

# 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 in space (**Infinite Unknowns**)

- **Numerical Calculation**

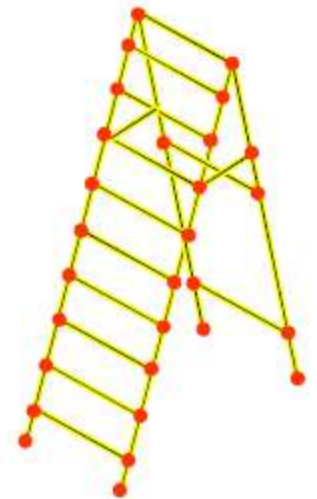
- Available at discrete location in space / time dimensions (**Finite Unknowns**)
- It is an inherently discontinuous approach with some averaging / blending to ensure physical correctness

- **Examples**

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

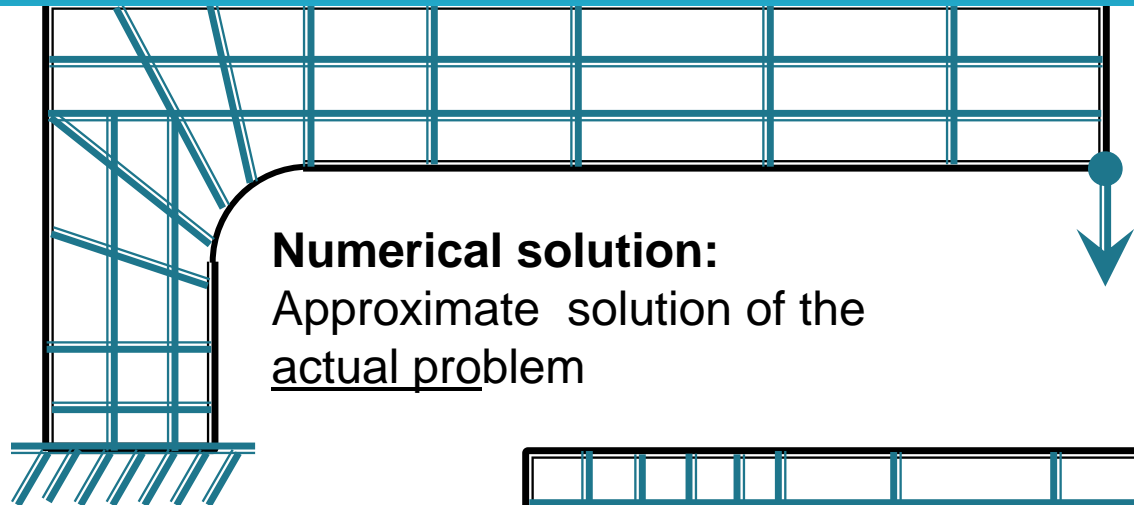


Physical System

