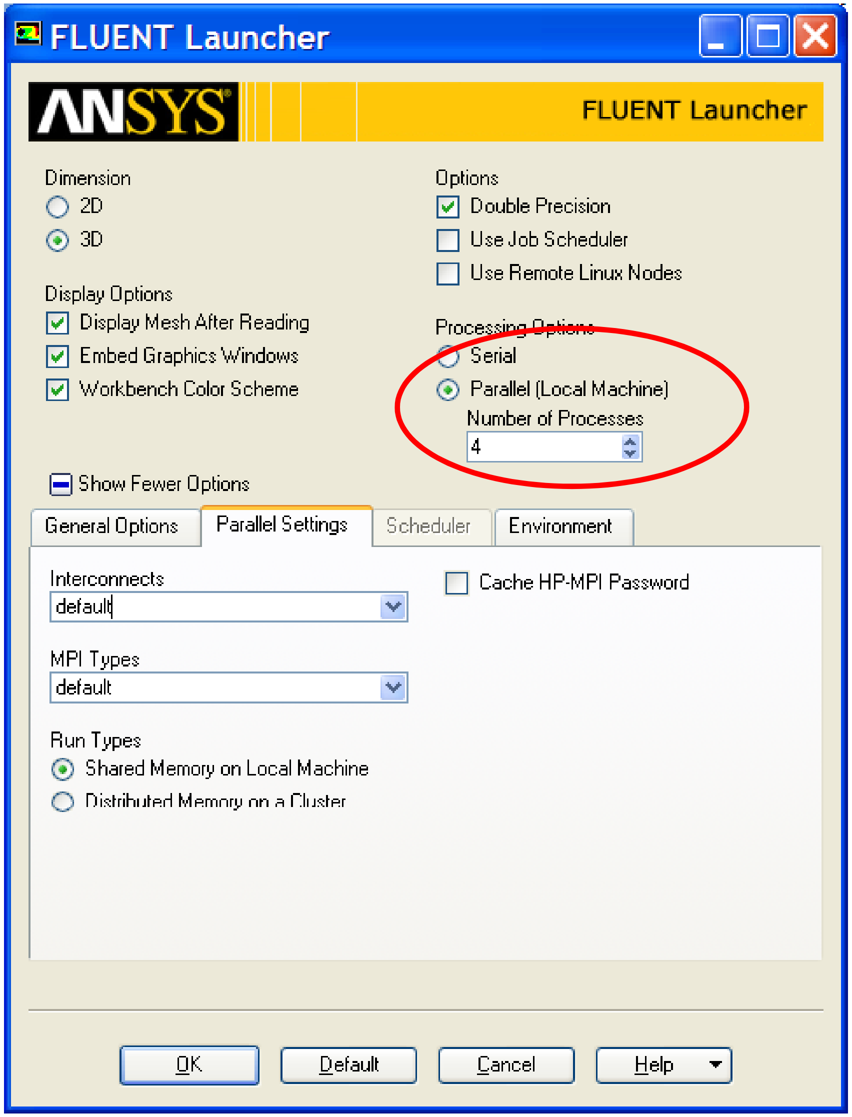
- It is common for modern generation computers to have several processors, and / or several compute ‘cores’ per processor. Each one of these can be a ‘node’ for the FLUENT simulation.

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

- Running FLUENT on a single machine with multiple cores can simply be set by selecting the ‘parallel’ option on startup (see image).

- Alternatively a distributed parallel cluster can be set up, and the simulation run across many* machines simultaneously

  * FLUENT scales well, and simulations using several hundred nodes are not unheard of.

