

4.0 USER'S GUIDE

In the previous chapter, we learned that GFSSP's interactive preprocessor generates the model input data file that is read by the solver module. The purpose of this chapter is to provide detailed instructions on how to create a data file, for any given flow circuit, with the help of the GFSSP interactive preprocessor. This chapter also includes a detailed description of the contents of the model input data file.

GFSSP includes two different interactive preprocessors, one is referred to as the Command Line Preprocessor and the second is referred to as the Visual Preprocessor. The Command Line Preprocessor runs in the DOS, Unix and Macintosh environments and the Visual Preprocessor runs only under Windows NT™ or Windows 95™. The Command Line Preprocessor prompts the user to input the information required by the program to build the desired model through a series of sequential on screen questions or statements. The Visual Preprocessor provides a graphical user interface type of method for inputting the required model data through a series of menus and sub-menus. The following sections provide details for using both of these interactive preprocessors.

4.1 DOS COMMAND LINE PREPROCESSOR

In order to run the code on a PC, the user must type the name of the GFSSP executable file, GFSSP2PC, at the DOS prompt. If the user has loaded the program and the associated files on the root directory of their C'drive, the command would appear as follows:

```
C:\GFSSP2PC
```

The first question the code will ask is:

```
'DO YOU WANT TO READ A DATA FILE? '
```

If the user answers 'yes' to this question, the code will prompt the user to supply an existing input data file name. After a successful reading of the input data file, GFSSP will proceed to calculate a solution for the specified model. If the user answers 'no' to the question, they are indicating to the program that they want to create a new model and the preprocessor subroutine will be invoked and the interactive session will begin.

The GFSSP preprocessor prompts the user for all of the necessary information to create the model input data file. After prompting the user for all of the data required to define the model, the code writes this data into a file specified by the user at the end of the interactive session. As part of the model definition process, the user will provide a file name to be used by the program for the output file and the analyst's name which is also recorded in the input and output files. Before building the desired model, the preprocessor will prompt the user to input a model title of

less than or equal to 80 characters. After the preceding information has been input, the preprocessor will proceed to construct the model.

The sequence of inputs defining the model in the preprocessor are as follows:

1. Selection of model options
2. Node information
3. Branch information
4. Boundary conditions
5. Additional information

These steps will be discussed in detail in the following sections.

4.1.1 Selection of Model Options

GFSSP offers the capability to model various physical situations. These capabilities are made available through model logical options which are set by the model builder to either true or false. The user can select one or more of these options during the development of a flow model. A complete listing of the available logical variables and their significance appear in Table 4.1.

During a model building session, the preprocessor will ask the user to select between various modeling options available in the code. The user can select the option by typing either an upper or lower case y to activate the current option or n to leave the option deselected. The code sets the logical variables either to TRUE or FALSE depending upon the user's answer to the preprocessor's prompt.

The interactive session is sequential. This implies that the preprocessor will prompt the user to supply information based on the choices made previously during this session. The following sections provide additional details for each of the available logical options.

Table 4.1.1 - GFSSP Logical Variables

Variable	Meaning
DENCON	= T; The program utilizes a user supplied constant density and viscosity.
	= F; All thermodynamic and thermophysical properties are computed.
GRAVITY	= T; Gravitational force effects will be calculated for branches utilizing Resistance Option 1 or 4.
	= F; Gravitational force effects are not calculated.
ENERGY	= T; The energy equation is solved (for DENCON = F and/or with heat sources).
	= F; The energy equation is not solved.
MIXTURE	= T; Used when there is more than one fluid in the circuit.
	= F; Used when there is single fluid in the circuit.