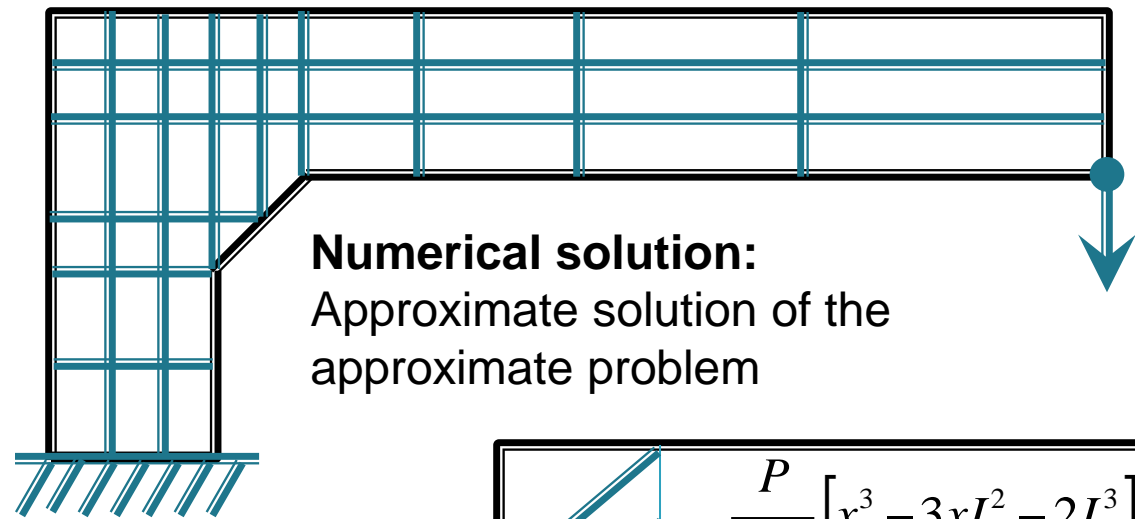


F.E. Model

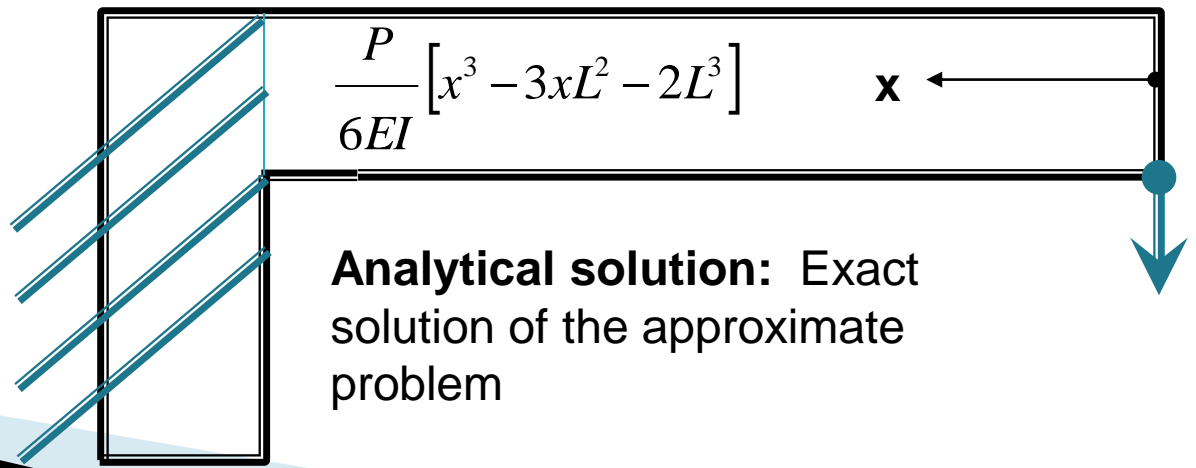
# Analytical Calculation vs. Numerical Calculation



**Numerical solution:**  
Approximate solution of the actual problem



**Numerical solution:**  
Approximate solution of the approximate problem

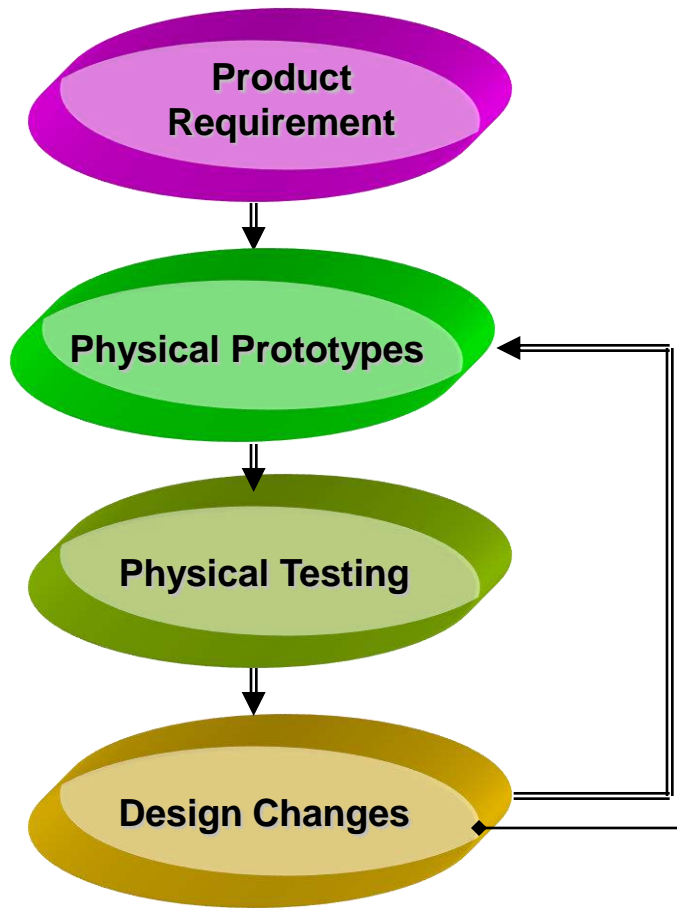


$$\frac{P}{6EI} [x^3 - 3xL^2 - 2L^3]$$

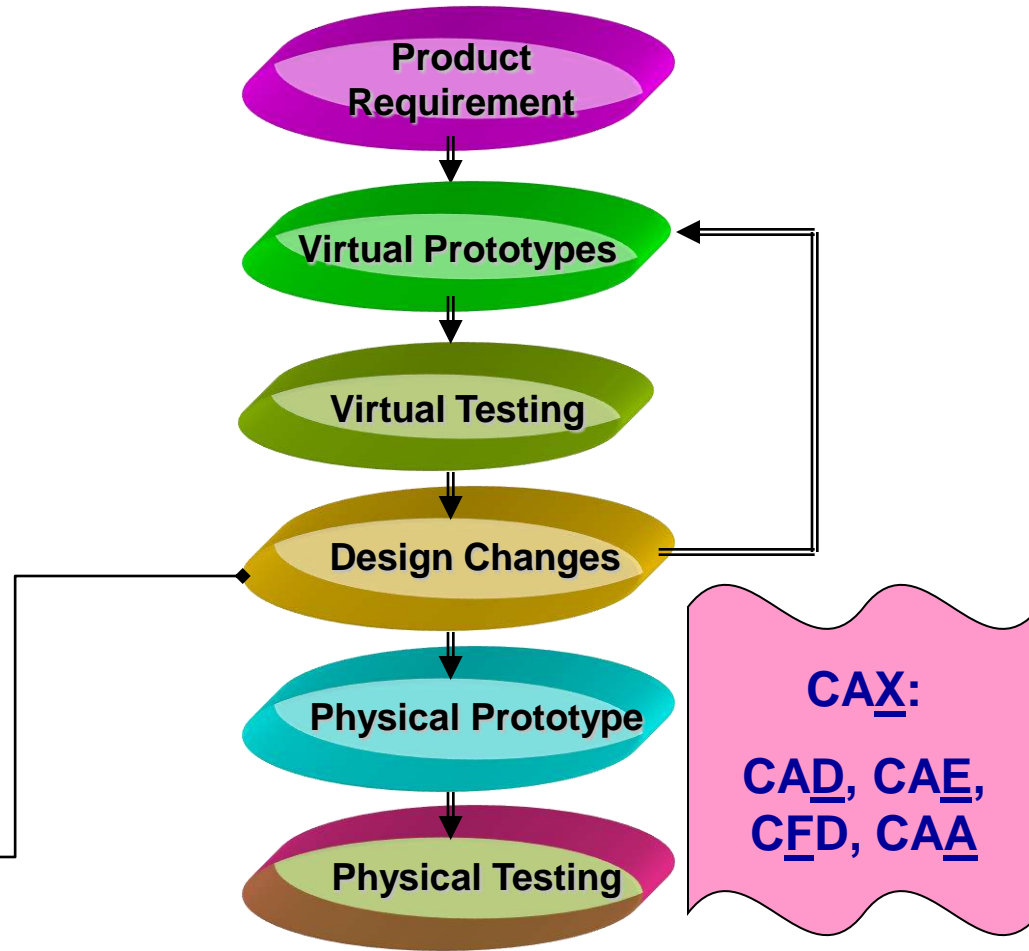
**Analytical solution:** Exact solution of the approximate problem