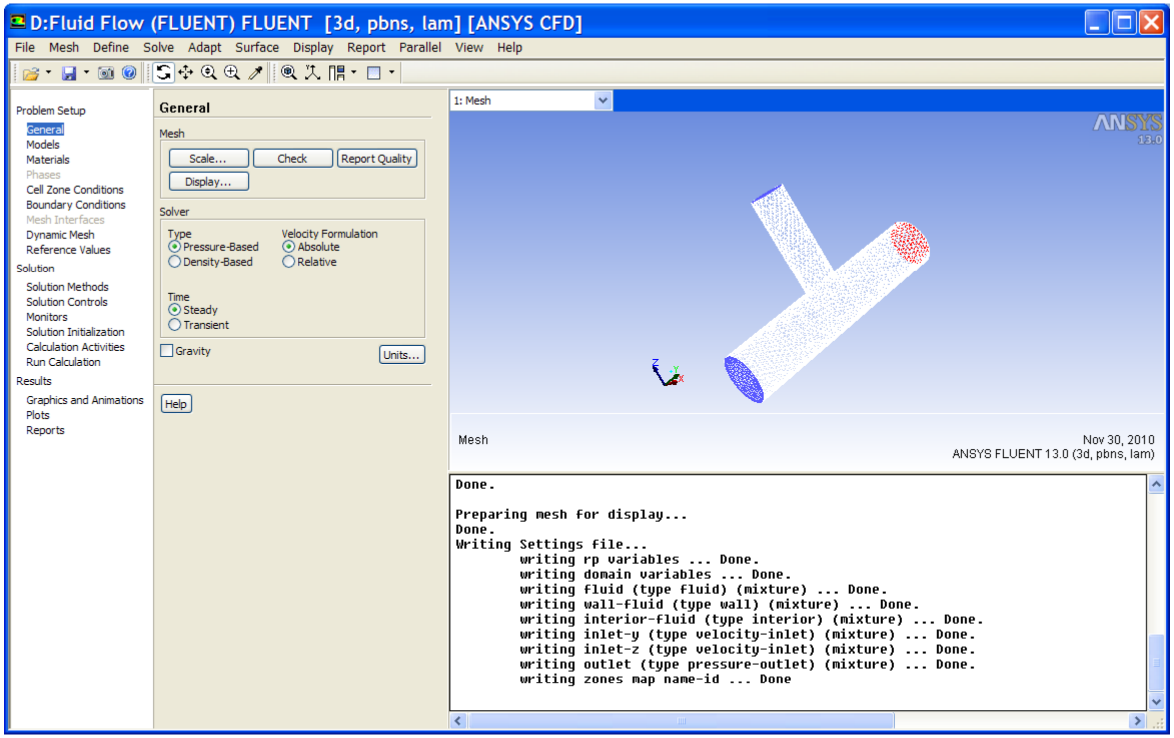


# FLUENT 13 GUI Navigation



- The FLUENT GUI is arranged such that the tasks are generally arranged from top to bottom in the project setup tree.

- Selecting an item in the tree opens the relevant input items in the center pane.
  - General
  - Models
  - Materials
  - Boundary Conditions
  - Solver Settings
  - Initialization and Calculation
  - Postprocessing

