Chapter 3 NCSA UIFlow Menus

About the NCSA UIFlow Menus

This section reviews each of the menus and commands that appear in NCSA UIFlow's menu bar. These commands appear in six menus: Apple, File, Edit, Simulation, Options, and Grid, as shown in Figure 3.1. The commands in these menus are dimmed when not applicable; NCSA UIFlow will not activate a dimmed command.

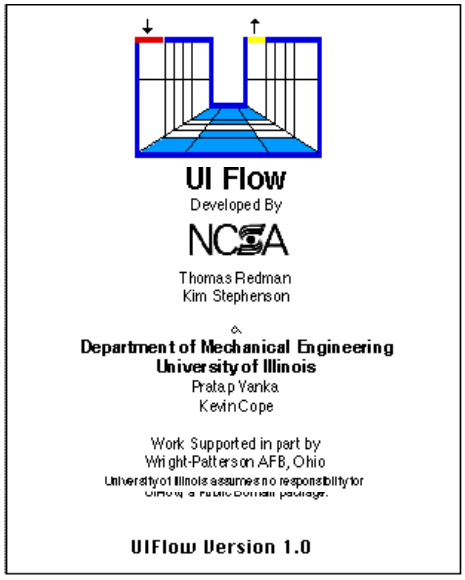
Figure 3.1 **NCSA UIFlow Menu Bar**

The NCSA UIFlow menus function as any other Macintosh menu. For additional information about using menus, refer to your Macintosh user's guide.

Apple Menu

The Apple menu appears in all Macintosh applications. It permits access to the Macintosh desk accessories, such as the Chooser, Calculator, and Control Panel. In addition, the Apple menu contains the command *About UIFlow*. Upon choosing About UIFlow from the Apple menu, the About Box, shown in Figure 3.2, appears. To remove the About Box and return to the application, press any key. For more information regarding desk accessories, refer to your Macintosh user's guide.

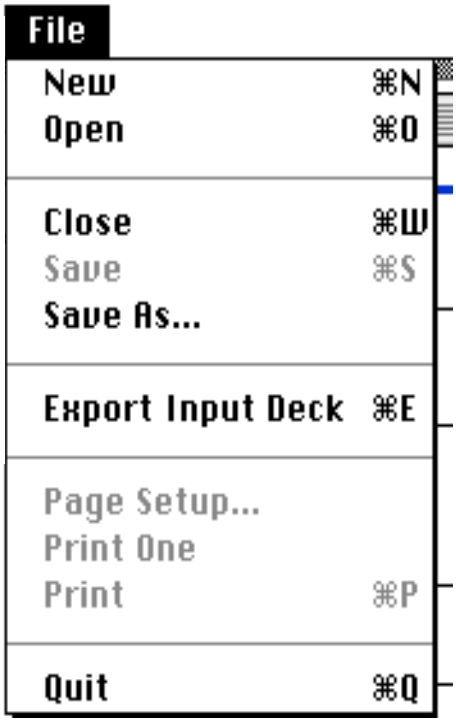
Figure 3.2 **About Box**


UI Flow
Developed By
NCSA
Thomas Redman
Kim Stephenson
Department of Mechanical Engineering
University of Illinois
Pratap Vanka
Kevin Cope
Work Supported in part by
Wright-Patterson AFB, Ohio
University of Illinois assumes no responsibility for
UIFlow Version 1.0

File Menu

The File menu, shown in Figure 3.3, contains the commands New, Open, Close, Save, Save As, Export Input Deck, Page Setup, Print One, Print and Quit.

Figure 3.3 **File Menu**



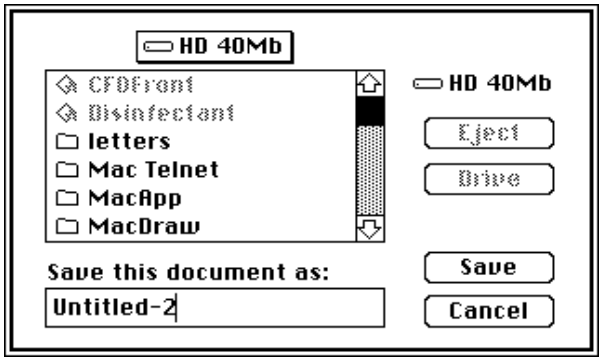
New
Creates a new dataset and displays it in the Geometry window.

Open
Opens an existing dataset stored on disk and displays it in the Geometry window.

Close
Closes the dataset. If you have made changes to the dataset since the last Save, a dialog box prompts you to Save the dataset before closing.

Save
Saves the dataset associated with the foremost window. When saving the dataset for the first time a dialog appears prompting the user for a file name (Figure 3.4.)