# Traditional vs Virtual Design Evaluations

## Traditional Design Methodology



## Modern Design Methodology

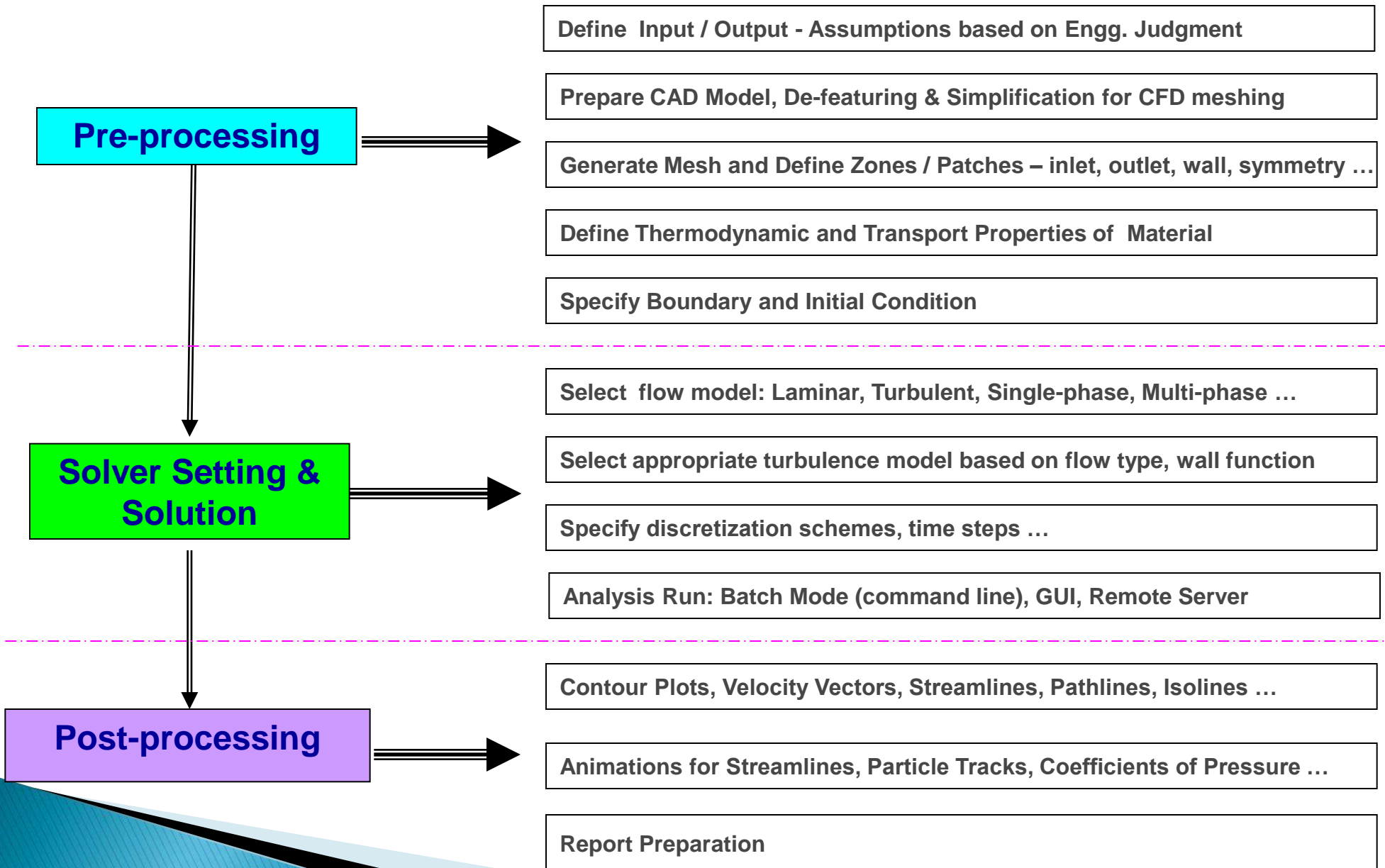


- Long Product Development Cycle
- Many prototypes
- Very Costly

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