Table 4.1.1 (Continued) - GFSSP Logical Variables

Variable	Meaning
THRUST	= T; Axial thrust is calculated by the program from node pressures and areas.
	= F; Axial thrust is not calculated by the program.
STEADY	= T; Calculates a steady state solution for the given flow circuit.
	= F; Performs quasi-steady and unsteady calculations for the flow circuit with a given time history for the boundary conditions.
TRANSV	= T; This option allows the program to perform unsteady calculations.
	= F; Unsteady effect is neglected
SAVER	= T; This option allows the program to save the solution in a restart data file.
	= F; The solution is not saved to a restart data file.
HEX	= T; This option indicates to the program that the user wants to include a heat exchanger in the model.
	= F; This option indicates to the program that the user does not want to include a heat exchanger in the model.
HCOEF	= T; This option sets the program to calculate the overall heat transfer coefficient for a heat exchanger.
	= F; This option requires the user to supply the overall heat transfer coefficient for a heat exchanger.
REACTING	= F; This option is currently inactive and must be set to FALSE.
INERTIA	= T; The inertial effects of the fluid are considered.
	= F; The inertial effects of the fluid are neglected.
CONDX	= T; This option calculates heat conduction within the fluids.
	= F; Heat conduction within the fluids is neglected.
PRINTI	= T; This option prints out the initial flow field.
	= F; This option suppresses the print out of the initial flow field.
ROTATION	= T; This option allows the user to input branches with rotation.
	= F; Branch rotation is not activated.
BUOYANCY	= T; This option activates buoyancy when GRAVITY = .TRUE.
	= F; Buoyancy is not activated.
HRATE	= T; The applied heat load is in Btu/sec
	= F; The applied heat load is in Btu/lbm
INVAL	= T; This option allows the program to read the initial flow field from a named restart data file.
	= F; The initial flow field is not read from a data file.
MSORCE	= T; This option allows the user to specify a momentum source in a given branch.
	= F; The user does not want to specify a momentum source.
MOVBND	= T; This option allows the user to model a moving boundary in a node.
	= F; The moving boundary option is not activated.

Table 4.1.1 (Continued) - GFSSP Logical Variables

Variable	Meaning
----------	---------

TPA	= T; This option allows the user to include a turbopump assembly.
	= F; This choice indicates that the user does not want to include a turbopump assembly.
VARGEO	= T; This option allows the user to model a time dependent geometry.
	= F; The model geometry is considered to be invariant in time.
TVM	= T; This option allows the user to model the transport of momentum in the transverse direction.
	= F; The transverse transport of momentum is neglected.
SHEAR	= T; This option allows the user to model transport of momentum by shear.
	= F; Momentum transport due to shear is neglected.
PRINTIN	= T; This option prints the model input data file information into the output file.
	= F; Input data file information is not printed into the output file
PRNTADD	= T; This option prints additional thermodynamic and thermophysical property data into the output file.
	= F; Additional printout is not activated.

4.1.1.1 Compressibility Option

The preprocessor will set the compressibility option based on the users response to the following question:

‘IS DENSITY CONSTANT IN THE CIRCUIT?’

If the user answers ‘yes’ to this question, the program will assume a constant density within the fluid circuit and the user will be prompted to supply the density and viscosity of the fluid. If user answers ‘no’ to this question, the program will assume that the density can vary and will prompt the user, later in the model building process, to select the appropriate fluid or fluids, in the case of a mixture, from the GFSSP library of fluids. These related questions will be asked at the end of the ‘model options’ portion of the current preprocessor interactive session.

4.1.1.2 Transient Option

The preprocessor will set the transient option based on the users response to the following question:

‘IS FLOW TRANSIENT?’

GFSSP has the capability to model steady, quasi-steady and unsteady flow in any given flow system. If the user answers ‘no’ to the preceding question, a steady state flow will be assumed. If the user selects the transient option by responding to the preceding question with a ‘yes,’ the program will require the user to distinguish between the quasi-steady and unsteady formulations. Therefore, to make the distinction, the program asks the following question:

WILL NODE VOLUME BE SUPPLIED?"

The unsteady formulation requires node volume information. A 'yes' answer to this question will select an unsteady formulation, while a 'no' answer will lead to a quasi-steady formulation. For either the quasi-steady or the unsteady formulation, the user will be required to supply time step information by answering the following question:

INPUT TIME STEP, START TIME, AND FINAL TIME IN SECONDS

The numbers can either be separated by a comma or by a space. The 'enter' key must be pressed after the requested data has been input. If the user presses the enter key before supplying all of the data requested by the preprocessor, the program will not proceed until it receives the correct number of values.

For unsteady flow, the volume could be a function of time (e.g., positive displacement pump, squeeze film damper). The user can select that option by answering 'yes' to the following question:

WILL GEOMETRY BE FUNCTION OF TIME?

The user will be asked to provide the name of the data file containing the variable geometry information during the sequence of supplying boundary conditions.

4.1.1.3 Gravitation Option

The next preprocessor question is:

DO YOU WANT TO ACTIVATE GRAVITY?"

If the user answers 'yes' to this question, the program will account for gravity effects in determining a solution for the current model. In other words, the gravitational term in Equation 2.1.2 will be included. The user will be asked to supply the orientation of the flow direction in the branches with respect to the gravitational force vector (angle θ in Figure 2.1.2) during the 'branch information' portion of the current preprocessor interactive session.

The next prompt the user must respond to is:

DO YOU WANT TO ACTIVATE BUOYANCY?"

