CFD Simulations in Fluent

Basics of CFD and Operations in Fluent

Outline of Presentation

- Basic concepts of Numerical Calculations
- Pre-processing in Fluent
- Solutions
- Post processing in Fluent
- Programming: Journaling, Scripting, UDF

Basic concepts of Numerical Calculations

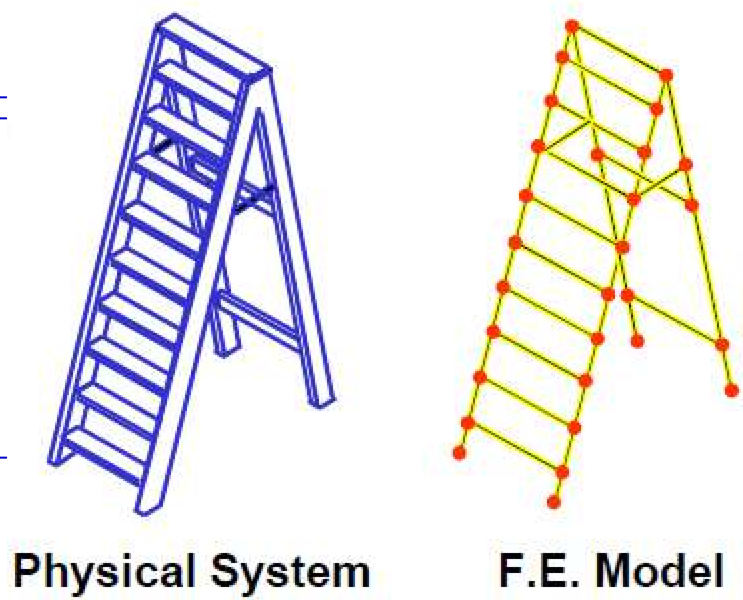
Analytical vs. Numerical Approach

Analytical Calculation vs. Numerical Calculation

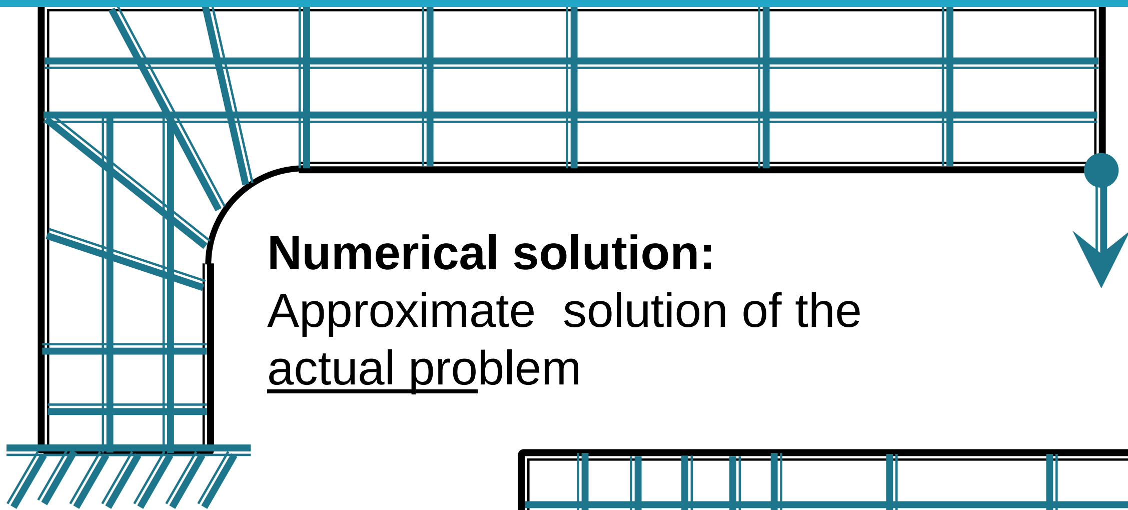
- **Analytical Results**
  - Available as explicit or implicit form of an equation such as a quadratic equation.
  - It is a continuous equation, available at each point is space **(Infinite Unknowns)**

- **Numerical Calculation**
  - Available at discrete location in space / time dimensions **(Finite Unknowns)**
  - It is an inherently discontinuous approach with some averaging / blending to ensure physical correctness

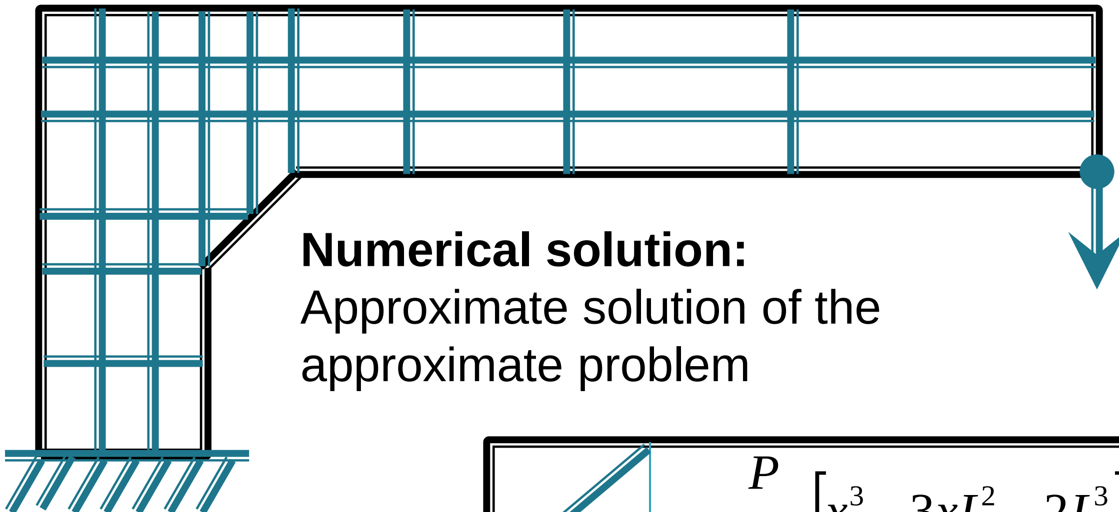
- **Examples**
  - Simply supported beam - Analytical
  - Plate temperature distribution - Numerical



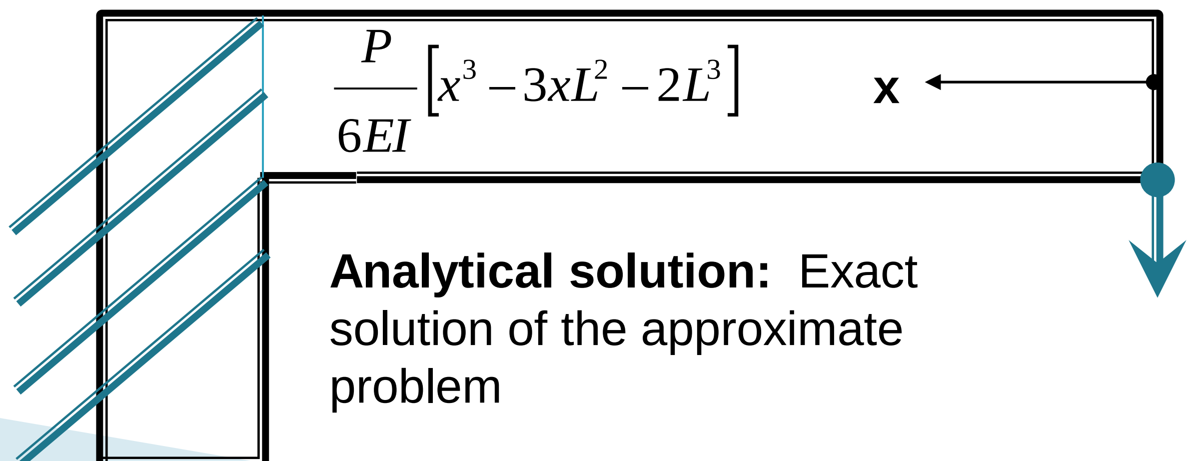
Analytical Calculation vs. Numerical Calculation