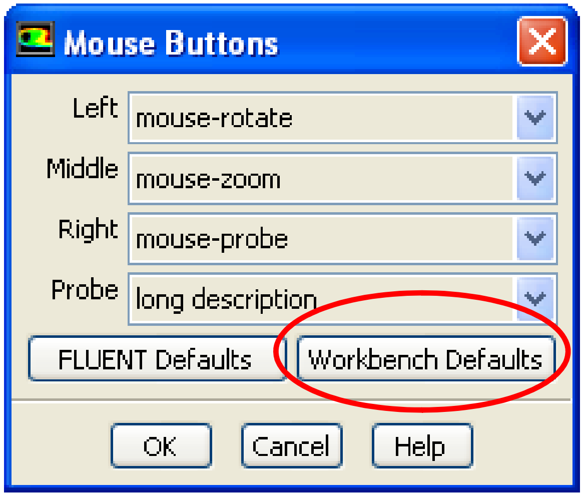


# Mouse Functionality



- 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…

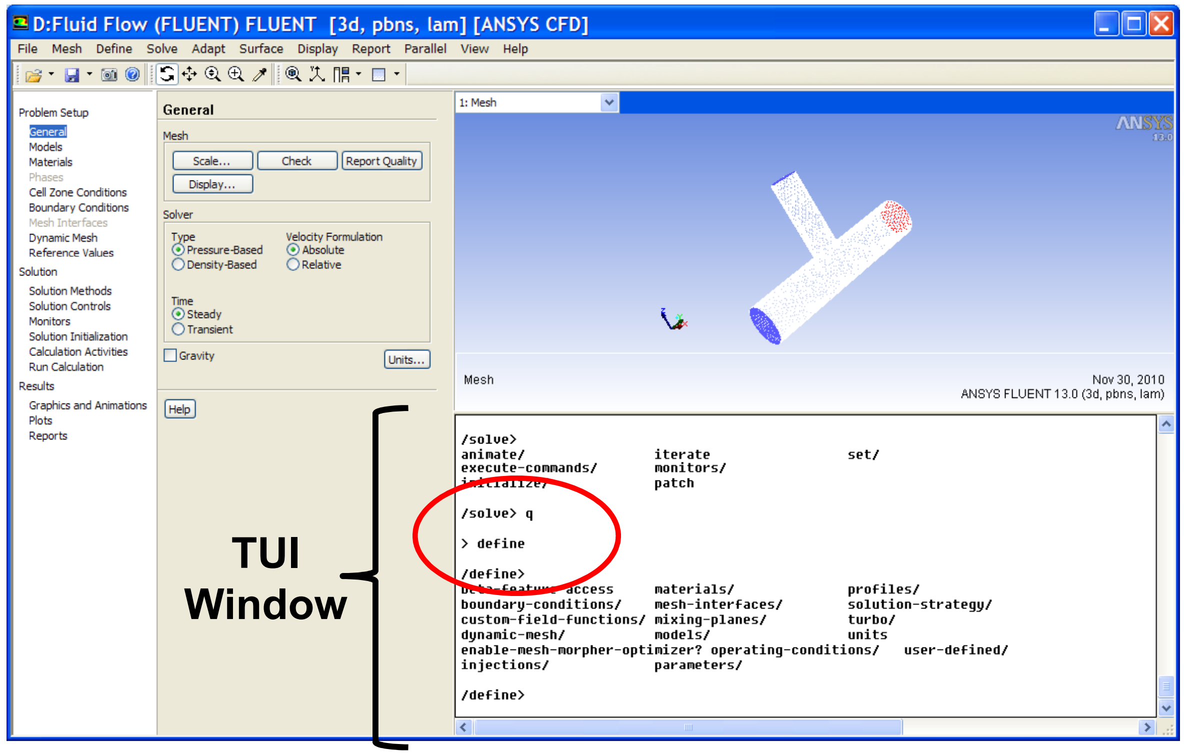



- Retrieve detailed flow field information at point with Probe enabled.
  - Right-click on the graphics display.
- User can choose between classic 'FLUENT' settings, or for mouse behaviour consistent with Workbench.

# Text User Interface



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




- The TUI offers many very valuable benefits:
  - Journal (text) files can be constructed to automate repetitive tasks.
  - FLUENT can be run in batch mode, with TUI journal scripts set to automate the loading / modification / solver execution and postprocessing.
  - Very complex models can be set using a spreadsheet to generate the TUI commands.

# Sample FLUENT Journal



- A journal file is a text file which contains TUI commands which FLUENT will execute sequentially.
- Note that the FLUENT TUI accepts abbreviations of the commands for example,
  - **`rcd`** Reads case and data files
  - **`wcd`** Writes case and data files