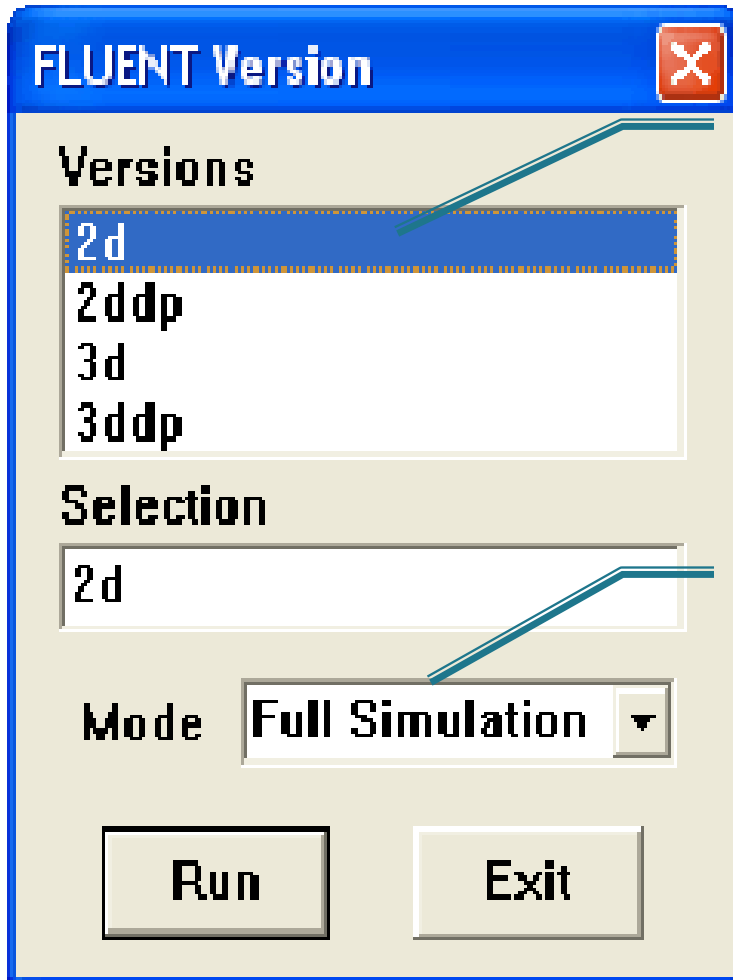
# Typical Flow Chart for Numerical Analysis



# Pre-processing in Fluent

Local Server: GUI-operations and their meaning

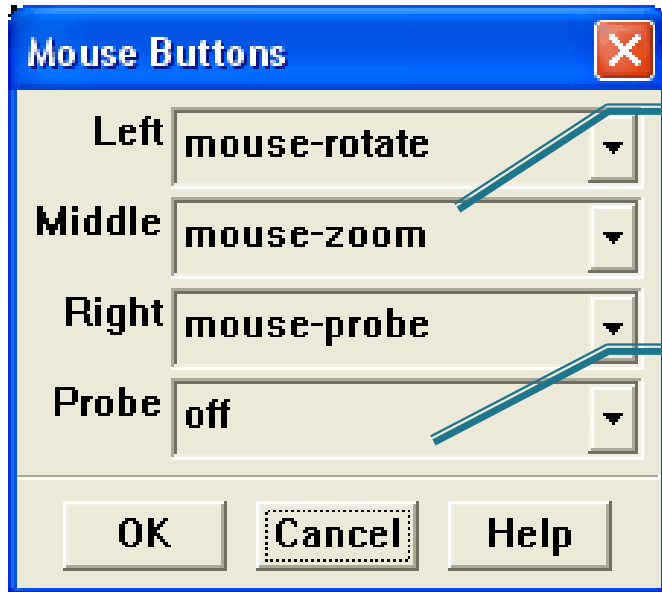




1. 2D or 3D, single precision or double precision (dp → double precision)