Numerical solution:
Approximate solution of the actual problem



Numerical solution:
Approximate solution of the approximate problem

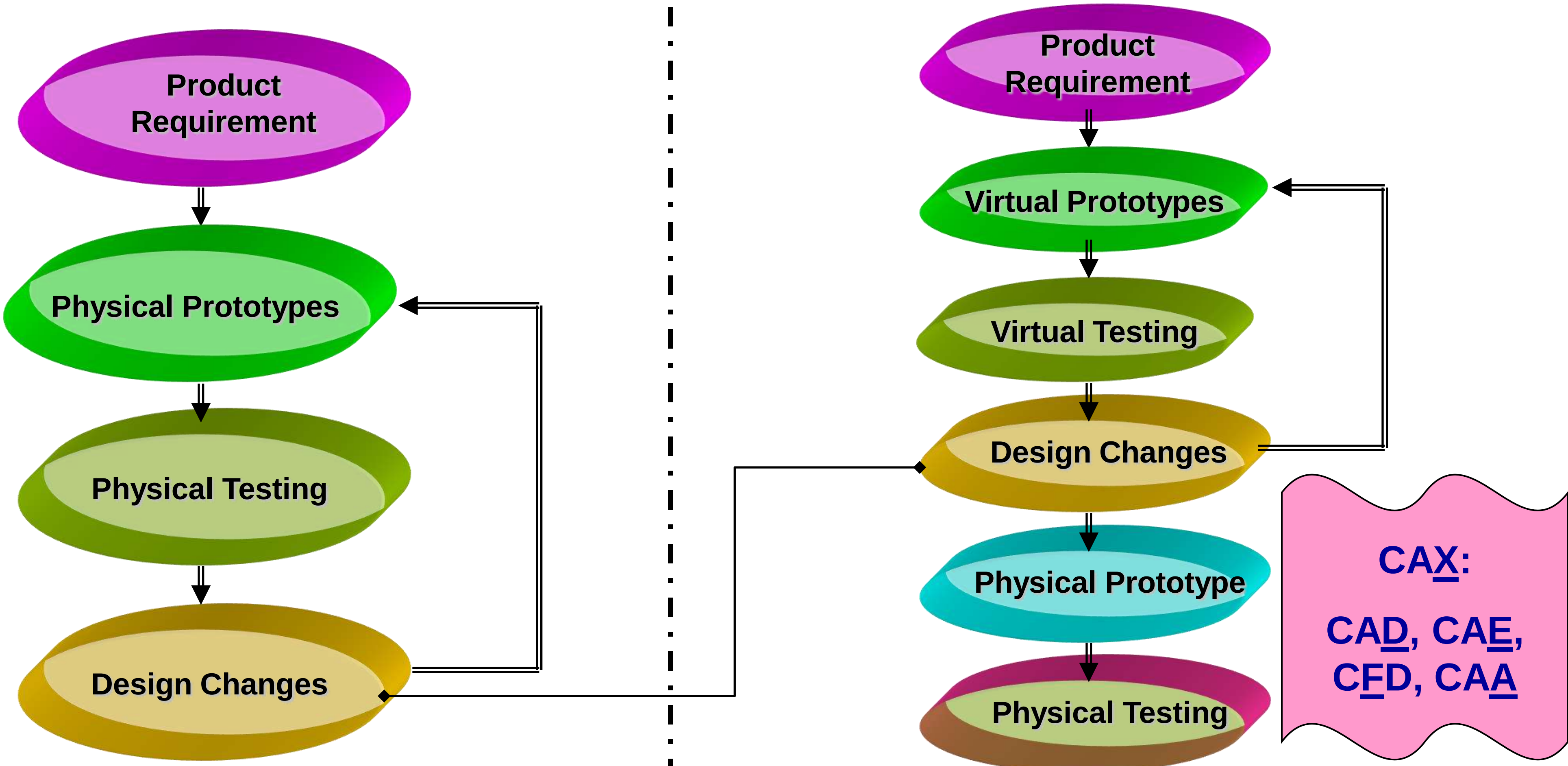


$$\frac{P}{6EI}\left[x^3 - 3xL^2 - 2L^3\right]$$

Analytical solution: Exact solution of the approximate problem

Traditional vs Virtual Design Evaluations

Traditional Design Methodology | **Modern Design Methodology**



- Long Product Development Cycle
- Many prototypes
- Very Costly

- Minimal Prototypes and Cost Effective
- Product Development Cycle Drastically Reduced
- Better Insight into various Design Aspects

Typical Flow Chart for Numerical Analysis

Pre-processing

- Define Input / Output - Assumptions based on Engg. Judgment
- Prepare CAD Model, De-featuring & Simplification for CFD meshing
- Generate Mesh and Define Zones / Patches – inlet, outlet, wall, symmetry …
- Define Thermodynamic and Transport Properties of Material
- Specify Boundary and Initial Condition

Solver Setting & Solution

- Select flow model: Laminar, Turbulent, Single-phase, Multi-phase …
- Select appropriate turbulence model based on flow type, wall function
- Specify discretization schemes, time steps …
- Analysis Run: Batch Mode (command line), GUI, Remote Server

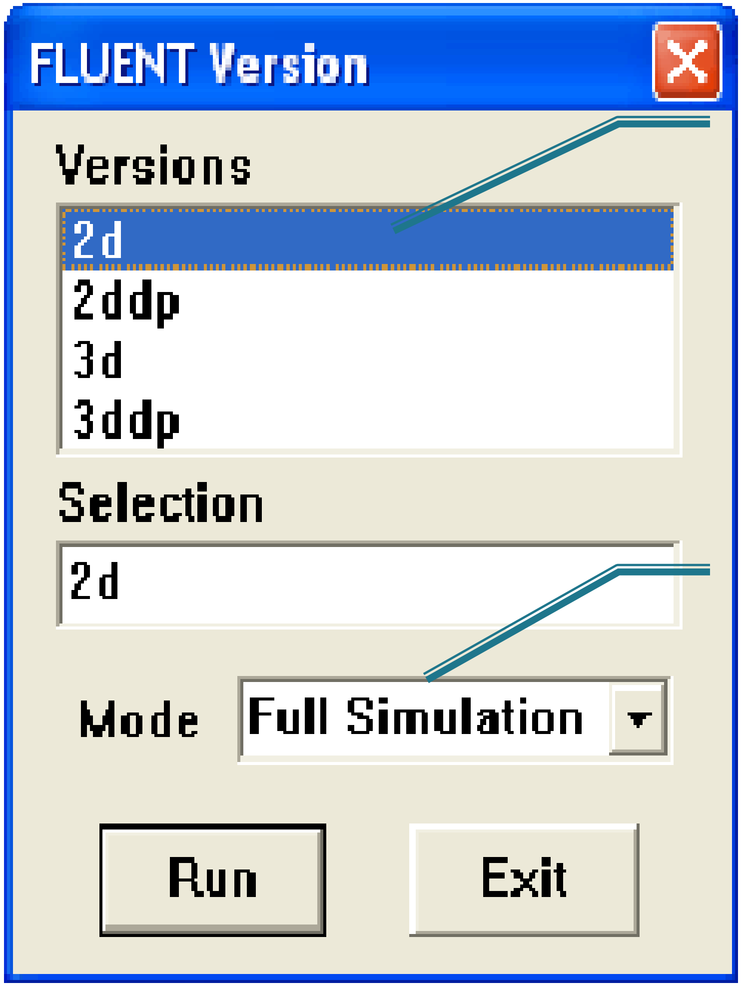
Post-processing

- Contour Plots, Velocity Vectors, Streamlines, Pathlines, Isolines …
- Animations for Streamlines, Particle Tracks, Coefficients of Pressure …
- Report Preparation

