

Introduction to Fluent and Simulation of Laminar Pipe Flow

By Michael A. Traetow

Purpose

The purpose of the pre-lab is to introduce students to the computational fluid dynamics (CFD) software Fluent. Laminar flow through a pipe will be simulated in order to achieve the friction factor and velocity profile. The data gained through the CFD simulation will then be compared to benchmark data.

Fluent



Fluent is the world leader in the rapidly growing field of design simulation software used to predict fluid flow, heat and mass transfer, chemical reaction and related phenomena. Fluent's software is used by engineers in corporations worldwide for product development, design optimization, troubleshooting and retrofitting.^[1]

With the use of Fluent and computational fluid dynamics (CFD) engineers are able to perform faster analysis at a reduced cost with the end result being an improved design. By solving the fundamental equations governing fluid flow processes, Fluent provides information on important flow characteristics such as pressure loss, flow distributions and mixing rates. CFD analysis complements traditional testing and experimentation, providing you with added insight and confidence in your designs. The benefits of using Fluent and CFD are better designs, lower risk and faster time to the market place for your product or process.^[1]

Fluent's CFD solvers provide a wide range of physical models and numerical techniques. From combustion to plastic extrusion, from supersonic airfoils to fluidized beds, Fluent provides the physics and numerics needed to get accurate answers and stable calculations.^[1]

Simulation Design

The simulation will be performed on the commercial CFD code Fluent. The simulation will be modeled after the laboratory set-up of experiment number two (see *Measurement of Flow Rate, Friction Factor, and Velocity Profile in Pipe Flow*) and therefore will use the same pipe geometry as well as the same flow parameters, such as fluid density and viscosity. By modeling the simulation after the experimental set-up, comparisons between the two test methods can be made.

In the remaining parts of this exercise and for those that will follow, **PLAIN BOLD** commands will symbolize commands entered using the mouse; **ITALIZED BOLD** will symbolize commands typed in at the Fluent command prompt.

Also worthy of note is the section titled "Additional Fluent Commands" attached to the end of pre-lab 1. These are commands that may or may not be needed for the pre-lab, however will be used in lab #1.

Introduction to CFD procedure

Each simulation using CFD, including Fluent, consists of five basic, but important steps. These steps will be described below. In each of the following steps, the user must specify the input parameters, which control the execution of the code and post processing of the results.

Step 1 Preliminary Inputs: During this step the user allocates memory for the CFD simulation that is going to be performed. At this point it is also helpful to gather the inputs needed for the rest of the simulation and prepare them to be entered into the CFD software.

Step 2 Grid Generation: This step is used to specify the geometry of the system, such as radius of a pipe that is to be modeled. It is also during this step that the user sets the boundary conditions, such as the inlet or outlet of a pipe. Figure 1-a is a component used in the aerospace industry, figure 1-b is the grid that was created to model this component for CFD analysis.

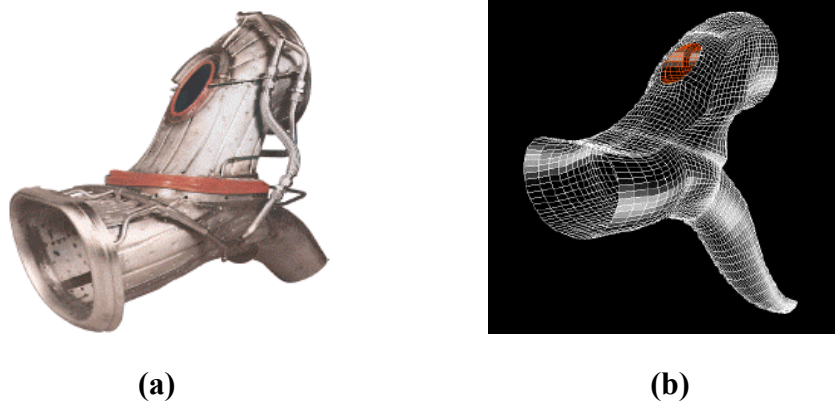


Figure 1: Computer modeling of an aerospace industry component
(Courtesy of Westland Aerospace Ltd.)^[1]

Step 3 Flow Parameters: The fluid characteristics, such as density and viscosity are very important to the CFD simulation. It is during this step that these two parameters, as well as the mass flow rate, will be set by the user.