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

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



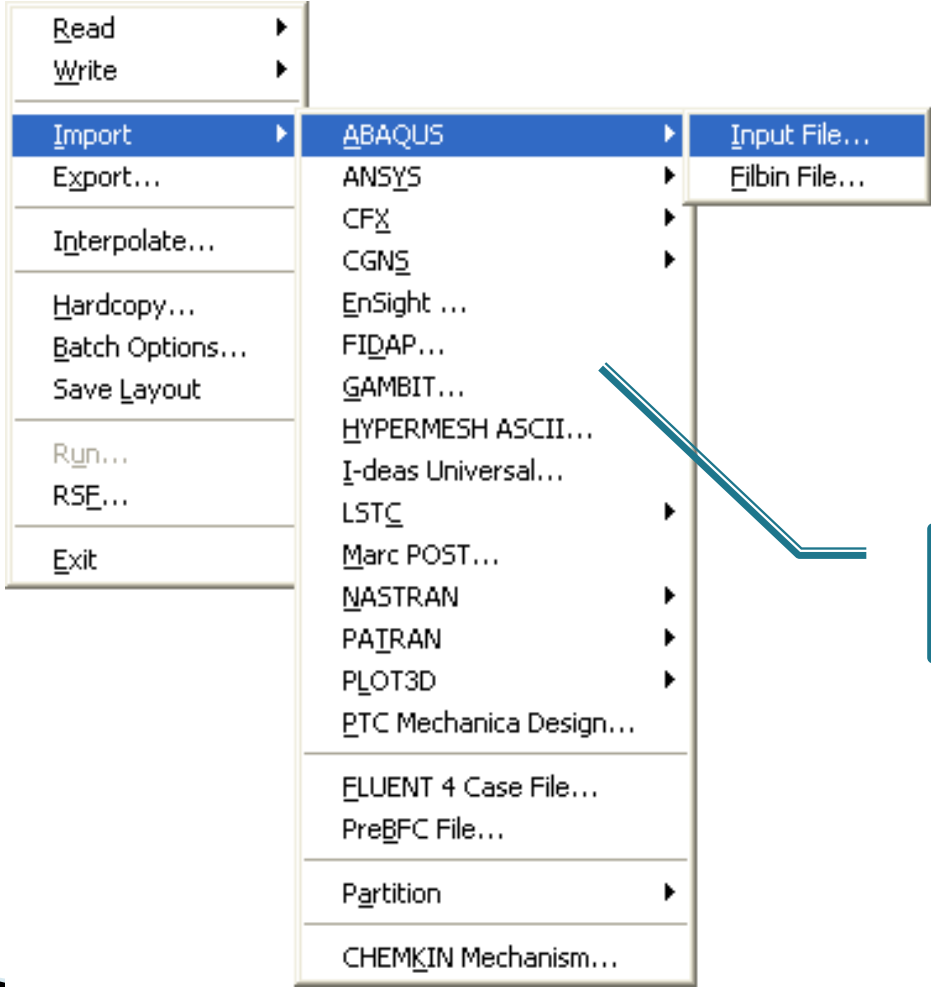
Basic Settings

The probe option with print the information when clicked inside mesh

# Reading Mesh: File → Import Mesh, Read Case ...

File

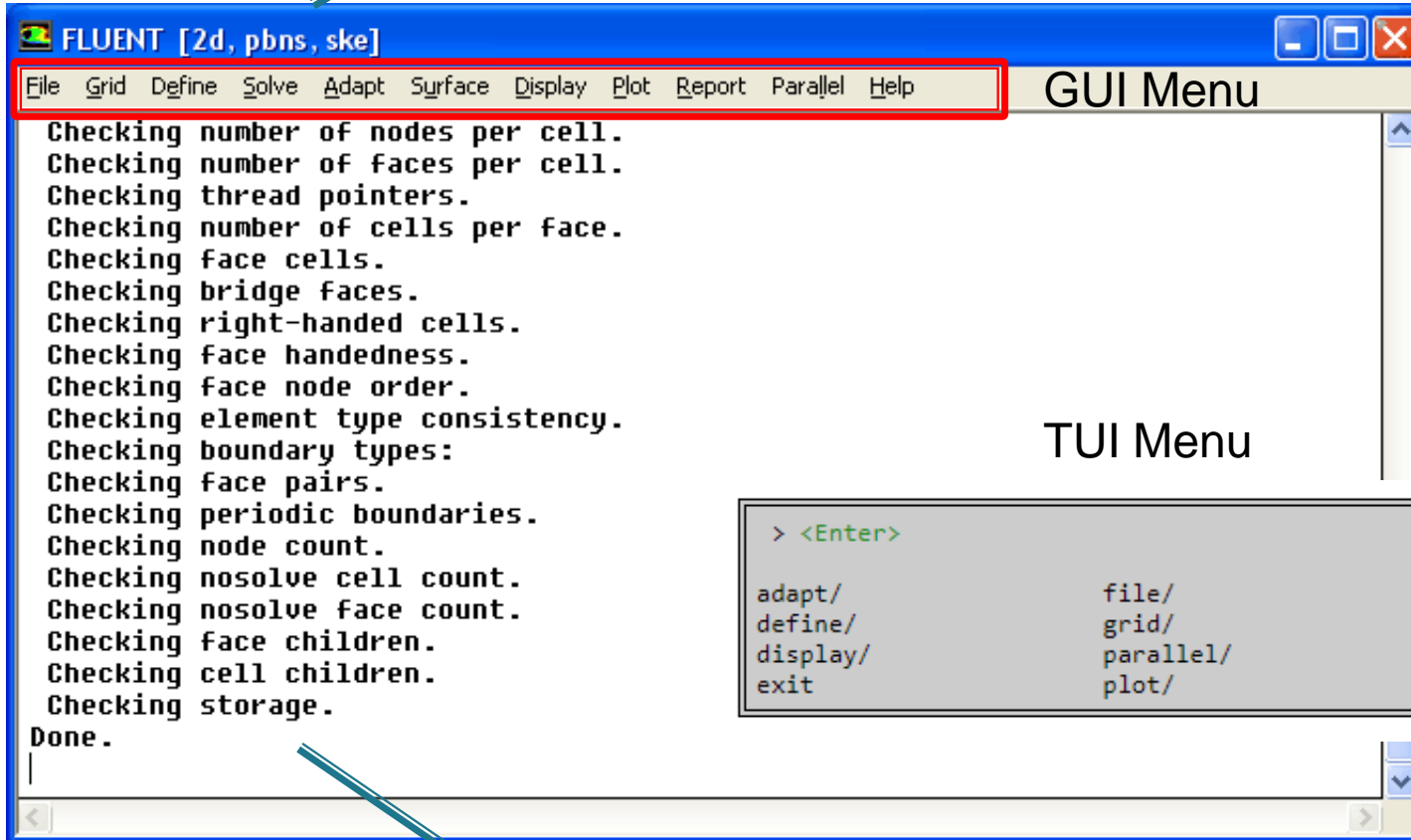
GUI Menu



Select as appropriate

# Reading Mesh: File → Import Mesh, Read Case ...

Top panel describes the summary of case file



```
FLUENT [2d, pbns, ske]
File  Grid  Define  Solve  Adapt  Surface  Display  Plot  Report  Parallel  Help
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 face node order.
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.
|
```