Pre-processing in Fluent

Local Server: GUI-operations and their meaning

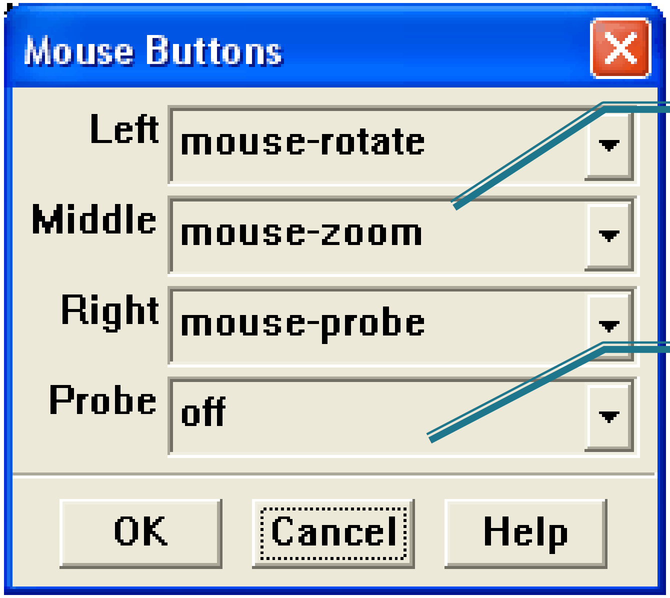
Starting Fluent



1. 2D or 3D, single precision or double precision (dp -> double precision)

2. Pre-processing, Solution and Post Process [Full Simulation] or just post-processing

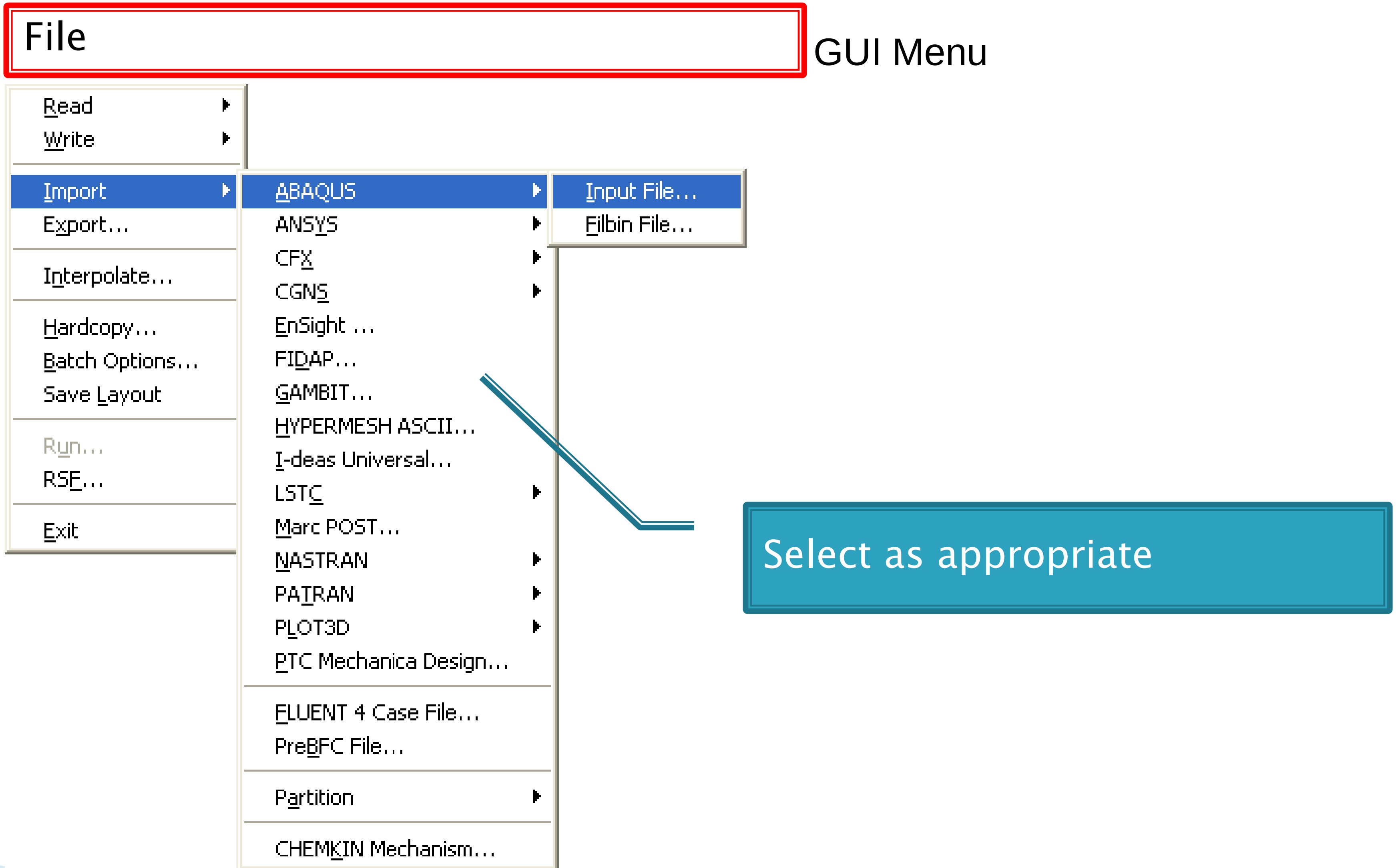
Setting of Mouse Button for PAN (MOVE) – ZOOM – ROTATE



Basic Settings

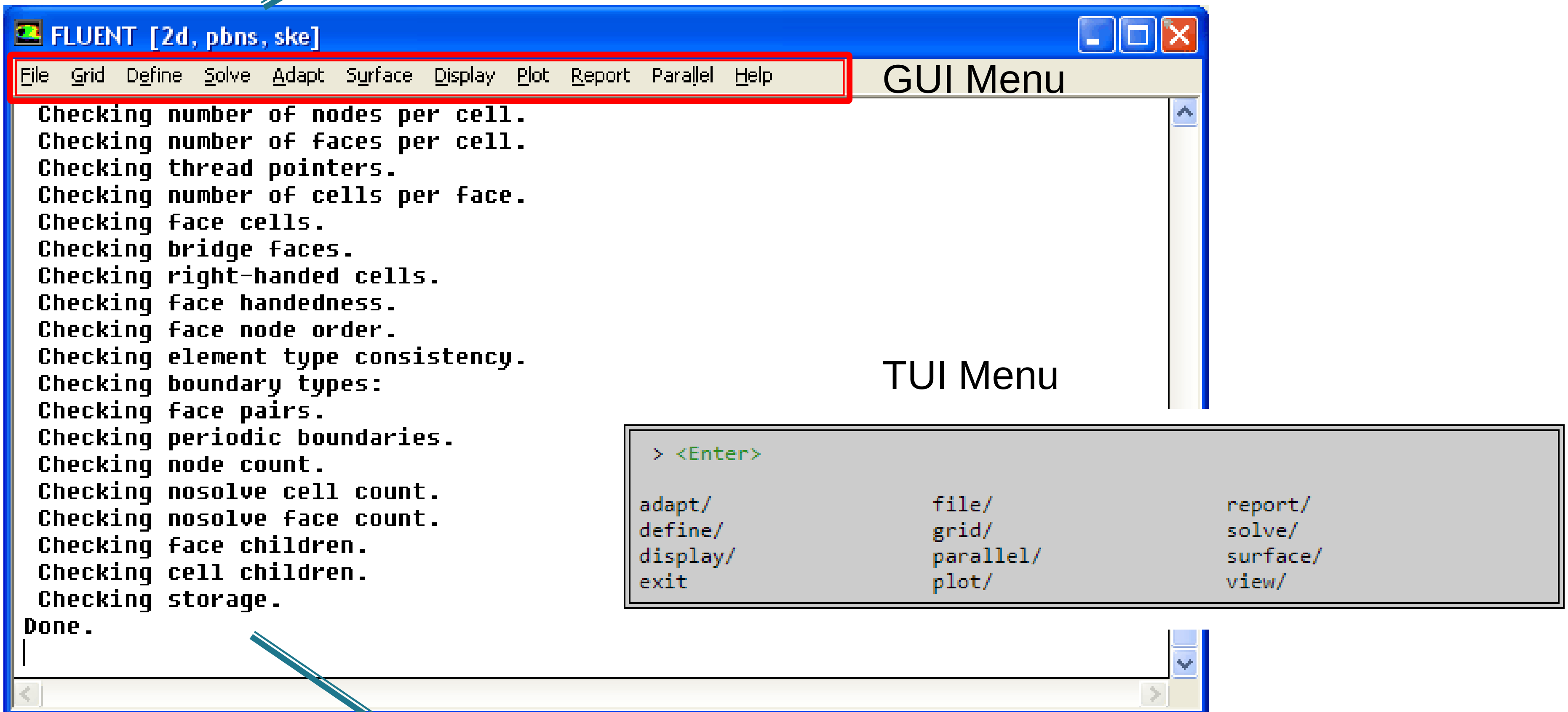
The probe option with print the information when clicked inside mesh