For a problem involving natural convection, the user must activate this option by responding with a 'y' at the prompt. In a situation where natural convection occurs, the fluid experiences a buoyancy force because of density differences in the presence of a gravitational field. Under the action of this force, the lighter fluid tends to move up. Therefore, the buoyancy force always acts in a direction opposite to the gravitational force. If this option is activated, the user must supply a

DO YOU WANT TO ACTIVATE ROTATION?"

GFSSP allows the user to model rotating flows in branches to account for the centrifugal forces on the fluid that occur in these branches due to the rotation. When the axis of rotation is not parallel to the main flow direction, the fluid experiences a centrifugal force. The magnitude of the centrifugal force depends on the radii of the axis of rotation and on the angular speed of the fluid. If this option is activated, the associated questions are asked in the 'additional information' portion of the current preprocessor interactive session.

4.1.1.6 External Momentum Source Option

The user has an option to add external momentum sources in a particular branch. This external momentum source is included as the S ' term in Equation 2.1.2 and could be due to any other body or surface force that was not accounted for elsewhere. The user can activate this option by answering 'yes' to the following question:

IS THERE ANY MOMENTUM SOURCE?"

If this option is activated, the associated questions are asked in the 'additional information' portion of the current preprocessor interactive session.

4.1.1.7 Axial Thrust Calculation Option

One of the main objectives for developing GFSSP was to provide a tool that could accurately calculate the axial thrust that occurs in a rocket engine turbopump. This option can be activated by answering 'yes' to the following question:

IS AXIAL THRUST CALCULATION REQUIRED IN THE CIRCUIT?"

This axial thrust is created when a pressure differential exists between opposing faces of a mechanism, such as a turbine disk. If the GFSSP axial thrust calculation option is activated, the user must supply surface areas normal to the thrust vector. If a normal vector to the input surface area aligns with the thrust vector, the magnitude of the surface area, in square inches (in^2), is entered with a positive sign. The surface area must be entered with a negative sign if a normal vector to the given surface area is opposite to the direction of the thrust vector. The user may choose to update the data file, using a text editor, to supply the areas once the data file is created. The user must answer 'n' to this option to avoid answering questions on areas during the interactive session.

4.1.1.8 Moving Boundary Option

A moving boundary is encountered in reciprocating machines and rotodynamic applications. There are two ways the user can model a moving boundary. In a steady state flow situation, the user is required to provide the surface area and velocity of the moving node. In an unsteady flow situation, the user is required to choose the variable geometry option. In either case, to select the moving boundary option the user must say 'yes' to the following question:

If this option is activated, the associated questions are asked in the 'additional information' portion of the current preprocessor interactive session.

4.1.1.9 External Heat Source Option

If the presence of heat sources or sinks in the flow circuit can affect the flow distribution, the user must activate this option by answering 'y' to the following question:

"ARE THERE ANY HEAT SOURCES?"

During the 'additional information' portion of the current preprocessor interactive session, the user will be required to identify the nodes where heat loads are applied and the magnitude of the heat loads in each of the identified nodes.

4.1.1.10 Heat Conduction Within Fluids Option

If the user wants to consider heat conduction within the fluid, the user can activate the heat conduction option between nodes by answering 'y' to the following question:

"DO YOU WANT TO ACTIVATE HEAT CONDUCTION?"

If this option is activated, the user must supply the distances between nodes and the cross sectional flow area normal to the heat conduction path during the 'branch information' portion of the current preprocessor interactive session.

4.1.1.11 Heat Exchanger Option

If the user wants to include any heat exchangers in the current model, the user must answer 'y' to the following question:

"IS THERE ANY HEAT EXCHANGER IN THE CIRCUIT?"

This option allows the user to include a heat exchanger, as shown in Figure 4.1.2, in a flow circuit. GFSSP has the capability to include ten heat exchangers in a flow system. If this option is activated, the user will be asked to provide the following information in the 'additional information' portion of the current preprocessor interactive session:

1. The number of heat exchangers to include in the circuit.
2. The branch names corresponding to the hot and cold branches of each heat exchanger.
3. User must select one of the following modeling methods:
 - a. Log Mean Temperature Difference Method
 - b. Effectiveness - NTU method.
4. If the user chooses the Effectiveness - NTU method, the code will ask the user either to supply the effectiveness or to identify the type of heat exchanger so that the program can calculate the effectiveness from the built-in relationships. The code has built-in relationships for counter flow and parallel flow heat exchangers. For other types, such as cross flow, multipass, the effectiveness must be supplied by the user.