Figure 3.4 **Save Dialog Box**



Save As...
Saves the dataset associated with the front most window. Using this command always causes the Save dialog box (Figure 3.4) to appear.

Export Input Deck
Creates two files in the current directory named *uiflow.in* and *uiflow.grid*. These files save all the information on the geometry in the front most window in a format compatible with the computational code.

Page Setup...
Dimmed. Not supported in this version.

Print One
Dimmed. Not supported in this version.

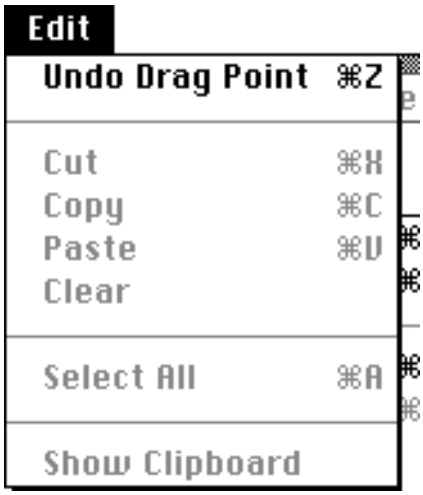
Print
Dimmed. Not supported in this version.

Quit
Exits the NCSA UIFlow application. If a dataset is open and you have made changes to it, NCSA UIFlow asks you if you wish to save the dataset before exiting the application.

Edit Menu

The Edit menu, shown in Figure 3.5, contains the following commands: Undo, Cut, Copy, Paste, Clear, Select All, Show Clipboard.

Figure 3.5 **Edit Menu**



Undo
Undoes the specified operation.

Cut
Dimmed. Not supported in this version.

Copy
Dimmed. Not supported in this version.

Paste
Dimmed. Not supported in this version.

Clear
Dimmed. Not supported in this version.

Select All
Dimmed. Not supported in this version.

Show Clipboard
Dimmed. Not supported in this version.

Simulation Menu

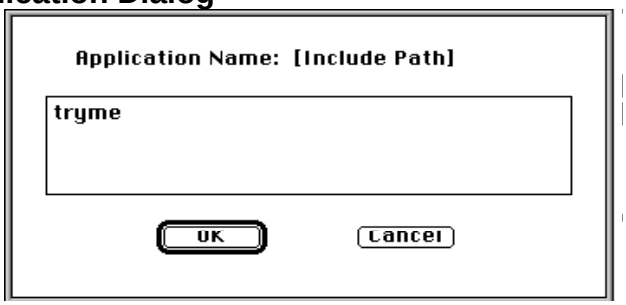
The Simulation menu, shown in Figure 3.6, contains the commands Set Remote Application, Remote and Local.

Figure 3.6 **Simulation Menu**

Set Remote Application

Stores the name of the program to execute on a remote machine. Included here should be everything required to access the program, (ie. the machine name, directory and program name.)

Figure 3.7 **Set Remote Application Dialog**



Remote

Runs the simulation by connecting to a remote machine with DTM [Data Transport Mechanism] and executing the program specified in Set Remote Application.

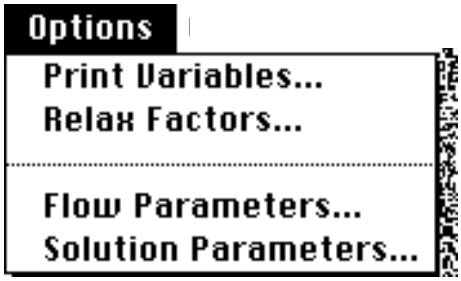
Local

Runs the simulation using UIFlow2D on the Macintosh. This command does not use DTM and does not connect to a remote machine.

Options Menu

The Options menu, shown in Figure 3.8, contains the following options: Print Variables, Relax Factors, Flow Parameters and Solution Parameters.

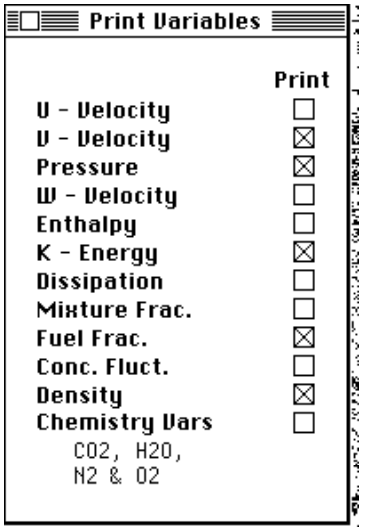
Figure 3.8 **Options Menu**



Print Variables...

Selecting the Print Variables command causes a dialog box to appear (Figure 3.9). This dialog is used to indicate for which variables UIFlow2D should create images. Selected variables are indicated with an X in the check box. The UIFlow2D program creates images only for the variables selected by the user. Exit the Print Variables window by clicking its close box.

