## STARTING WITH FLUENT-4

### Grid Generation (FLUENT-4)

Define → Allocate (Allocate the maximum number of grid, 15000 default).
Define → Domain (Specify dimension and grid size).
Define → Cell → Display (Display the computational grid) → Use left mouse to mark cells and specify inlet and outlet. (You can also add walls and/or additional inlet or outlets).

### Setup Boundary Conditions and Model (FLUENT-4)

Define → Boundary Conditions → Inlet velocity→ Set (Specify inlet velocity boundary condition).
Define → Model (You can modify the model and use turbulence models as needed).
Solve → Iterate (specify number of iterations)

### Displaying Results (FLUENT-4)

Solve → Monitor (To view convergence).
Display → Contours (To view pressure, velocity, etc contours plot. You have the choice of filled or unfilled contours). (Right mouse move the objects, middle mouse changes size).
Display → Vectors (To view velocity vector plots).
Plot → (To plot xy-plots).

### Save and Hard Copy (FLUENT-4)

File→Write (CAS and DAT files) (Saving the cas and dat files).
File→Read (CAS and DAT files) (Reading existing cas and dat files).
File→hardcopy →(Color, Tiff) save the picture on the graphic box in tiff format for hard copy printing.

### FLUENT-6

FLUENT-6 provides more flexibility as it can handle unstructured grid and body fitted coordinates. The geometry and the mesh have to be developed in GAMBIT code. It can then be imported into fluent.