Step 4 Solve: This step many times proves to be the easiest for the user. The user simply tells the program how many calculations to perform and activates the solver.

Step 5 Post Processing: The final step consists of the analysis of the results as well as interpretation. Fluent provides output in both visual and numerical form. Both are key in understanding the flow results.

In every CFD simulation whether simple or complex, these five basic steps are followed. It is of the utmost importance that care be taken while entering the input in each of these steps to ensure quality results. CFD, if used correctly, is as very useful and powerful tool.

CFD procedures for smooth laminar pipe flow

```

Fluent [V4]
File Define Solve Slices Display Plot Help

*****
*
*          FLUENT (V4.48) Fluid Flow Modeling
*
*          Copyright (C) 1984, 1989, 1991, 1995 by Fluent Inc.
*          All rights reserved. Use of this code is subject
*          to the terms of the software license agreement.
*          Use, reproduction, or disclosure by the U.S.
*          Government is subject to restrictions set forth
*          in Government Contracts with Fluent Inc. as
*          governed by FAR 52.227.19(c)(2).
*
*          FLUENT is a registered trademark of:
*
*          Fluent Inc.
*          Centerra Resource Park
*          10 Cavendish Court
*          Lebanon, New Hampshire 03766 USA
*          (800) 445-4454
*
*****
*          Number of Cells and Species Dynamically Allocated
*
*****

(*)- FILENAME ALIASED TO /disk4/apps/Fluent.Inc/fluent4.4/lib/HELPPFILE

Current fluent usage:
  1. muste@pca00.iuhr.uiowa.edu Wed Dec 31 18:00 /dev/ttyq2

License for fluent expires 9-apr-99.

COMMANDS AVAILABLE FROM *MAIN*:
ALLOCATE-MEMORY      READ-CASE-FILE      READ-CASE-DATA
EXPERT              FORMATTED-FILES    SETUP-1
OPTIONS             VIEW-GRAPHICS      QUIT
HELP
ENTER HELP (COMMAND) FOR MORE INFORMATION.
(*MAIN*)-

```

Figure 2: Fluent main menu

Initial Log-in:

- 1) At a UNIX machine, logon to the ICAEN account assigned to you for the class.
- 2) In the main directory, it may be helpful to create subdirectories for each of the CFD simulations. This is done by typing “mkdir <directory name>”.
- 3) In an xterm window type “fluent” to begin the Fluent software

Step 1: Preliminary Inputs

The inputs that are required for this simulation are the following:

- Fluid viscosity
- Fluid density
- Pipe diameter
- Reynolds number (1000)
- Mass flow rate (found by using 2-D mass flow rate equation, (1))

$$Q = \frac{Re \ v \ D \ \rho}{8} \quad (1)$$

Where Re is the Reynolds number, D is the pipe diameter, ρ is the density and ν is the viscosity. The derivation of this equation is available upon request.

- 1) Go to the **DEFINE** menu located in the command bar at the top of the Fluent window (fig. 2). Choose **ALLOCATE**.
- 2) From this menu, select **k-e/RNG Turbulence** model option in the “Create Space for” section. Select **OK**.

