**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 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 post-processing.
- Very complex models can be set using a spreadsheet to generate the TUI commands.

**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 Define menu.

**Material Properties**

Fluent provides a standard database of materials and the ability to create a custom user-defined database.

Your choice of physical models may require multiple materials and dictate which material properties must be defined:
- Multiphase (multiple materials)
- Combustion (multiple species)
- Heat transfer (thermal conductivity)
- Radiation (absorption coefficient)

Material properties can be customized as function of temperature, mass fraction or pressure (density).

- Use of other solution variable(s) requires a User-Defined Function (UDF).

*Select Materials in the Project Setup tree
* Or Define → Materials... in the Menu bar
**Operating Pressure**

Represents the absolute pressure datum from which all relative pressures are measured.

\[ P_{\text{absolute}} = P_{\text{operating}} + P_{\text{relative}} \]

- Pressures specified at boundary conditions and initial conditions are relative to the Operating Pressure.
- Used to avoid problems with round-off errors which occur when the dynamic pressure differences in a fluid are small compared to the absolute pressure level.

**Cell Zones and Boundary Zones**

The mesh consists of a large number of finite volumes, or cells.

The cells are grouped into one or more cell zones.

- For instance, in a conjugate heat transfer calculation there may be one cell zone for the fluid region and a second cell zone for the solid material.

Each cell is bounded by a number of faces.

These faces are grouped into a number of faces zones.

Some of these faces are located on the boundaries of the model.

The zones to which such faces belong are called **boundary zones**.

**Opening the Cell Zone Conditions Panel**

In the Problem Setup tree, select **Cell Zone Conditions**.

A list of all cell zones will appear.

Select the zone of interest and click **Edit**.

- The cell zone conditions panel can be used to define a porous zone, prescribe energy sources in solid and fluid zones, specify inputs for rotating machinery, fix the values of one or more solution variables and many other operations.
**Boundary Conditions**

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

- Specify fluxes of mass, momentum, energy, etc. into the domain.

Poorly defined boundary conditions can have a significant impact on your solution.

**Defining boundary conditions involves:**

- Identifying types (e.g., inlets, walls, symmetry, ...)
- Identifying location
- Supplying required data depending on boundary type, location and physical models

**Choice depends on:**

- Geometry
- Availability of data at the boundary location
- Numerical considerations

---

**Setting Boundary Condition Data**

Explicitly assign data in BC panels:

- To set boundary conditions for particular zones:
  - Select Boundary Conditions in the project tree
  - Choose the boundary name in the zone list
  - Click the Edit... button

Boundary condition data can be copied from one zone to another.

Boundary conditions can also be defined by User-Defined Functions (UDFs) and profiles:

- Profiles can be generated by:
  - Writing a profile from another CFD simulation
  - Creating an appropriately formatted text file with boundary condition data
  - See Appendix for details of UDFs
  - See Appendix for details of using profiles

---

**Available Boundary Condition Types**

**External Boundaries**

- General
  - Pressure Inlet
  - Pressure Outlet
- Incompressible
  - Velocity Inlet
  - Outflow (not recommended)
- Compressible
  - Mass Flow Inlet
  - Pressure Far Field
- Other
  - Wall
  - Symmetry
  - Axis
  - Periodic
- Special
  - Inlet / Outlet Vent
  - Intake / Exhaust Fan

**Internal Boundaries**

- Fan
- Interior
- Porous Jump
- Radiator
- Wall outlet
ANSYS Fluent Workflow / Solver Settings

Navigation Pane Guides Basic Workflow
- Read and check mesh (scale if needed)
- Select Physical Models
- Energy
- Turbulence
- Multiphase
-...
- Create/Assign Materials
- Assign Cell & Boundary Conditions
  - Choose Solver Settings
  - Create Solution Monitors
  - Initialize Solution
  - Run Calculation
  - Post-Process Results

Solution Procedure Overview

Setting the solution parameters
- Initialize the solution
- Enable the solution monitor of interest
- Calculate a solution
- Check for convergence
- Accept

Choosing a Solver

Fluent has two types of solver, pressure-based and density-based.
- Pressure-Based is the default and should be used for most problems
  - Handles the range of Mach numbers from 0 to ~2
  - Density-Based is mainly used for higher Mach numbers, or for specialized cases such as capturing interacting shock waves
**Spatial Discretization Settings**

Use of the default settings for spatial discretization is recommended for most cases.

- For natural convection problems, where gravity has been activated, the pressure discretization must be changed to PRESTO or Body-Fitted Weighting.

**Solution Methods**

- Pressure-velocity coupling method
  - Scheme
  - Cell

- Spatial discretization
  - Gradient
  - Laplacian Coefficients
  - Pressure
  - Second order

**Under-relaxation Factors**

Implicit under-relaxation factors are used for SIMPLE, SIMPLEC, PISO:

- The under-relaxation factor, α, is included to stabilize the iterative process for the pressure-based solver.
- The final converged solution is independent of the under-relaxation factor.
- Only the number of iterations required for convergence is dependent.

**Initialization**

Fluent requires that all solution variables be initialized before starting iterations:

- Basically, this means that in every individual cell in the mesh, a value must be assigned for every solution variable to serve as an initial guess for the solution.
- A realistic initial guess improves solution stability and accelerates convergence.
- In some cases, a poor initial guess may cause the solver to fail during the first few iterations.

5 initialization methods are available:

- Hybrid initialization (default)
  - Use this for most cases.
- FMG initialization
  - Provides a more realistic initial guess, but the initialization process takes much longer than other methods.
- Can be especially beneficial for compressible flows and rotating machinery.
- Standard initialization
- Patch values
- Starting from a previous solution.
**Run Calculation – steady**

- Iterations solution will run
- Continues from current solution
- Solution will stop sooner if convergence checks are met

**Convergence**

The solver must perform enough iterations to achieve a converged solution.

At convergence, the following should be satisfied:

- All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance (Residual).
  - The residual measures the imbalance of the current numerical solution and is related to but not equal to the numerical error.
- Overall mass, momentum, energy, and scalar balances are achieved.
- Target quantities reach constant values.
  - Integral: e.g., Pressure drop.
  - Local: e.g., Velocity at specified position.

**Monitoring convergence using residual history**

- Generally, a decrease in residuals by three orders of magnitude can be a sign of convergence.
- Scaled energy residual should decrease to $10^{-6}$ (for the pressure-based solver).
- Scaled species residual may need to decrease to $10^{-7}$ to achieve species balance.

**Best practice** is to also monitor quantitative variables to decide convergence.

- Ensure that overall mass/heat/species conservation is satisfied.
- Monitor other relevant key variables/physical quantities for confirmation.
  - The use of one or more monitors like this is strongly recommended for all simulations.
Convergence Monitors – Residuals

- Residual plots show when the residual values have reached the specified tolerance. It is possible to modify or disable the default checking criteria for convergence.
- Prevents calculation being stopped prematurely.

All equations converged.