Reading Mesh: File → Import Mesh, Read Case …



Reading Mesh: File → Import Mesh, Read Case …

Top panel describes the summary of case file



Software operation summary. Note ‘done’ at the end! Any error will be reported here.

Check Mesh: Grid → Check

```
Grid Check

 Domain Extents:
   x-coordinate: min (m) = 0.000000e+00, max (m) = 6.400001e+01
   y-coordinate: min (m) = -4.538534e+00, max (m) = 6.400000e+01
 Volume statistics:
   minimum volume (m3): 2.353664e-05
   maximum volume (m3): 7.599501e-03
     total volume (m3): 2.341560e+00
   minimum 2d volume (m3): 4.027890e-04
   maximum 2d volume (m3): 1.230393e-03
 Face area statistics:
   minimum face area (m2): 1.300719e-04
   maximum face area (m2): 3.781404e-02
 Checking number of nodes per cell.
 Checking number of faces per cell.
 Checking thread pointers.
 Checking number of cells per face.
 Checking face cells.
 Checking bridge faces.
 Checking right-handed cells.
 Checking face handedness.
 Checking for nodes that lie below the x-axis.
 Checking element type consistency.
 Checking boundary types:
 Checking face pairs.
 Checking periodic boundaries.
 Checking node count.
 Checking nosolve cell count.
 Checking nosolve face count.
 Checking face children.
 Checking cell children.
 Checking storage.
Done.
```

Check at the bottom for error message

Check Mesh: Repair Shadow Zones in Periodic Mesh

WARNING: node on face thread 2 has multiple shadows.

This warning message appears only in case of periodic (translational of rotational) faces!

These faces can be repaired only through the Text User Terminal (TUI)

TUI: grid → modify-zones → repair-periodic

1. The program will automatically try to detect the periodic distance or angles though will ask to user inputs as well
2. The command can be shortened as: grid → mz → rp

Manipulate Mesh: Optional for ease of simulation

1. **Merging Zones:** combining multiple zones of similar type – process not fully reversible (de-merging to previous state not possible): keep back-ups
2. **Separating Zones:** Opposite of “Merging Zones” – required if say there are multiple outlets and all grouped into single zone in the meshing software.
3. **Creating Periodic Zones, Slitting Periodic Zones:** For periodic zones
4. **Scaling the Grid** – FLUENT is a metric solver. Scale the mesh appropriately to convert into meters. E.g. if mesh was generated in inch, scale factor = 0.0254
5. **Translating the Grid:** Move the grid in required to move near origin
6. **Rotating the Grid:** Rotate the mesh to orient to particular axis

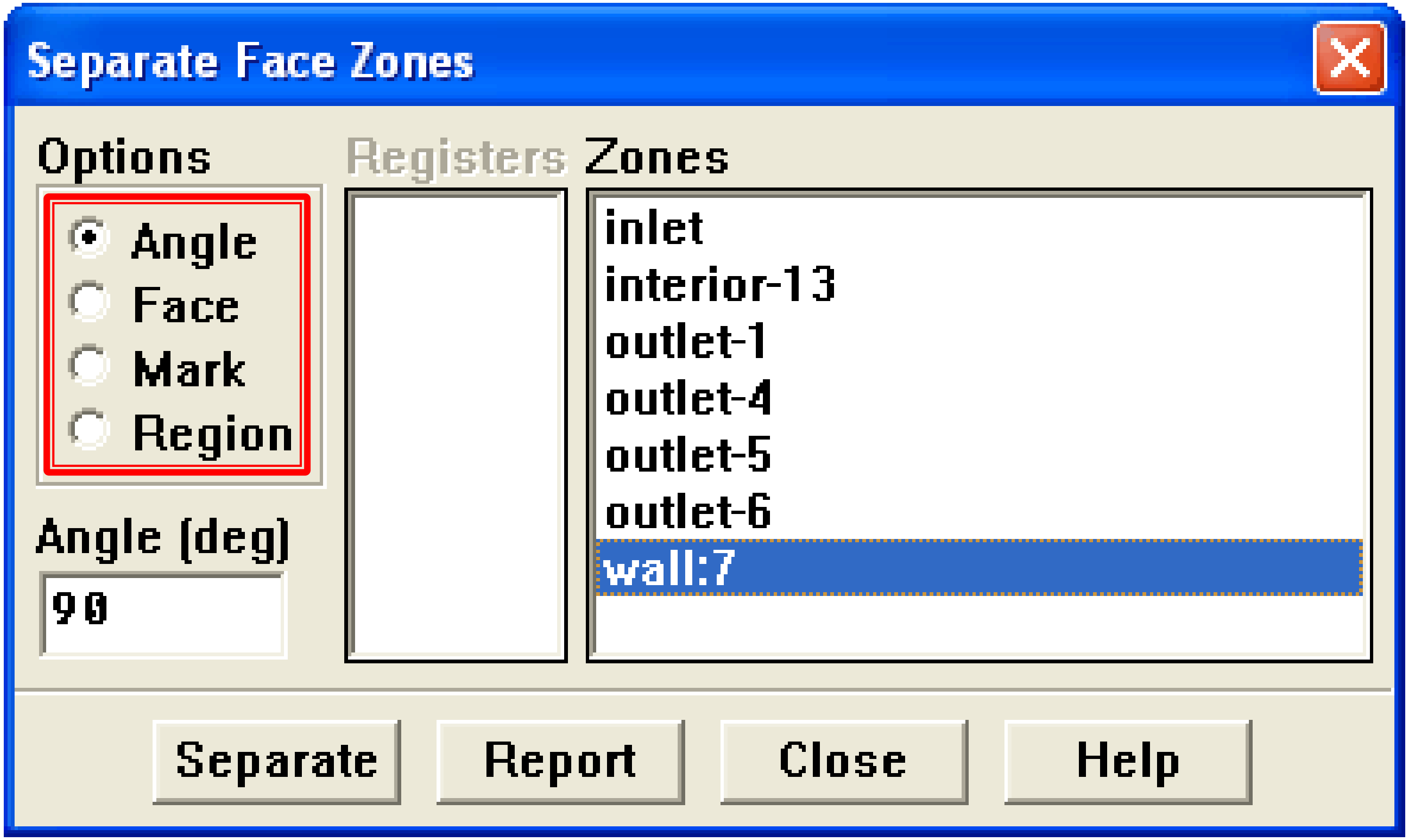
Manipulate Mesh: Optional for ease of simulation

Some other options for the sake of completeness

1. **Fusing Face Zones:** fuse boundaries (and merge duplicate nodes and faces) created by assembling multiple mesh regions.
2. **Slitting Face Zones:** Not same as separating a face zone! Slit an internal wall or coupled wall zone into two distinct uncoupled zones.
3. **Extruding Face Zones:** A face can be extruded to increase the domain size say changing location of the outlet to prevent reverse flow.
4. Replacing, Deleting, Deactivating, and Activating Zones
5. Reordering the Domain and Zones