Step 2: Grid Generation

- 1) Go to the **DEFINE** menu in the command bar at the top of the Fluent window and select **DOMAIN**.
- 2) From this window, you will choose **Axisymmetric** from the “options box”.
- 3) Enter the **Length** (0.1 m) and the **Outer Radius** (radius of the pipe).
- 4) Enter values for the size of the grid in the **I-direction (7)** and the **J-direction (62)**.
- 5) **APPLY - CLOSE**.
- 6) **DEFINE CELLS**
- 7) Once in the “Set Cells” window, select **DISPLAY**. This will open a graphics window with a picture of the grid.
- 8) In this graphics window use the mouse to highlight the cells (I=1, J=61 to 2) and (I=7, J=61 to 2). Once these are highlighted, in the “Set Cells” window click on the arrow in the “Zone-type” section and select **CYCLIC – APPLY**.
- 9) A window will appear that will ask if the geometry is rotationally cyclic, it is not so choose no.
- 10) In the graphics window highlight the cells (I=1 to 7, J=1) and define them as **AXIS** in the “Set Cells” window. Remember to hit **APPLY**
- 11) The cells (I=1 to 7, J=62) will be defined as wall type boundary conditions, but since the wall type boundary condition is set as the default in Fluent, you will not have to change these. **CLOSE**.
- 10) You may view the grid by choosing **DISPLAY GRID LIVE - DISPLAY**.

Step 3: Flow Parameters

Reminder: *italicized* commands signify that the user must type these commands at the prompt

- 1) From the *setup-1* command prompt go to the *Physical Constants* menu.
- 2) To input the density of the fluid, choose *Density* from this menu. You will be asked if you would like to employ the ideal gas law, you do not so answer no.
- 3) Type *Viscosity* to set the desired viscosity. Please note the units of the viscosity requested.
- 4) Return to the main menu by typing *Quit* at the prompt until you have reached the “main menu” command prompt.