GUI Menu

TUI Menu

```
> <Enter>
adapt/      file/      report/
define/     grid/     solve/
display/    parallel/ surface/
exit        plot/     view/
```

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

# Check Mesh: Grid → Check

Check at the bottom for  
error message

## 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 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 or 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

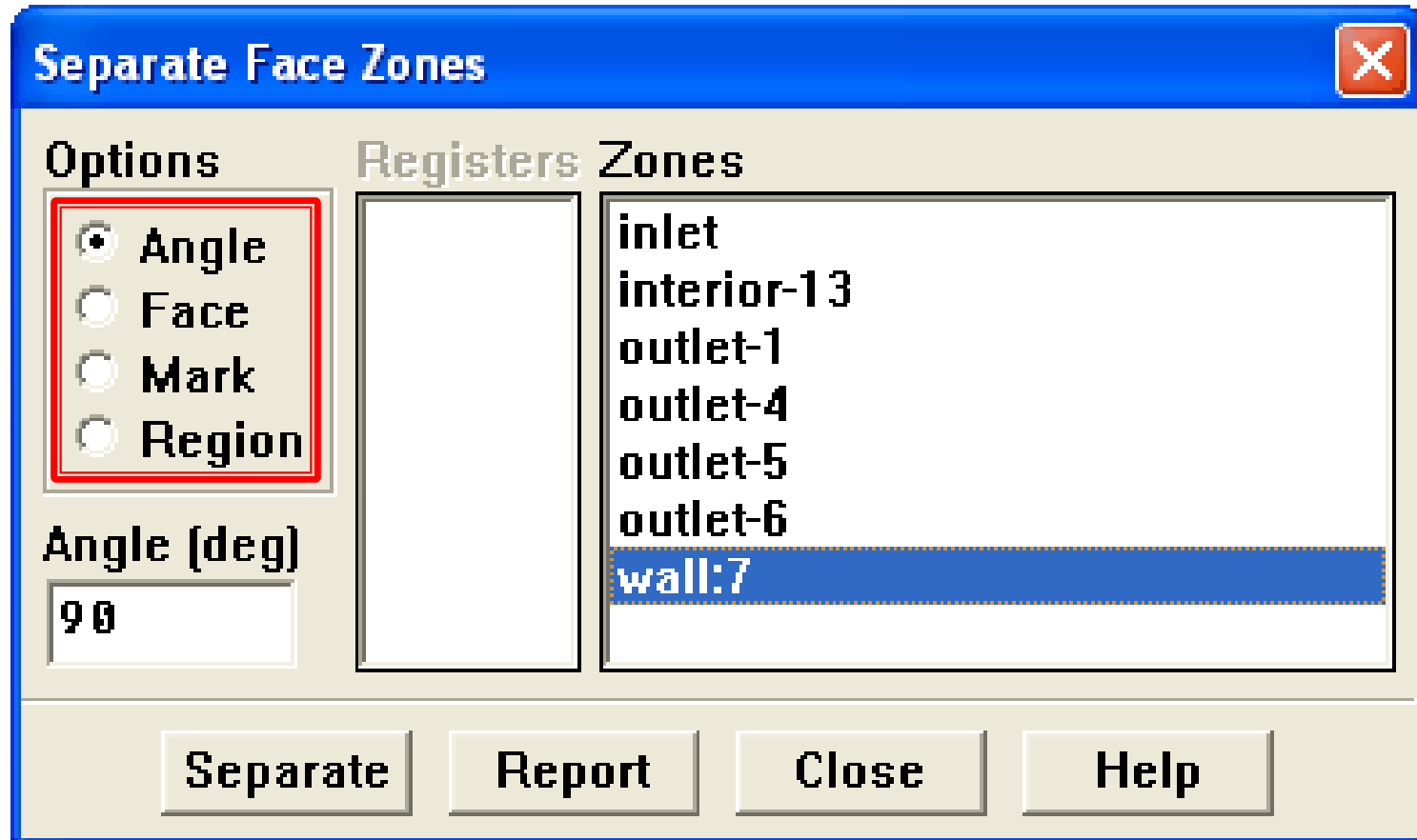
# Manipulate Mesh: Optional for ease of simulation