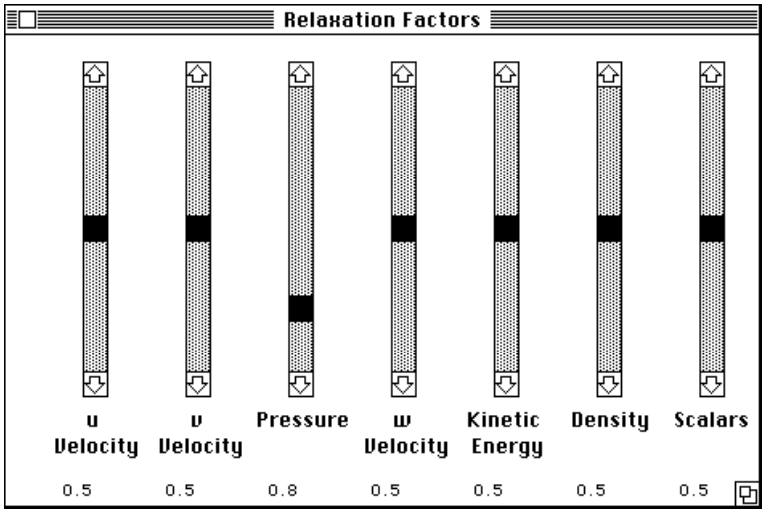
Figure 3.9 **Print Variables Menu**



Relax Factors...

Selecting the Relax Factors command brings up the Relaxation Factors dialog box (Figure 3.10.) Each variable has a range from zero to one, available in increments of 0.1. To edit a value click and drag the slider up or down. The selected value is displayed at the bottom of the bar. Exit the Relaxation Factors window by clicking its close box.

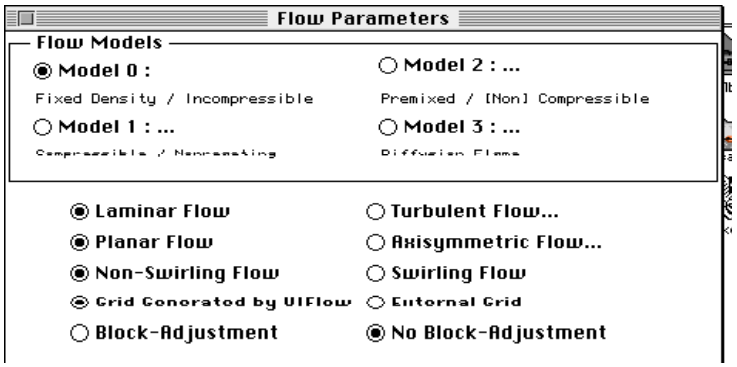
Figure 3.10 **Relaxation Factors Selection Box**



Flow Parameters...

Selecting the Flow Parameters command brings up the Flow Parameters dialog box (Figure 3.11.) This box is divided into two sections. The first contains four possible model types. Only one model can be selected at a time. In the second section there are five pairs of options.

Figure 3.11 **Flow Parameters Selection Box**

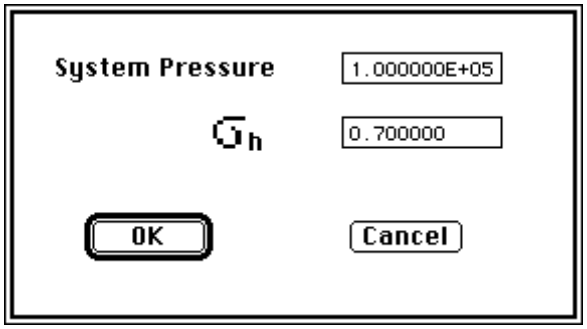


Models 1, 2, and 3 require additional information. When these are selected a sub-dialog box appears. Additional information for each data type can be found in the technical section of this document.

Model 0 is a fixed density, incompressible flow which requires no additional data.

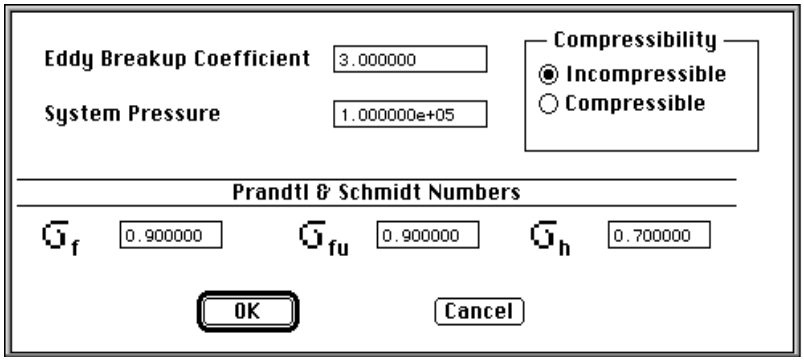
Model 1 is a nonreacting, compressible air flow. Choosing Model 1 brings up a dialog box (Figure 3.12) which prompts the user to set the system pressure and σ_h.