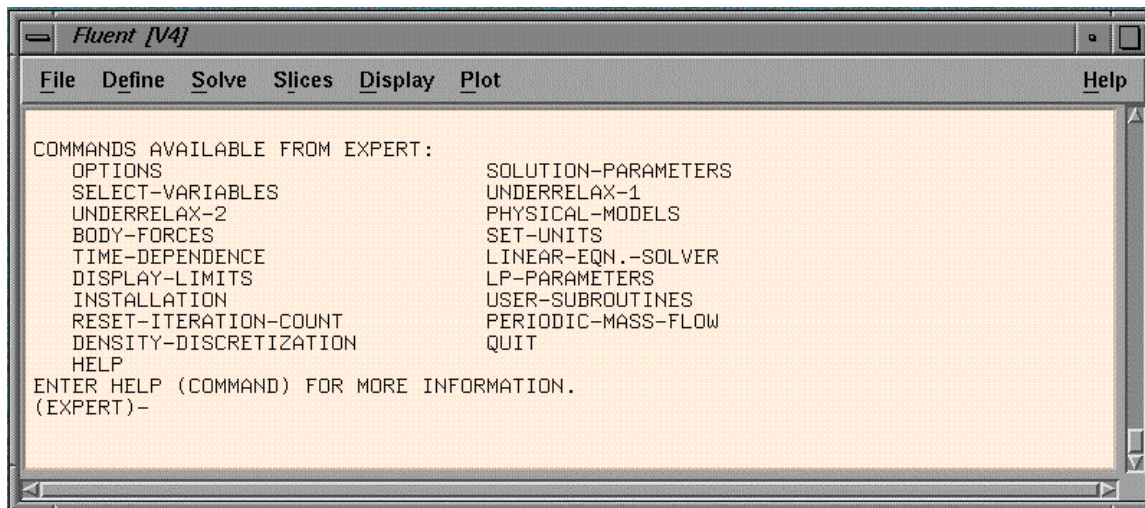


Figure 3: Expert menu

- 5) You will be asked once again if the geometry is rotationally cyclic, as before you will answer no.
- 6) *Expert* (figure 3) - *Periodic Mass Flow*.
- 7) The first line asks if you would like to set the periodic mass flow rate, type yes. On the following line the mass flow rate is requested. Here you will enter the Two dimensional mass flow rate calculated using the equation in the assignment handout.
- 8) On line 3, enter a value of one as the “Guess for Pressure Drop Parameter”.
- 9) Set the “Number of Correction Sweeps” at two on the following line.
- 10) Return to the expert menu by typing *Done* on the last line.
- 11) *Solution Parameters*. Change “Min Residual Sum” to $1 * 10^{-5}$. *Done Quit*.
- 12) *Patch*. A series of question about the regions you would like affected by this command will appear. For each of these questions (there will be a total of four) the default value set by Fluent is the value you will use so press enter.
- 13) *U-velocity*. You will want to set an initial guess for the velocity of the fluid in the axial direction. The value you will enter here will be calculated using the Reynolds number, however this is a guess so any number will do.
- 14) Return to the main menu.

Step 4: Solve

- 1) **SOVLE CONTROLS UNDERRELAXATION**
- 2) Change the underrelaxation factor for the velocities to 0.7. The rest of the default values will be sufficient for your calculations.
- 3) **APPLY CLOSE.**
- 4) **SOVLE – MONITERS RESIDUALS**
- 5) From the “Residuals” window, select **Plot** from the options section. A plot of the residuals is shown in.
- 6) **APPLY - CLOSE**
- 7) **SOLVE ITERATE**
- 8) In the “number of iterations” window enter the desired number of iterations. It is recommended that at least 600 iterations be performed, or until convergence occurs. Click on the **Iterate** button to begin the solver.

Step 5: Post Processing

- 1) **DISPLAY PROFILES**
- 2) The Profiles window allows you to view profiles of various data acquired by the solver. This window also gives the minimum and maximum values of the variable selected in the “Profiles of” section.
- 3) To change the variable, click on the arrow in the “Profiles of “ section and select the desired variable.
- 4) Once selected, click on **Display** to view graphically the profile of the selected variable. The velocity profile for fully developed turbulent pipe flow is shown in.
- 5) From the main menu, select **View Alpha Select Variable**. This will bring up a menu titled “variable options”.
- 6) From this menu choose **Xtended Xoptions**.
- 7) You are able to determine the wall shear stress by entering **X-dir-wall-shear-stress**. The desired value for wall shear stress will be the only non-zero value given on this screen. From the wall shear stress you are able to calculate the friction factor.

Parametric Studies

The accuracy and, therefore, the quality of CFD results depend upon the magnitude of the numerical (i.e., those due to discretization and solution of the algebraic equations) and modeling (e.g., those due to turbulence modeling) errors. The numerical errors can be determined through grid studies and diagnosis of the iterative convergence history. This process is known as CFD verification. The modeling errors, on the other hand, can be determined through comparison with benchmark experimental fluid dynamics (EFD) data. This process is known as CFD validation.

Because the numerical error is dependent upon the grid resolution, it is important that the grid contain a suitable number of points, especially in regions of high gradients. For pipe flow, the gradients are largest near the pipe wall and rough estimates of the required resolution can be related to the turbulent-flow length scale,

$$y^+ = \frac{U_t y}{\nu} = \frac{\sqrt{t_w/r} y}{\nu}$$

Typically, the first grid point should be located at $y^+ \approx 1$. These calculations have already been performed for the turbulent flow through a pipe and have been incorporated into the grid design parameters which will be provided in class.

In order to understand the effects that the number of grid points as well as grid spacing have on the simulation; rerun the laminar case using a different number of points in the radial direction. It is preferable to perform one simulation with a greater number of points equal to two times that of the current value, and one simulation with one half the number of points than that of the original simulation. The technique of increasing and decreasing by a factor of two for each grid is called grid doubling. One generally starts with a coarse grid and doubles the number of points for each following grid. By using this grid doubling technique one is able to assess the uncertainty of the simulation.