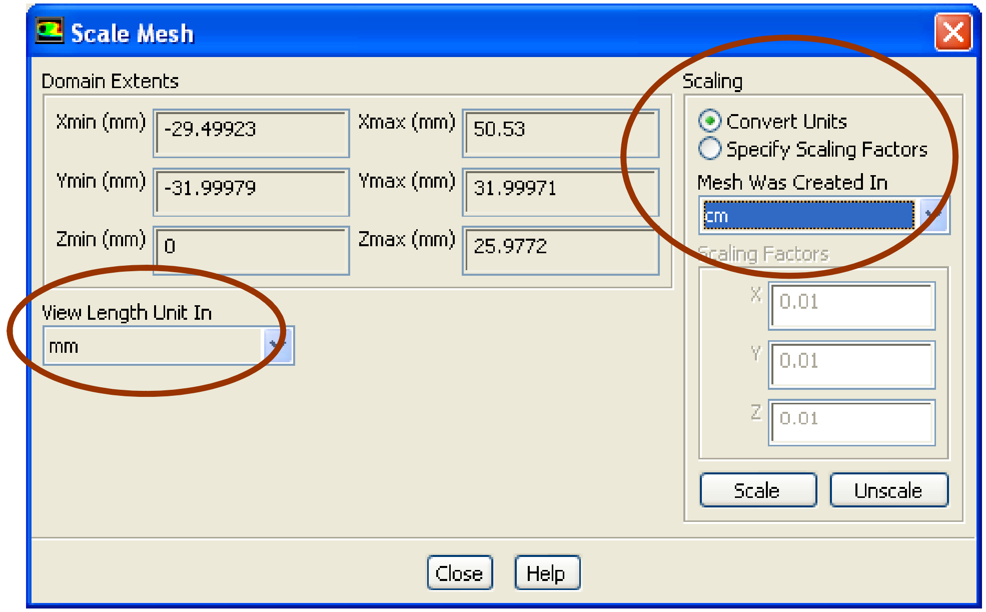
**Sample Journal File**

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

# Scaling the Mesh and Selecting Units

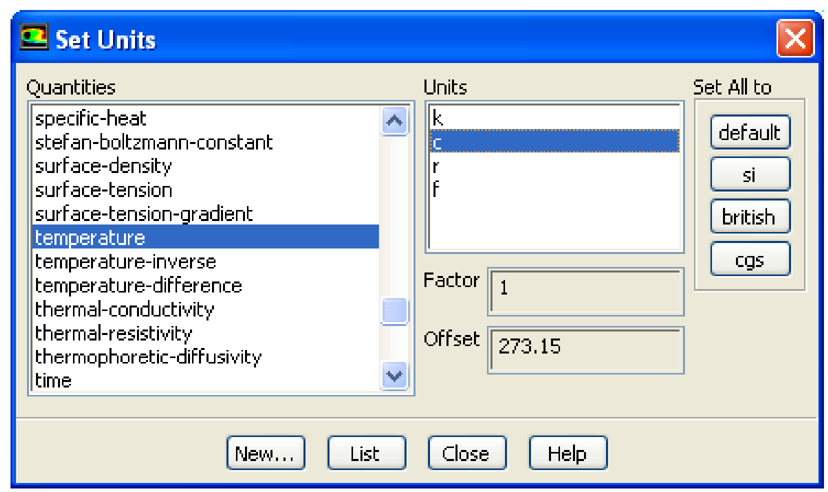


- When FLUENT reads a mesh file (.msh), all dimensions are assumed to be in units of meters.
  - If your model was not built in meters, then it must be scaled.
  - **Always verify that the domain extents are correct.**




- When importing a mesh under Workbench, the mesh does not need to be scaled; however, the units are set to the default MKS system.

- 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, accessed from the top menu.