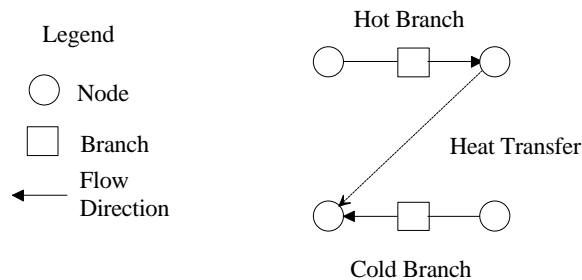


Figure 4.1.2 - Modeling of Heat Exchanger in a Flow System

4.1.1.12 Turbopump Option

If the user wants to include any turbopumps in the current model, the user must answer 'y' to the following question:

IS THERE ANY TURBOPUMP ASSEMBLY IN THE CIRCUIT?"

This option allows the user to include a turbopump assembly (TPA) in a flow circuit as shown in Figure 4.1.3. GFSSP has the capability to include ten turbopump assemblies in a flow system. If this option is activated, the user will be asked to provide the following information in the 'additional information' portion of the current preprocessor interactive session:

1. The number of turbopumps to include in the flow circuit.
2. Which branches correspond to the pump and turbine branches for each turbopump assembly.
3. The name of the file containing the pump performance characteristics.
4. The speed and efficiency of the turbine.

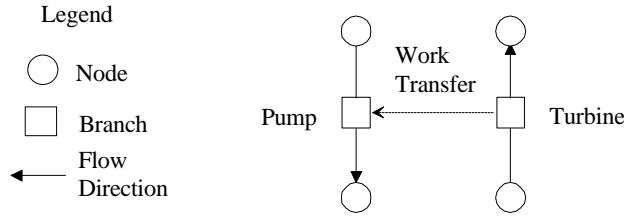


Figure 4.1.3 - Modeling of Turbopump in a Flow System

4.1.1.13 Fluid Selection Option

The next set of information the preprocessor will collect from the user concerns the fluid or fluids used in the flow circuit. The program will prompt the user to input whether or not multiple fluids are present in the system:

IS THE FLUID A MIXTURE?"

Once the user answers this question, either 'y' or 'n', the code will print a list of fluids. The fluids available with GASP and WASP are shown below and in Table 4.1.2.

GFSSP HAS A LIBRARY OF THE FOLLOWING FLUIDS:

- 1 - HELIUM
- 2 - METHANE
- 3 - NEON
- 4 - NITROGEN
- 5 - CARBON-MONOXIDE
- 6 - OXYGEN
- 7 - ARGON
- 8 - CARBON-DIOXIDE
- 9 - FLUORINE
- 10 - HYDROGEN
- 11 - WATER
- 12 - RP1

NOTE: RP1 PROPERTY RANGE HAS LIMITED VALIDITY;
 PRESSURE RANGE:.01 TO 650 PSI
 TEMPERATURE RANGE: 440 TO 600 R"

GFSSP can calculate the properties of the listed fluids. If the mixture option is not chosen, the user will need to identify only one fluid from the list. If the user answers 'y' to the previous question, the code will ask:

HOW MANY OF THESE FLUIDS ARE PRESENT IN THE CIRCUIT?"

The user must provide the total number of fluids as well as identify the index number of each fluid that is present from the given list.

Table 4.1.2 - Fluids Available in GFSSP

Index	Fluid	Index	Fluid
1	HELIUM	7	ARGON
2	METHANE	8	CARBON DIOXIDE
3	NEON	9	FLUORINE
4	NITROGEN	10	HYDROGEN
5	CARBON MONOXIDE	11	WATER
6	OXYGEN	12	RP-1

4.1.1.14 Transverse Momentum Transport Option

GFSSP can be used to model multidimensional flow by extending the momentum conservation equation to account for momentum transport by the transverse component of the velocity and shearing stress. GFSSP's flow network of nodes and branches must be visualized as a grid network for a finite volume formulation of mass and momentum conservation equations. Instead of considering branches as a pipe and nodes as the junction of interconnected pipes, we may visualize the nodes as being the locations in a fluid continuum where all scalar properties such as pressures, temperatures and concentrations are located and the branches are paths through the fluid medium that connect the adjacent nodes. The vector properties, such as flow rate, are assumed to exist at a point located midway between two connected nodes.