Analysis and Discussion

Table 1: Data Comparison for Fully Developed Laminar Flow

Case	Friction Factor	E=S-D	ϵ	Center Line Velocity (m/s)
124	0.0644	0.0004	-	0.609
62	0.0642	0.0002	-0.002	0.610
31	0.0643	0.0003	0.001	0.609
16	0.0644	0.0004	0.001	0.607
8	0.0658	0.0018	0.0014	0.594
theory	0.064	-	-	0.611

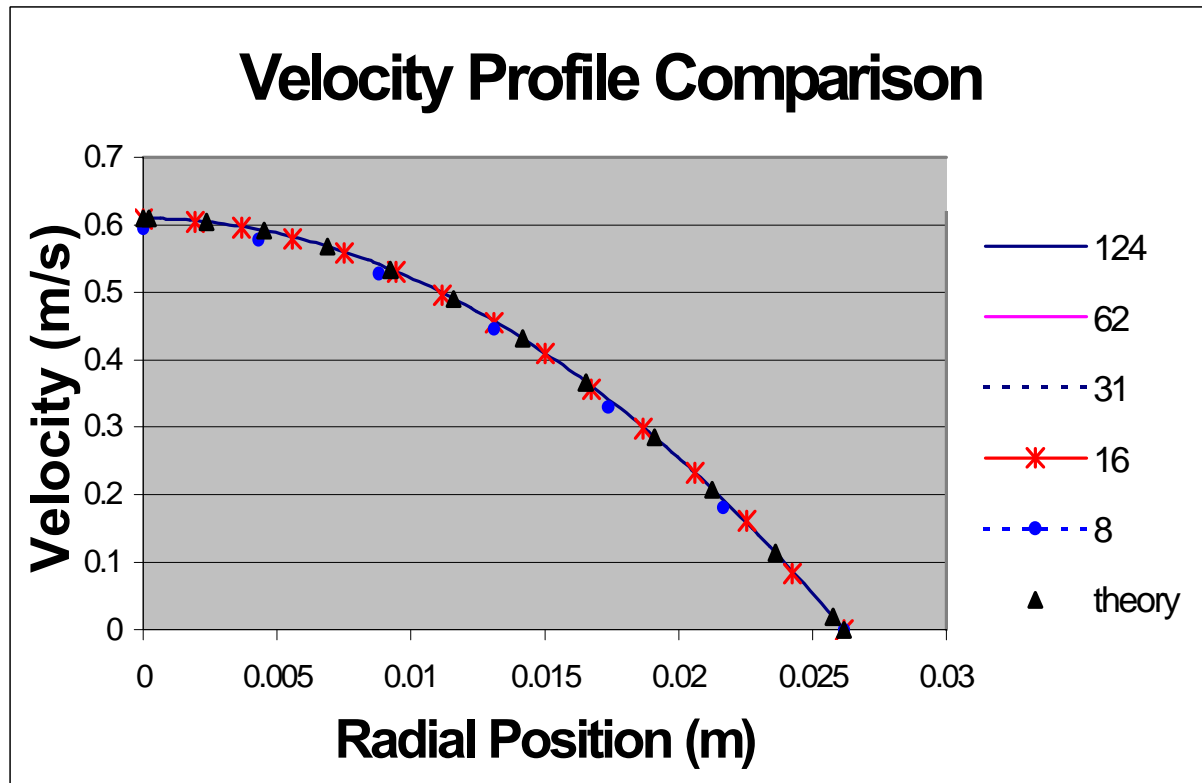


Figure 4: Laminar velocity profile comparison

Additional Fluent Commands

The following section provides additional commands that may be helpful in per-lab as well as for lab 1.

Saving a Fluent File

- **FILE WRITE CASE & DATA**
- Enter the <file name>.cas and click on **OK**.

Saving a Graph/Plot in Fluent

- **FILE HARDCOPY**
- If you will be printing this graph make sure that the **postscript** option is selected.
- Click on **Save**.
- Enter the <file name>.ps.
- This can be accessed and printed while in the directory in which it was saved.

Saving numerical data in Fluent

This option will be used when making graphs outside of Fluent (in Excel).