# Polyhedral Mesh Conversion

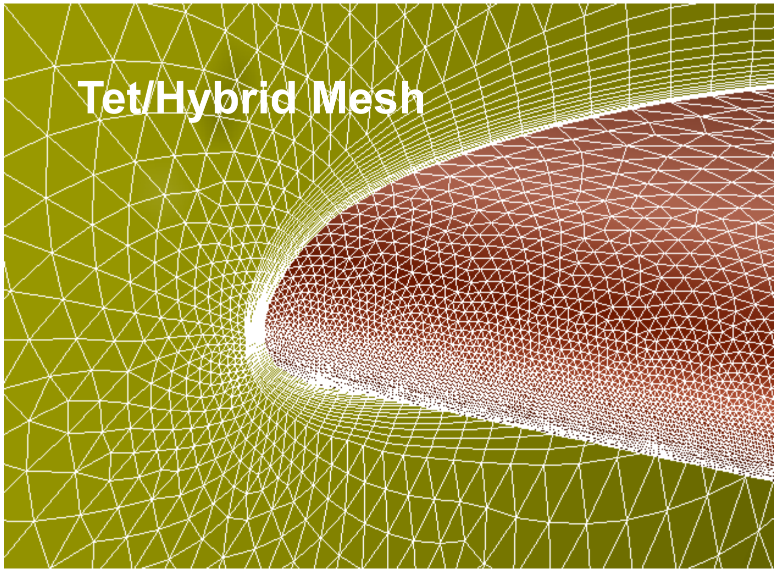


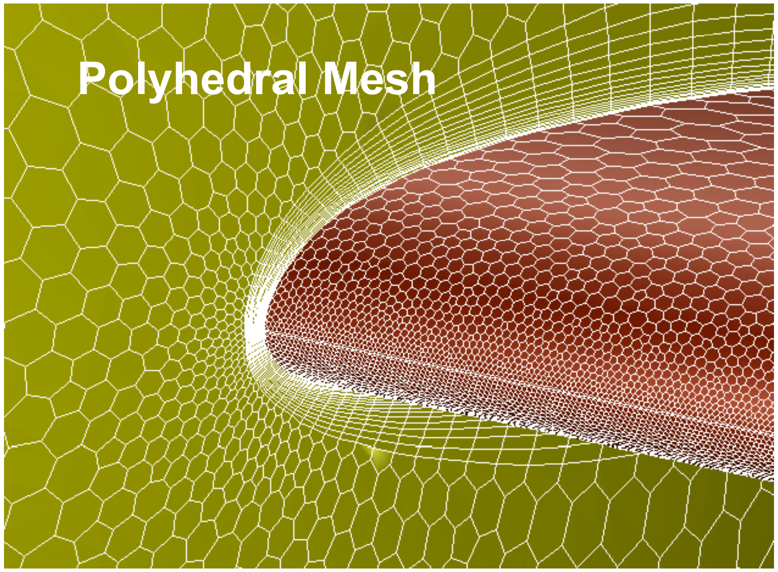
- A tetrahedral or hybrid grid can be converted to polyhedra in the FLUENT GUI (not in the preprocessor).
  - Generate a tetrahedral mesh then convert inside FLUENT.
  - Advantages
    - Improved mesh quality.
    - Can reduce cell count significantly.
    - User has control of the conversion process.
  - 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 Mesh menu:

  **`Mesh > Polyhedra > Convert Domain`**
  - Convert all cells in the domain (except hex cells) to polyhedra
    - Cannot convert adaped meshes with hanging nodes
  - Convert only highly skewed cells to polyhedra