Manipulate Mesh: Separate Face Zones

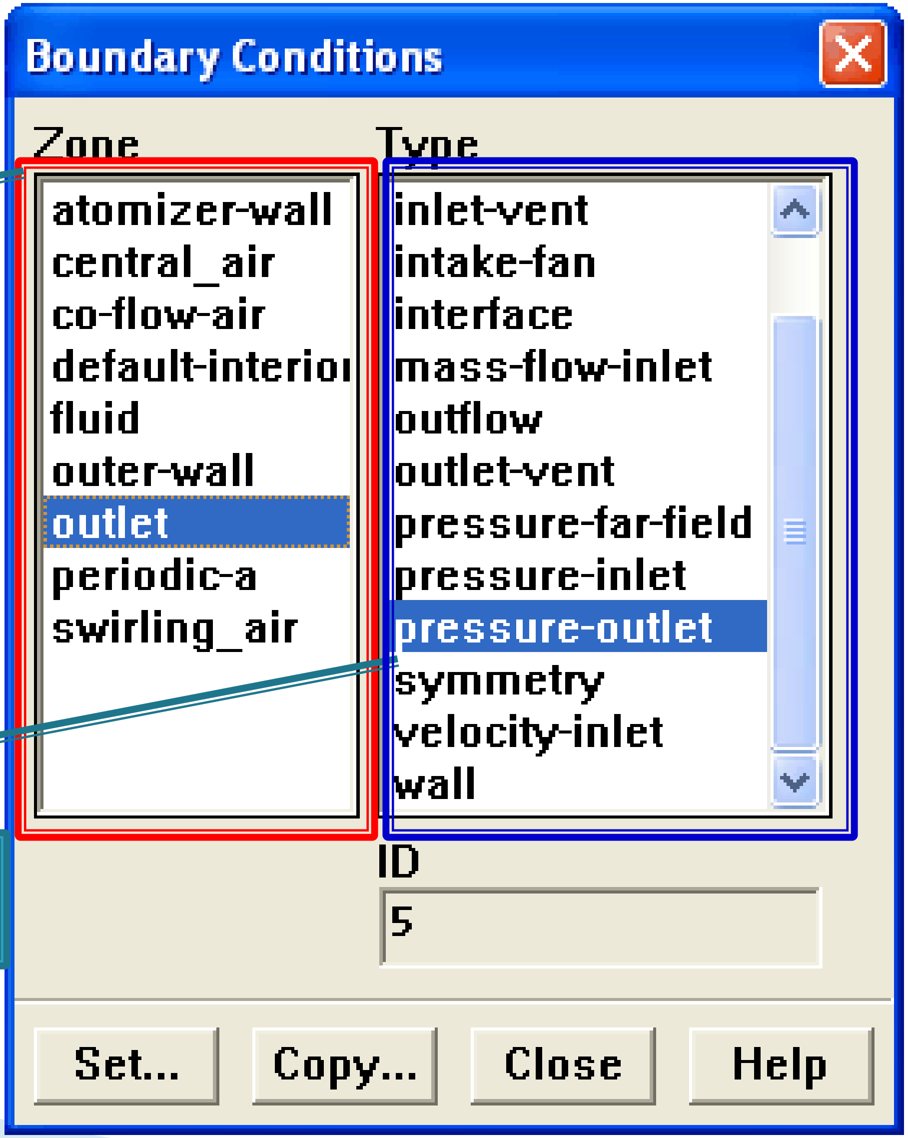
This feature is most used among all the options described earlier



Apply Boundary Conditions

Boundaries are representation of physical state of the computational domain!