- **PLOT XY PLOT**
- In the “Options” section choose **write to file**.
- Select the data that you would like to have saved in a data file from the “Y Axis Function” section.
- Click on **WRITE**.
- Enter the <file name>.xls.
- **OK**.

Defining turbulence models and near wall spacing

This step can be performed anytime during the grid generation after defining the pipe length and outer radius. This command determines which turbulence model will be used to solve the problem and how the selected solver will treat the near wall points.

- **DEFINE-MODELS** (figure 5)
- To select the view the turbulence models available to the user, click on “Turbulence Model”. For the CFD labs, the **K-Epsilon** model will be used.
- One can view the different near wall treatments by clicking on the respective are. The **Two-layer Zonal Model** will be implemented for lab #1.

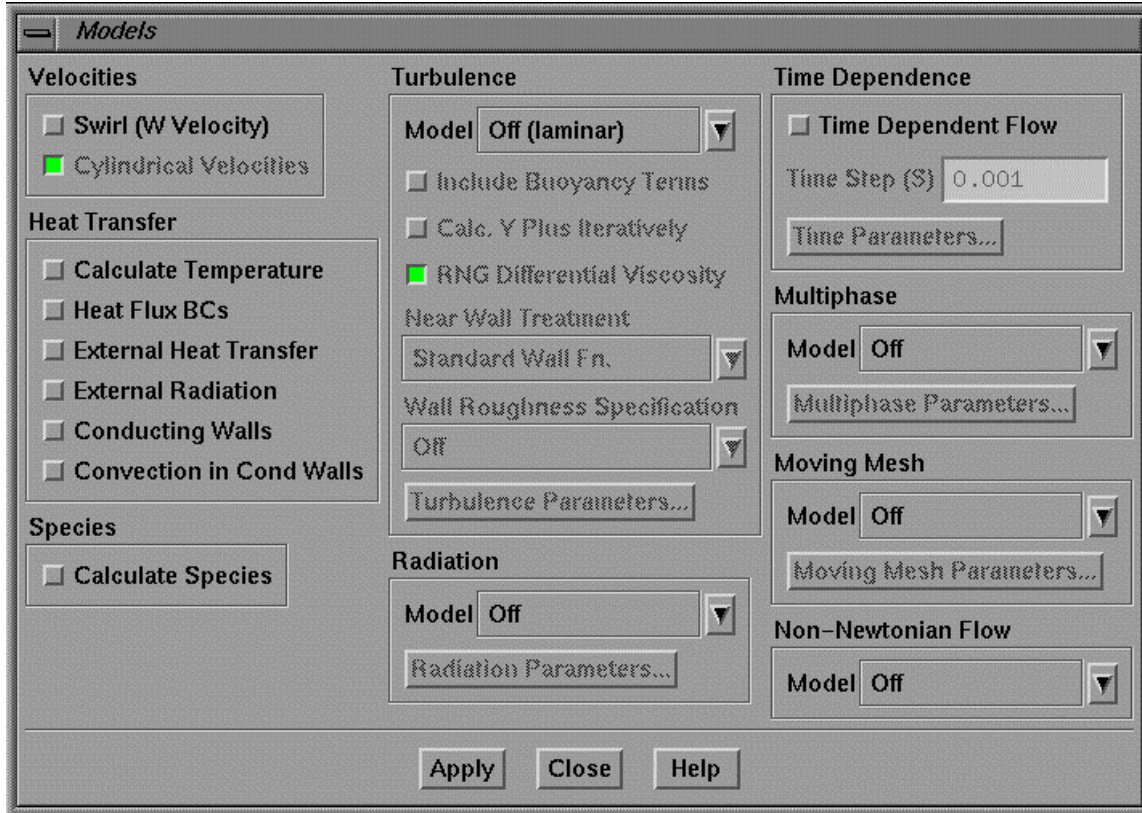


Figure 5: Models menu

Creating a non-uniform grid in Fluent

Many times it is necessary to have a very fine point distribution in one section of the grid in order to improve accuracy of the calculations, as will be the case in lab one. You will be required to create a non-uniform grid such that the near wall spacing is very small in order to acquire quality data from the simulation. This step is performed after all of the boundary conditions are applied.