    **`Mesh > Polyhedra > Convert Skewed Cells`**



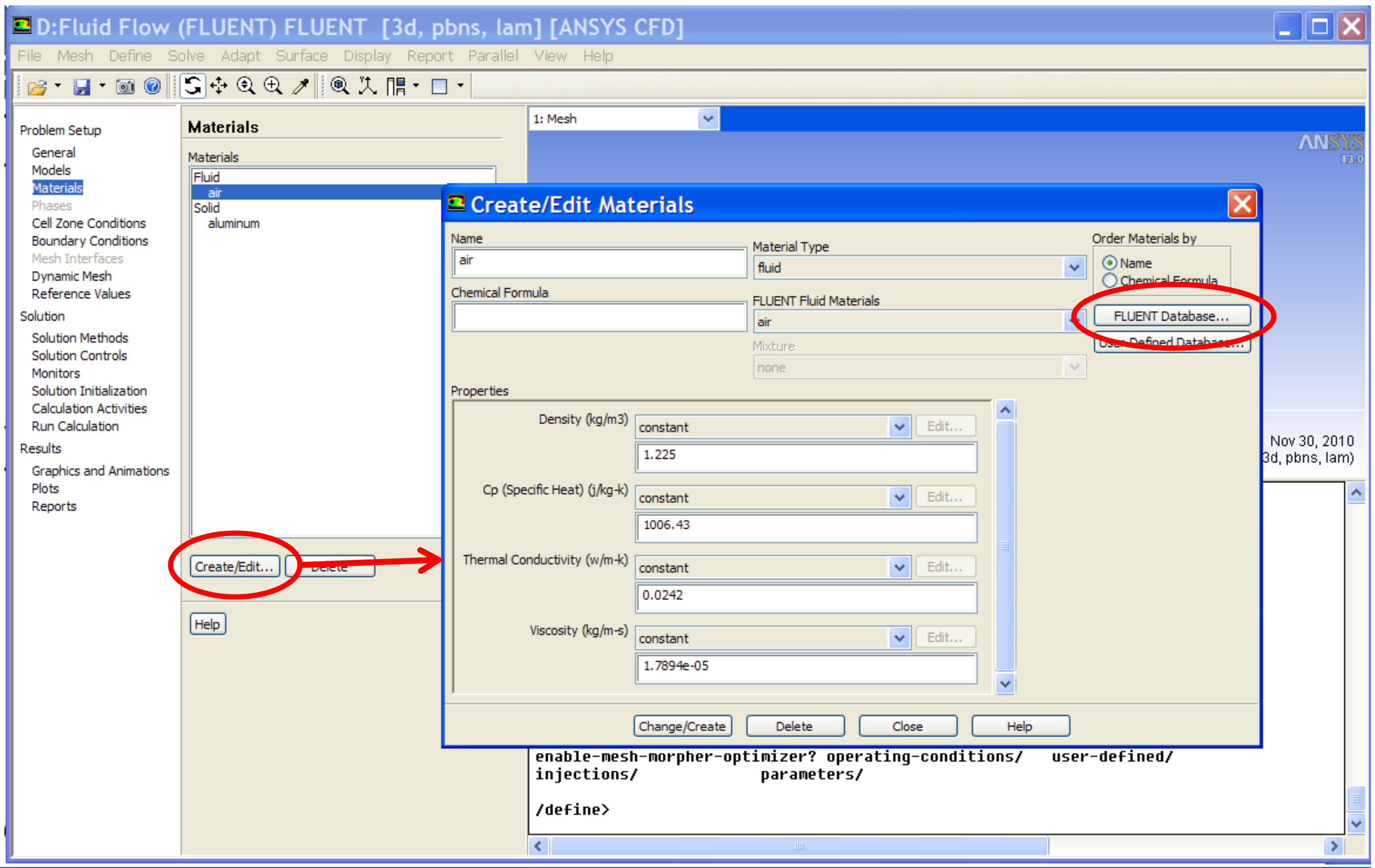
# Material Properties



- Material properties need to be defined for all fluids and solids to be simulated

- The parameters asked for will depend on the models selected for the simulation

- Many common materials are already defined in the 'FLUENT Database' and can easily be copied over to the model