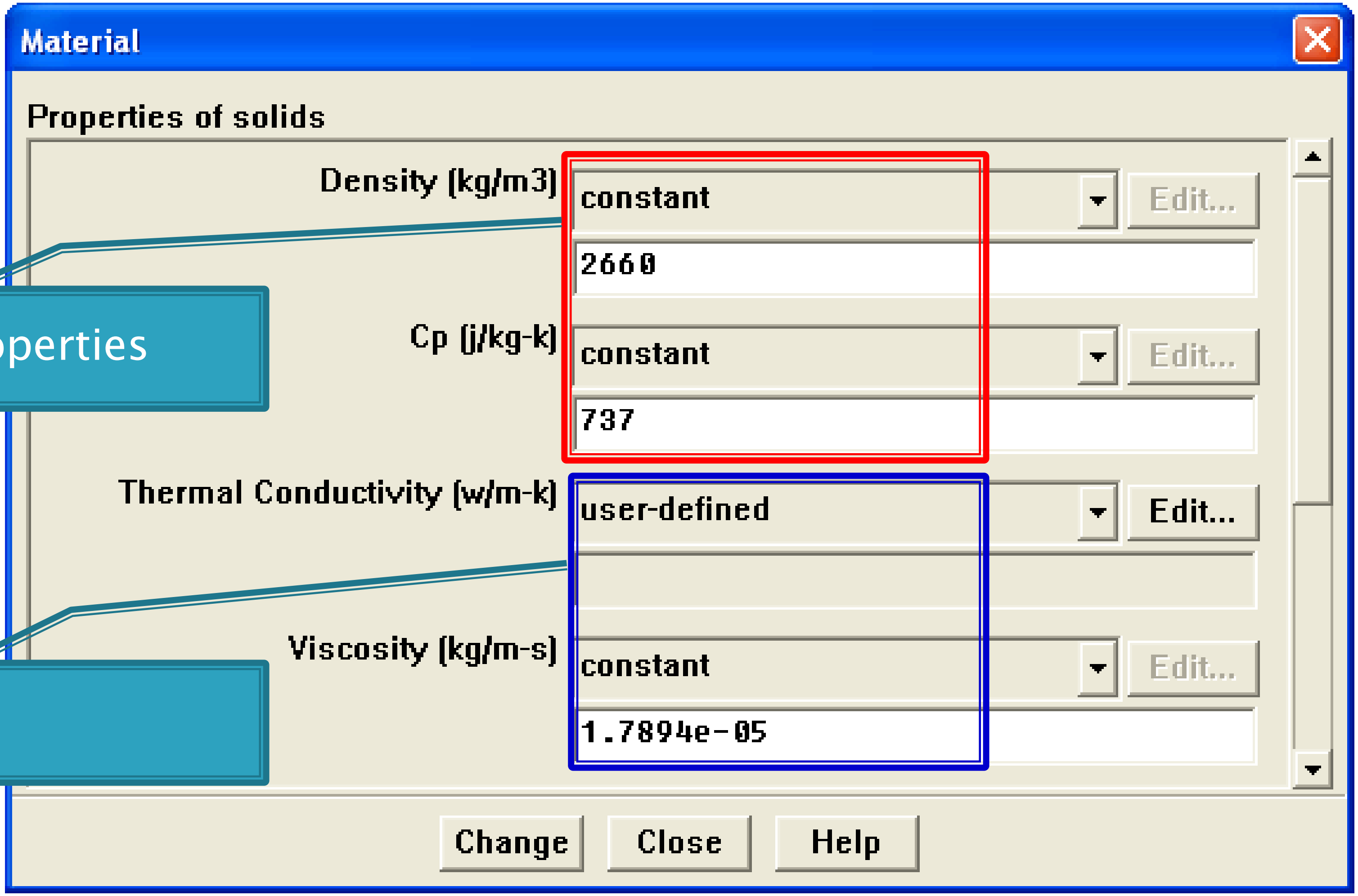
Zones defined by CFD Analyst



Boundary condition type available in FLUENT

Define Material Properties

Thermodynamic and transport properties of all the phases



Thermodynamics properties

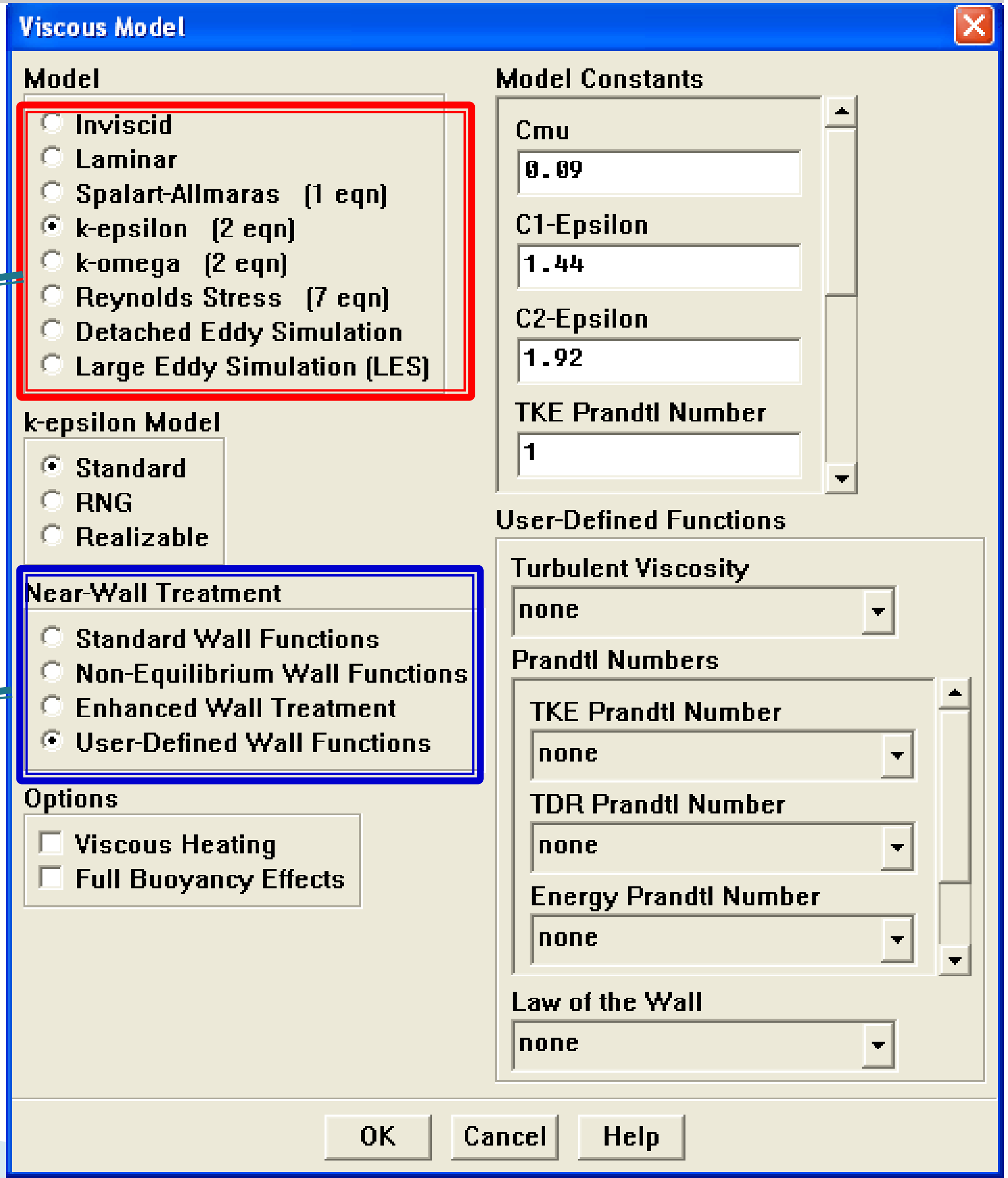
Transport properties

Define Turbulence Model

RANS: Reynolds-Averaged Navier Stokes and k-ε are the workhorse of industry!

Turbulence Models

Wall Function

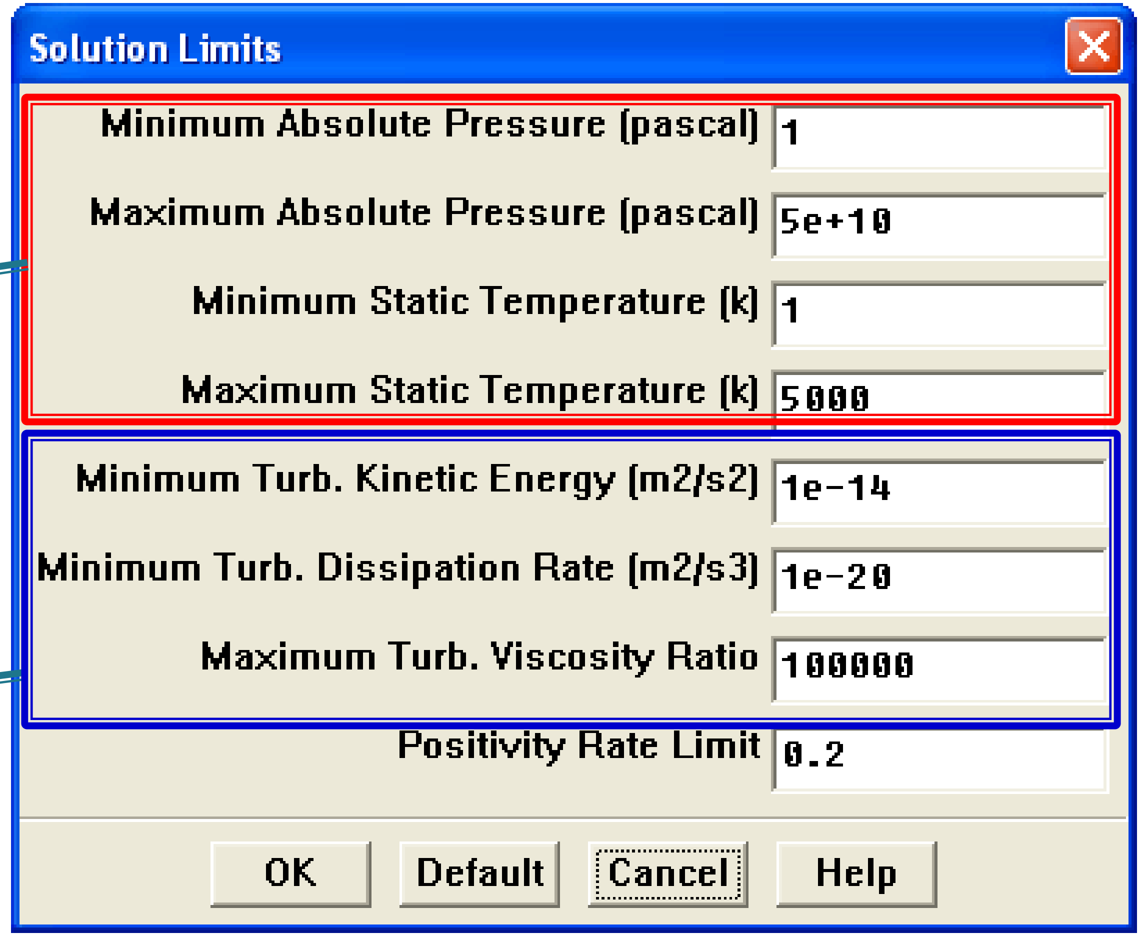


Solver Settings

Solver running, monitor and convergence parammeters

Define Solution Limits

If during computation value exceed these limits, solver will clip to range defined.