- From the Fluent command prompt enter *Setup1* (figure 6).
- You will need access to the grid generation tools, so enter *generate-grid*.
- The user is able to perform tasks both in the radial and axial direction. However, since the point of interest for lab 1 will be in the radial enter *Radial(2)*.
- The intent of this task is to make changes to the existing uniform grid, so you will now enter *Modify Segments*.

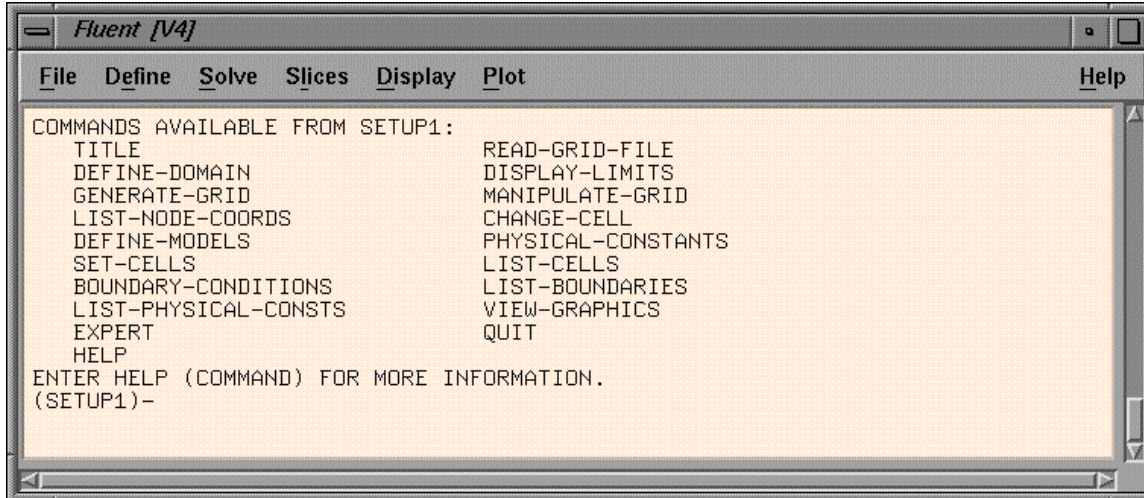


Figure 6: Setup-1 menu

Once you have entered into this menu, you are able to make various modifications to the existing grid. However the only modification you will be required to make is to alter the “End Point Weighting Factor”.

- Move the cursor to the “End Point Waiting Factor” line by pressing return
- Once on the correct line enter a new value. A value of seventy will be used for lab 1.
- If all input is correct, type **Done** on the last line.

If you enter an incorrect value on any of the lines in this screen, at the last line type **Refresh** or **Top**, both of which will bring the cursor back to the first line.

Setting simulation to model turbulence

For the case of turbulent flow, additional inputs must be set in order to accurately model the real life conditions. For example the level of turbulence, which entered into Fluent in terms of energy, must be specified. The following will aid you in setting the simulation to model turbulent flow.

- In the same menu where the initial guess for the U-velocity is set, (**Expert-Periodic Mass Flow**) an option is available to set the dissipation and energy values for the system.
- In order to quantify the level of turbulence in the system values for the dissipation and the energy must be entered. Since both of these values will negligible in the pipe flow simulation, they will be set to **Dissipation = $0.001 \text{ m}^2/\text{s}^3$** and **Energy (Turb) = $0.01 \text{ m}^2/\text{s}^2$** .

Fluent has many options available to the user. The best way to become familiar with Fluent, like any program, is to experiment with its capabilities. If you ever have any questions, please do not hesitate to ask.

References

- [1] Fluent Home Page, <http://fluent.com>