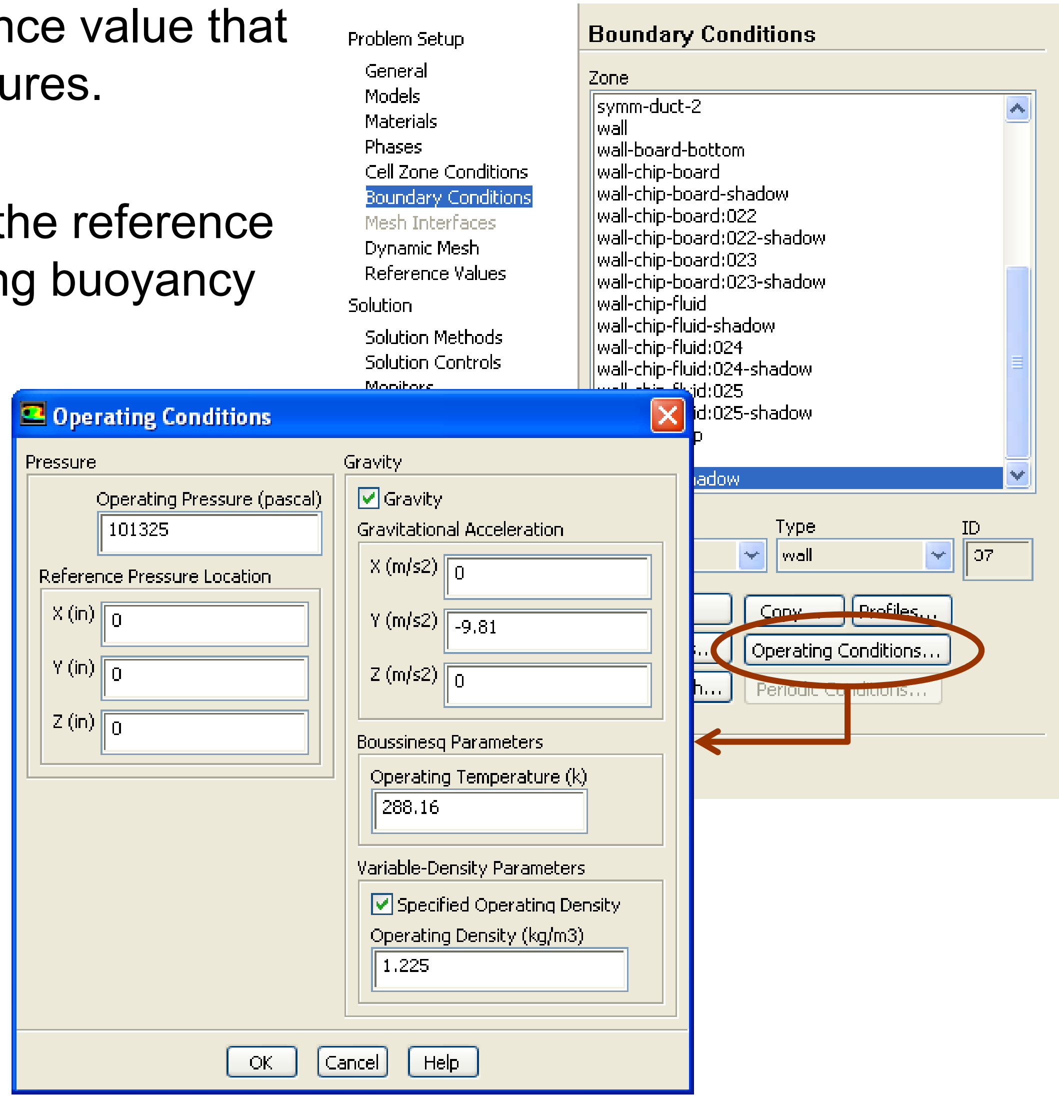



Note that these values may be either:
- Constants
- Functions of temperature
- Other built in functions following common relationships
- Defined by the user in a UDF.

# Operating Conditions



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

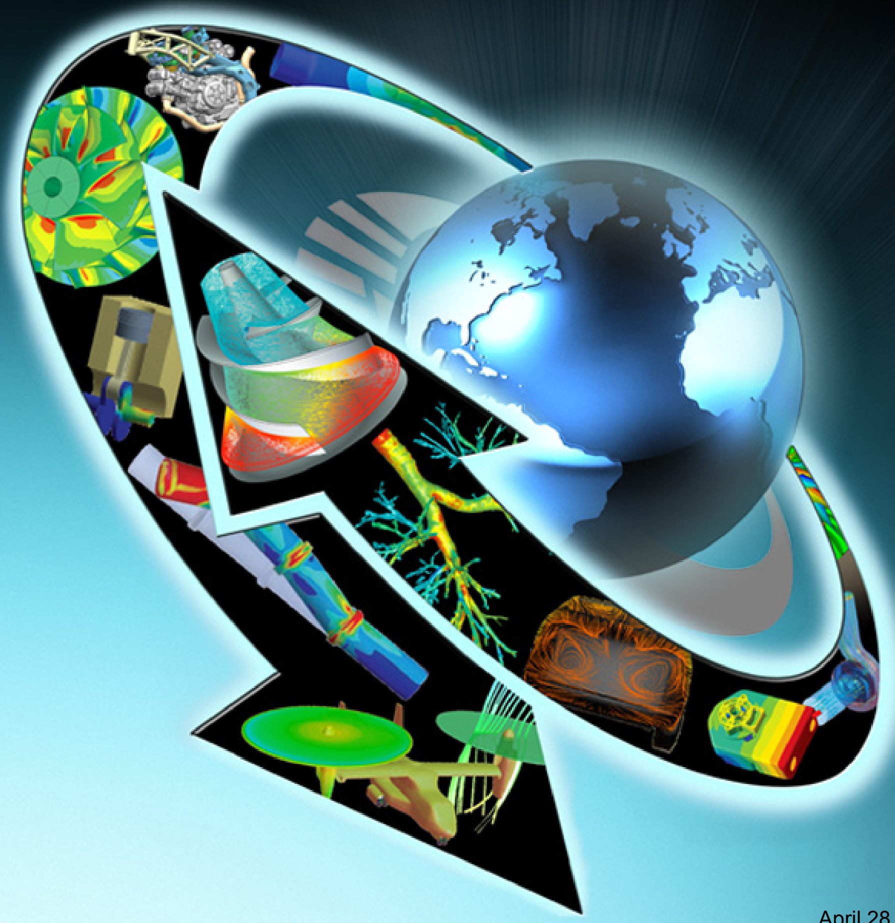


# Computing the Solution



- The remaining steps are covered in the subsequent lectures (and the practical workshops)
- Setting **Boundary conditions** (all outer boundaries to the fluid must be prescribed - for example velocity inlet / pressure outlet / symmetry / wall)
- **Solver Settings** (optimising settings for speed, accuracy and stability)
- **Initialisation** (initial values for the flowfield)
- **Solving** the problem, and checking for convergence
- **Postprocessing** the results.