Limits of mean values

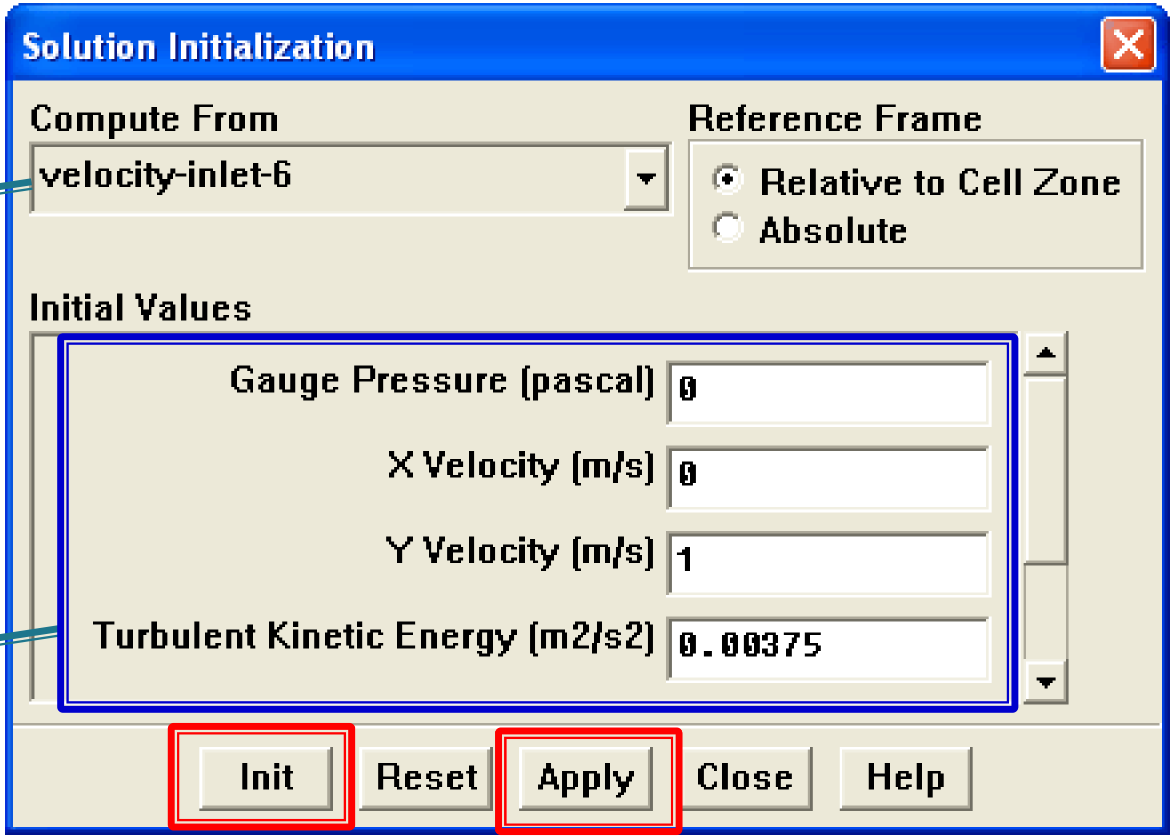
Limits of fluctuating values

Define Initial Values: solve → Initialize → Initialize …

A better guess helps improve the convergence sometimes!

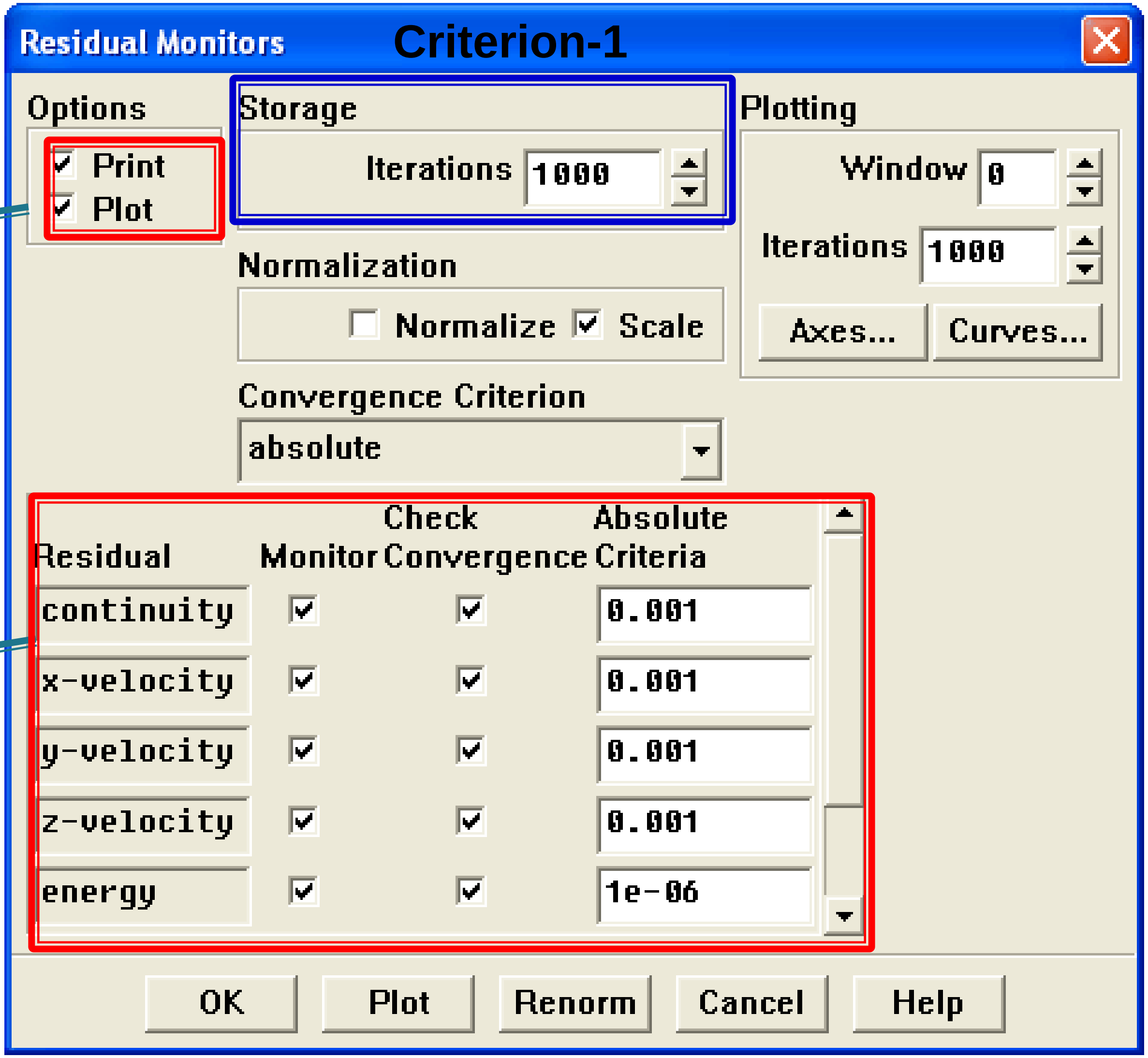
Based on selected boundary

Calculated value



Define Convergence Criteria: Solve → Monitors → Residual …

When the solver should stop running? Either criterion-1 or criterion-2 is met!