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

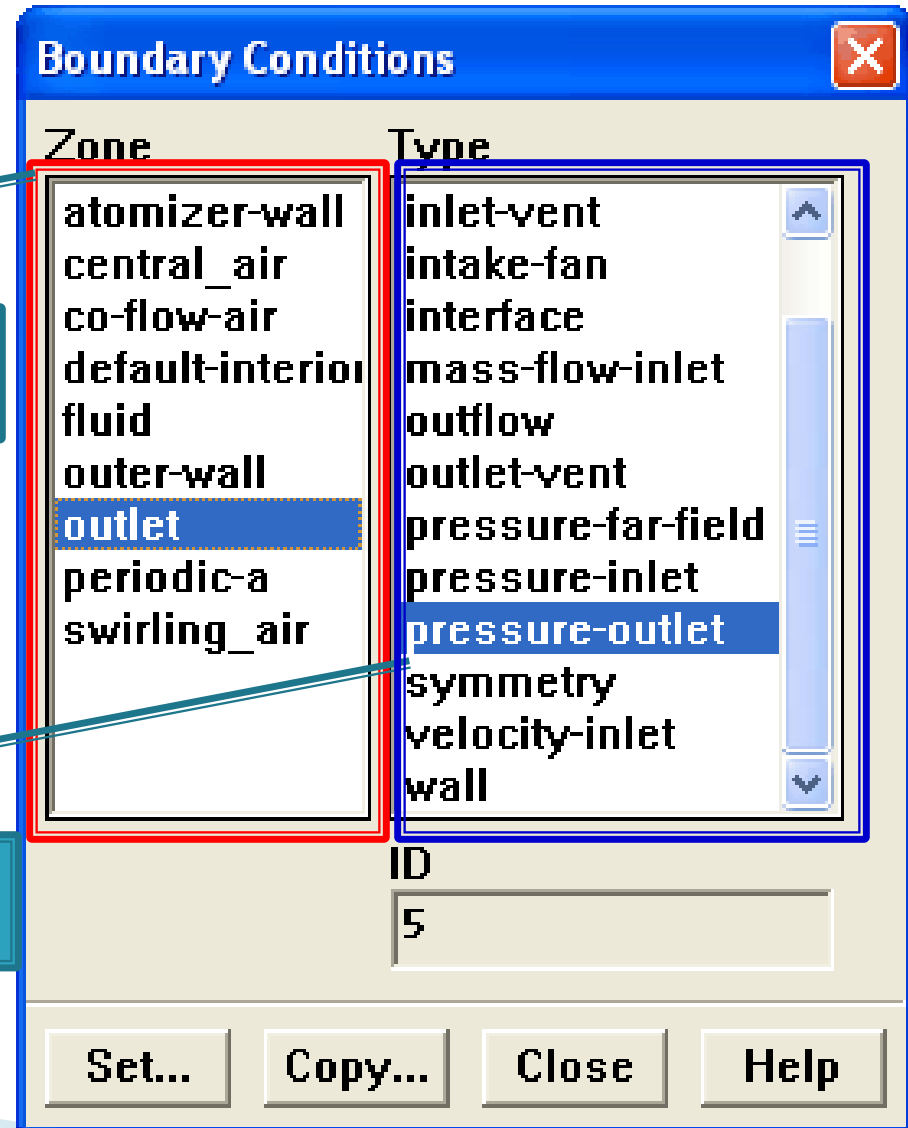


# Apply Boundary Conditions

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

The screenshot shows a 'Material' dialog box with a blue title bar and a close button. The main area is titled 'Properties of solids' and contains four rows of property settings. Each row has a label, a dropdown menu, an 'Edit...' button, and a text input field. A red box highlights the 'Density' and 'Cp' rows, and a blue box highlights the 'Thermal Conductivity' and 'Viscosity' rows. Callout boxes on the left point to these groups.

Property	Value
Density (kg/m <sup>3</sup> )	constant 2660
Cp (j/kg-k)	constant 737
Thermal Conductivity (w/m-k)	user-defined
Viscosity (kg/m-s)	constant 1.7894e-05

Buttons at the bottom: Change, Close, Help

Thermodynamics properties

Transport properties

# Define Turbulence Model

RANS: Reynolds-Averaged Navier Stokes and  $k-\epsilon$  are the workhorse of industry!

Turbulence Models

Wall Function

**Viscous Model**

**Model**

- Inviscid
- Laminar
- Spalart-Allmaras (1 eqn)
- k-epsilon (2 eqn)
- k-omega (2 eqn)
- Reynolds Stress (7 eqn)
- Detached Eddy Simulation
- Large Eddy Simulation (LES)

**k-epsilon Model**

- Standard
- RNG
- Realizable

**Near-Wall Treatment**

- Standard Wall Functions
- Non-Equilibrium Wall Functions
- Enhanced Wall Treatment
- User-Defined Wall Functions

**Options**

- Viscous Heating
- Full Buoyancy Effects

**Model Constants**

Cmu: 0.09

C1-Epsilon: 1.44

C2-Epsilon: 1.92

TKE Prandtl Number: 1

**User-Defined Functions**

Turbulent Viscosity: none

**Prandtl Numbers**

TKE Prandtl Number: none

TDR Prandtl Number: none

Energy Prandtl Number: none

**Law of the Wall**

none

OK Cancel Help



# Solver Settings

Solver running, monitor and convergence  
parameters

# Define Solution Limits

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

Limits of mean values

Parameter	Value
Minimum Absolute Pressure (pascal)	1
Maximum Absolute Pressure (pascal)	5e+10
Minimum Static Temperature (k)	1
Maximum Static Temperature (k)	5000
Minimum Turb. Kinetic Energy (m2/s2)	1e-14
Minimum Turb. Dissipation Rate (m2/s3)	1e-20
Maximum Turb. Viscosity Ratio	100000
Positivity Rate Limit	0.2

Limits of fluctuating values

# Define Initial Values: solve → Initialize → Initialize ...

A better guess helps improve the convergence sometimes!