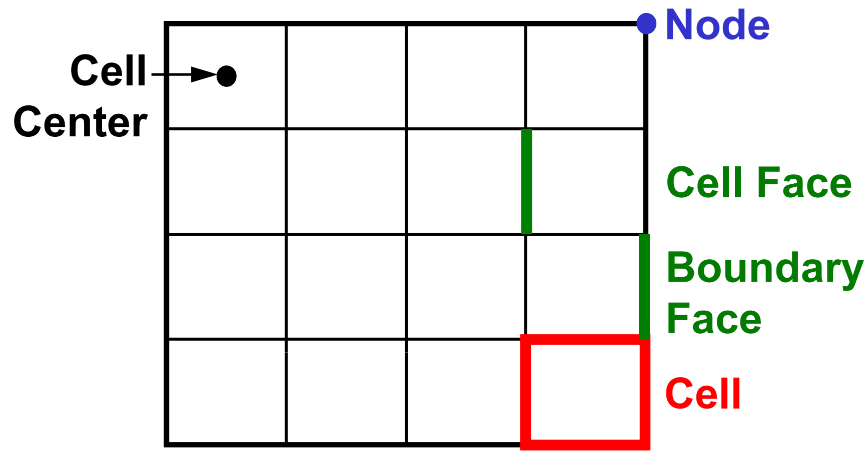
# Appendix : Additional notes



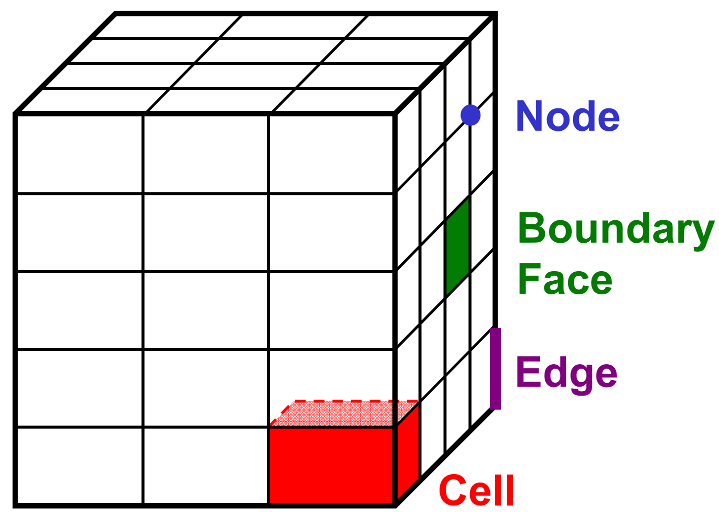
# Mesh Information and Hierarchy



- All mesh information is stored in the mesh file.
  - Node coordinates
  - Connectivity
  - Zone definition
- Similar to the way geometry is defined, mesh entities obey a hierarchy:
  - Node — Edge intersection / grid point
  - Edge — Boundary of a face (defined by two nodes
  - Face — The boundaries of cells, defined by a collection of edges
  - Cell — The control volumes into which the domain is discretized.
  - Zone — A collection of nodes, edges, faces or cells.
- The computational domain is defined by all members of the hierarchy
  - For fluid flow simulation only, the domain consists only of the fluid region.
  - For conjugate heat transfer or fluid-structure interaction problems, the domain needs to include any solid parts that are present.
- Boundary data is assigned to face zones.
- Material data and source terms are assigned to cell zones.




Simple 2D Mesh




Simple 3D mesh

# 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.

    **`Mesh > Reorder > Domain`**    **`Mesh > Reorder > Zones`**

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

    **`Mesh > Reorder > Domain`**

- 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

# 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 a coarse mesh to fine mesh.

    **`File > Write > Profile`**

    **`File > Read > Profile`**

- 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 > Interpolate`**

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