Figure 3.12 **Model 1 Dialog Box**



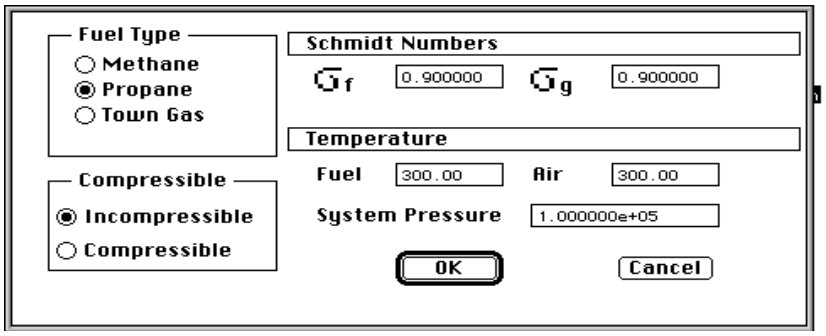
Model 2 refers to a low mach number premixed flame simulation. It requires information about the eddy breakup coefficient, system pressure, compressibility and prandtl numbers σ_f, σ_{fu}, and σ_h.

Figure 3.13 **Model 2 Dialog Box**



Model 3 sets up a diffusion flame. It requires information about the fuel type, compressibility, temperature, system pressure and Schmidt numbers σ_f and σ_g.

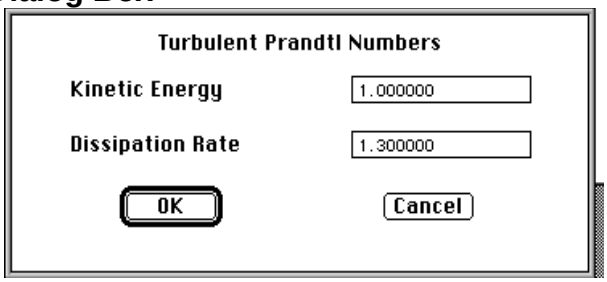
Figure 3.14 **Model 3 Dialog Box**



The Flow Parameters dialog has five pairs of additional options. They are Laminar Flow / Turbulent Flow, Planar Flow / Axisymmetric Flow, Non-Swirling / Swirling Flow, Grid Generated by UIFlow / External Grid, Block-Adjustment / No Block-Adjustment. To set an option click on the button next to the data type. This will deselect its partner. Two of these, Turbulent Flow and Axisymmetric Flow, have sub-dialogs to request additional data.

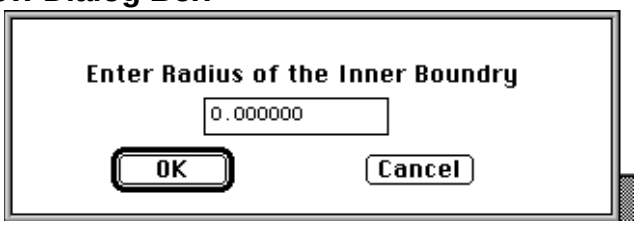
The Turbulent Flow dialog box (Figure 3.15) requires information about kinetic energy and dissipation rate.

Figure 3.15 **Turbulent Flow Dialog Box**



The Axisymmetric Flow dialog box (Figure 3.16) requires information about the radius of the inner boundary.

Figure 3.16 **Axisymmetric Flow Dialog Box**



Solution Parameters...

Selecting the Solution Parameters command brings up the Solution Parameters dialog box (Figure 3.17). The Solution Parameters dialog is used to enter six pieces of information; Number of Fine Grid Levels, Solution Accuracy, Maximum Iterations, Number of Sweeps on Momentum Equations, Number of Sweeps on Pressure Corrections, and Number of Sweeps on Scalar Equations. These are used by the computational program to determine how accurate the solution needs to be.

To edit the Solution Accuracy and Maximum Iterations use the mouse to highlight the value currently in the box and type the new value. The new value will replace the old value. To edit the Number of Fine Grid Levels, Number of Sweeps on Momentum Equations, Number of Sweeps on Pressure Corrections, or Number of Sweeps on Scalar Equations click the mouse on the slider and drag it left (for a smaller value) or right (for a larger value.) The range for Number of Fine Grid Levels is one to five, Number of Sweeps on Momentum Equations is one to ten, Number of Sweeps on Pressure Corrections is one to twenty, and Number of Sweeps on Scalar Equations is one to ten.

Figure 3.17 **Solution Parameters Selection Box**