**Solution Initialization**

Compute From: velocity-inlet-6

Reference Frame:  
 Relative to Cell Zone  
 Absolute

Initial Values

Gauge Pressure (pascal)	0
X Velocity (m/s)	0
Y Velocity (m/s)	1
Turbulent Kinetic Energy (m2/s2)	0.00375

Buttons: Init, Reset, Apply, Close, Help

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!

**Options**

- Print
- Plot

**Storage**

Iterations: 1000

**Plotting**

Window: 0

Iterations: 1000

Buttons: Axes..., Curves...

**Normalization**

Normalize  Scale

**Convergence Criterion**

absolute

Residual	Check	Absolute
continuity	<input checked="" type="checkbox"/>	0.001
x-velocity	<input checked="" type="checkbox"/>	0.001
y-velocity	<input checked="" type="checkbox"/>	0.001
z-velocity	<input checked="" type="checkbox"/>	0.001
energy	<input checked="" type="checkbox"/>	1e-06

Buttons: OK, Plot, Renorm, Cancel, Help

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

**Contours** Criterion-1

**Options**

- Filled
- Node Values
- Global Range
- Auto Range
- Clip to Range
- Draw Profiles
- Draw Grid

**Contours of**

Pressure...

Static Pressure

Min Max

0 0

**Surfaces**

- internal-3
- pressure-outlet-7
- velocity-inlet-5
- velocity-inlet-6
- wall-4

**Surface Name Pattern**

Match

**Surface Types**

- axis
- clip-surf
- exhaust-fan
- fan

Levels Setup

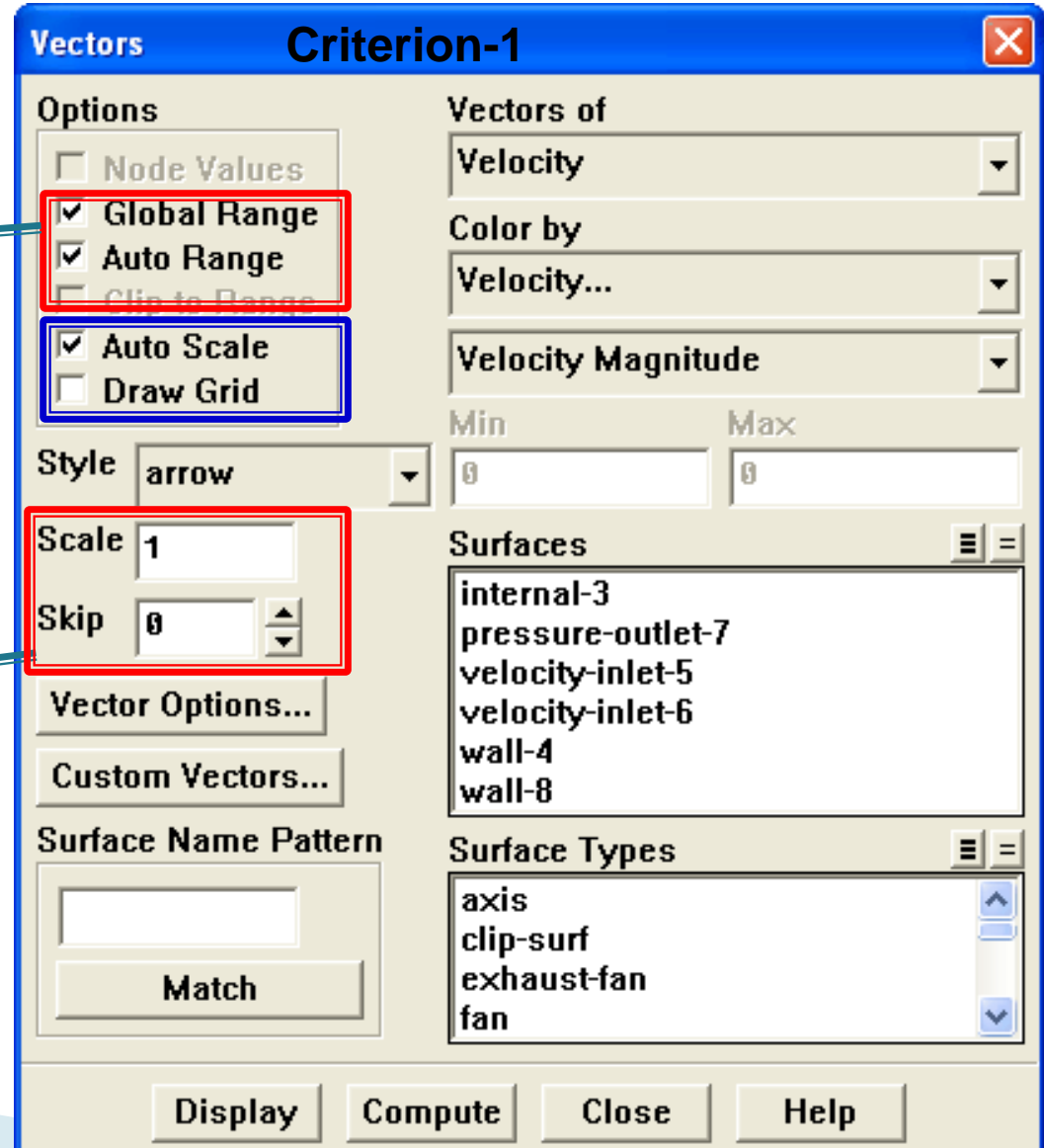
20 1

Display Compute Close Help



# 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](mailto:fb@cfdyna.com) to get help on any advance topic!