Print in the console and plot as graphical chart

Criterion-2

Post-processing

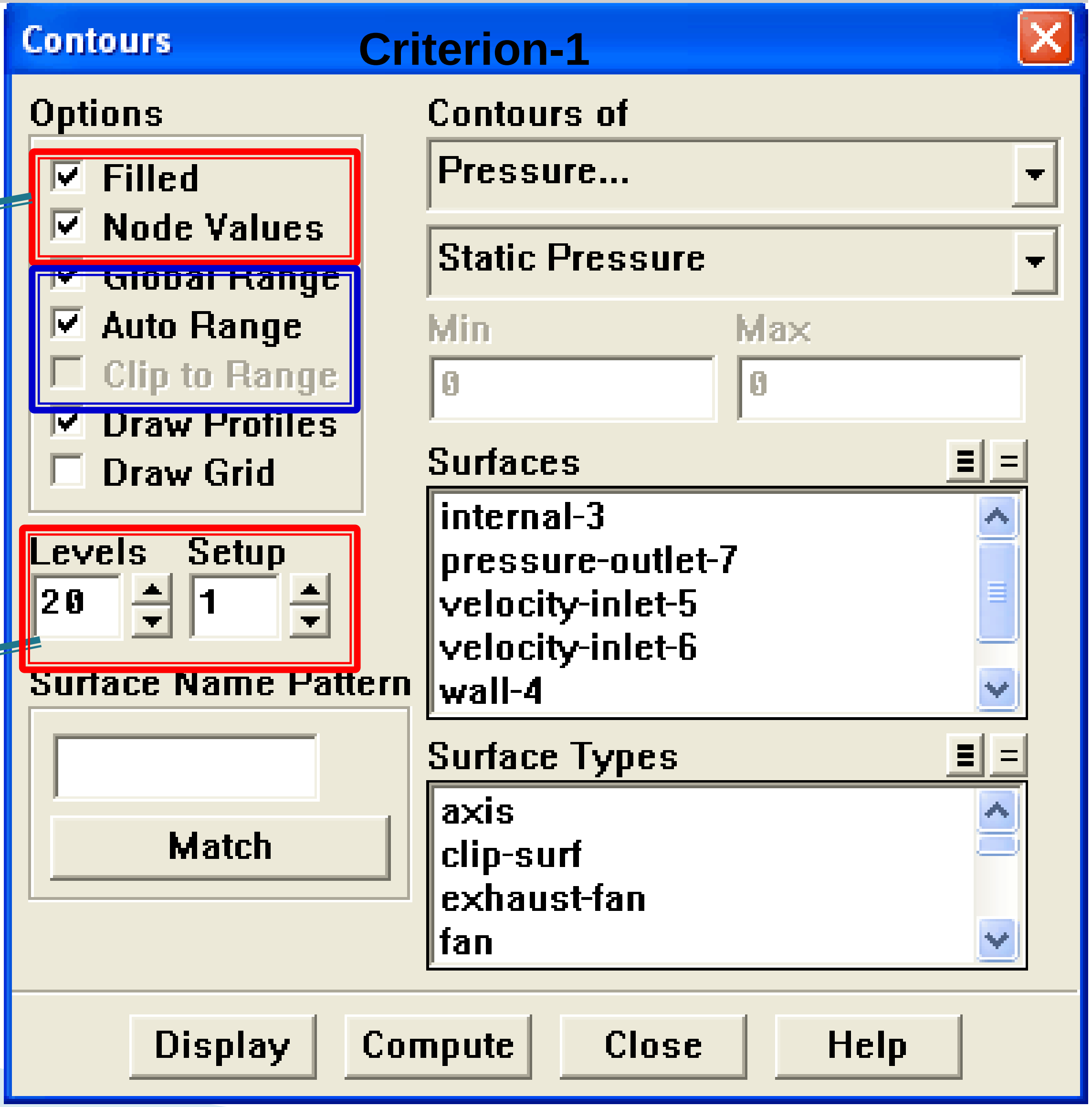
Qualitative plots, quantitative integration and averaging

Display Contour Plots: Display → Contours …

Contour: coloured representation of field variables on a plane or surface

How to show the plot and interpolate the results?

Number of sub-divisions in legend

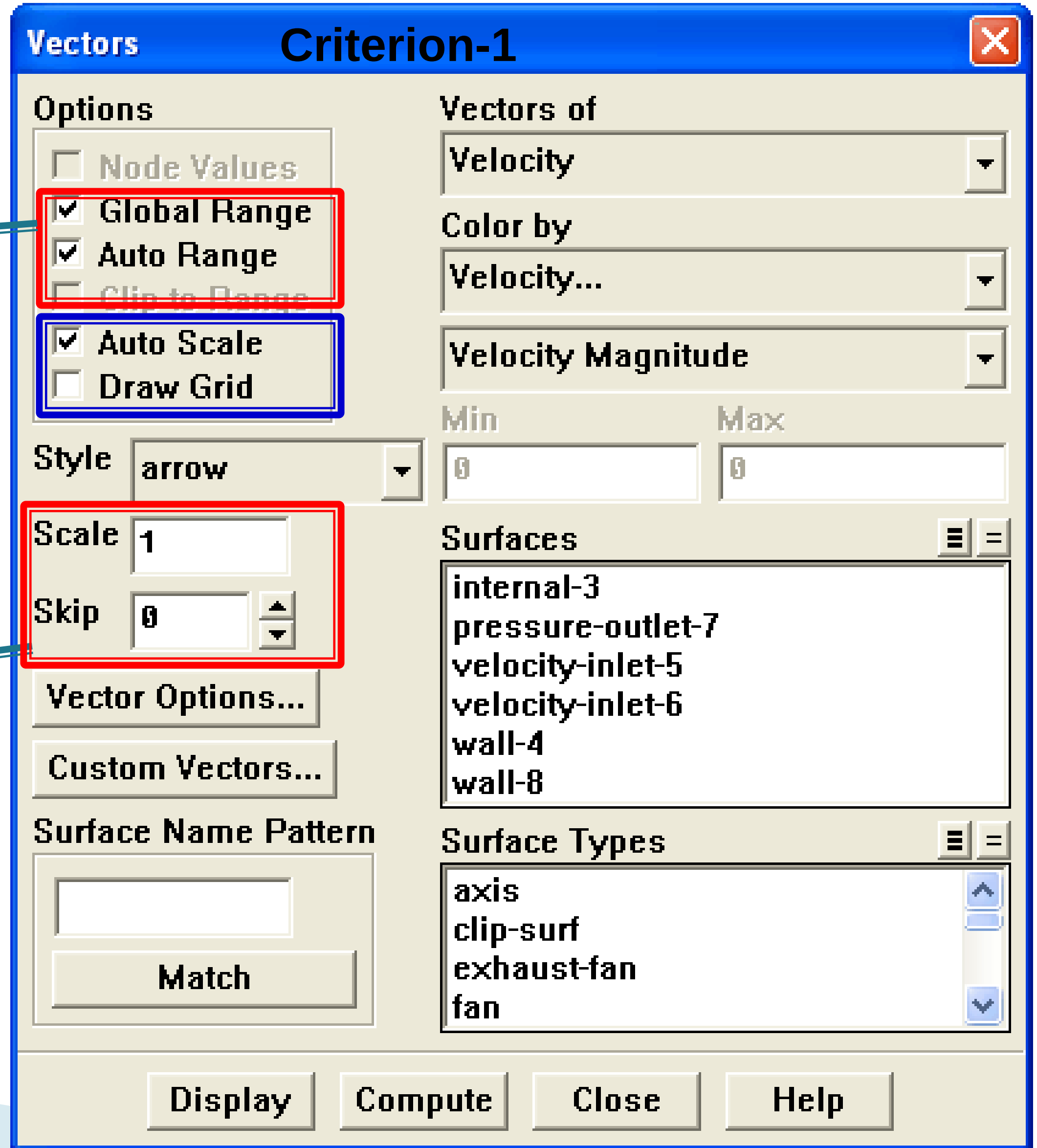


Display Vectors: Display → Vectors …

Vectors field represented as arrows

Range of plot and scale, whether to show mesh?

Vector scale and spatial distribution



Programming: Journaling, Scripting, UDF

Volumetric heat source, temperature dependent material properties …

UDF: User-Defined Function, Journals and Transcripts

1. FLUENT uses programming languages SCHEME (TUI), FORTRAN (back-end mathematics) and Tcl/Tk (GUI)
2. **UDF:** FLUENT is a general-purpose CFD simulation program and cannot address all the physical variations. UDF fills this gap.
3. **Journals and Transcripts** are similar recording of VBA scripts in EXCEL.
4. The details of this feature is covered under advance topic once you get mastery of the topics covered so far!

Thank you for your attention!

- Please visit http://www.cfdyna.com for explore more about CFD and related stuff.

- You may send e-mail to fb@cfdyna.com to get help on any advance topic!