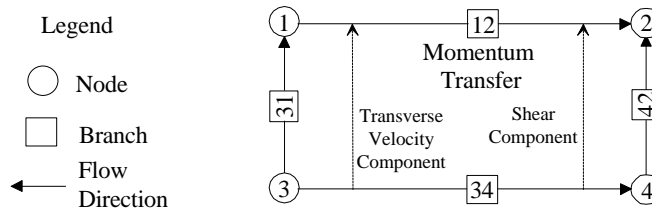


Figure 4.1.4 - Multidimensional Flow Modeling by GFSSP

Figure 4.1.4 shows a multidimensional flow situation as modeled by GFSSP. In a multidimensional environment, the momentum conservation in branches 12 and 34 must account for inter-branch transport of momentum. There are two possible mechanisms of transport. First, momentum can be transported by shear when there is a velocity gradient between two parallel branches. Secondly, the momentum transport occurs due to transverse flow through branches 31

and 42. To activate multidimensional flow the user must respond with a 'y' to the two following questions asked by the preprocessor:

DO YOU WANT TO ACTIVATE TRANSVERSE MOMENTUM?"

DO YOU WANT TO ACTIVATE SHEAR?"

Once these options are activated, the user is required to supply the neighboring branch information for each branch where transverse momentum and shear transport are required to be calculated.

4.1.1.15 Print Control Option

The last of the program options that the user will be asked to set, relates to the data that will be included in the output file. The user will be asked to select three print control options. First, the code will ask:

DO YOU WANT PRINT OUT OF INITIAL FIELD VALUES?"

If the user does not need to see the initial solution, the user should respond with a 'n' to this question. This will be followed by a second question:

DO YOU WANT TO SUPPRESS PRINT OUT OF INPUT DATA?"

By accepting this option, the user can suppress the inclusion of information in the output file that is already available in the input data file.

GFSSP also provides an additional print control option used to activate an extended print out. If the user is interested to see a print out of enthalpy, entropy and thermophysical properties, an affirmative answer to the following question is required:

DO YOU WANT EXTENDED PRINT OUT?"

4.1.2 Node Information

Following the completion of the portion of the interactive session that sets the required program logical options, the preprocessor prompts the user to input information for the nodes that will be used for the current model. The user will first be required to supply the total number of nodes. The code will then ask to designate a number for each of the nodes. The numbering scheme is completely arbitrary. The user can devise any numbering scheme, using a maximum of four digits.

Next, the user is required to specify the type of each of the nodes. GFSSP allows two types of nodes. A node could be either an internal node or a boundary node. The code calculates pressures, temperatures and mixture concentrations at the internal nodes. The pressures, temperatures and concentrations must be supplied in the boundary nodes. GFSSP does not use the temperatures and concentrations at the outflow boundary nodes. However, the user must supply those values because GFSSP does not distinguish between inflow and outflow boundary nodes during problem setup. A boundary node can have either an inflow or outflow depending upon the specified boundary conditions.

The format of the questions in this portion of the current interactive session are as follows:

ENTER TOTAL NUMBER OF NODES

ENTER NUMBER ASSIGNED TO NODE 1

IS IT AN INTERNAL NODE?

The preprocessor will repeat the last two questions until the number of nodes input by the user has been reached.

4.1.3 Branch Information

After the user has completed the portion of the current interactive session that inputs the node information, the preprocessor prompts the user to input information for the branches that will be used for the current model. Every node in the circuit is connected to the circuit through at least one branch. The code will visit every internal node, identified by the user in the previous session, and ask the user to supply the number of branches connected with each internal node.

Next the preprocessor will prompt the user to supply the following information for each branch specified previously:

- a) A branch number within four digits.
- b) The assumed upstream and downstream nodes for the given branch.
- c) A branch type (resistance option) and all of the necessary information required for that resistance option.

If the gravity option is activated, the user must supply the angle that the branch makes with the gravity vector. If the heat conduction option is activated, the user must also supply the distances between the nodes and the cross sectional flow area normal to the heat conduction flux.

The format of the questions in this portion of the current interactive session are as follows (the responses to the prompts are in bold):

HOW MANY BRANCHES ARE CONNECTED WITH NODE 2?

2
 ENTER BRANCH NUMBER(J = 1 OF 2) OF NODE 2
12
 ENTER UPSTREAM NODE OF BRANCH NO. 12
1
 ENTER DOWNSTREAM NODE OF BRANCH NO. 12
2
 SELECT RESISTANCE OPTION FOR BRANCHES:
 OPTION - 1: PIPE FLOW
 OPTION - 2: FLOW THROUGH RESTRICTION
 OPTION - 3: NON-CIRCULAR DUCT
 OPTION - 4: PIPE FLOW WITH ENTRANCE & EXIT LOSS
 OPTION - 5: THIN SHARP ORIFICE
 OPTION - 6: THICK ORIFICE
 OPTION - 7: SQUARE REDUCTION
 OPTION - 8: SQUARE EXPANSION
 OPTION - 9: ROTATING ANNULAR DUCT
 OPTION - 10: ROTATING RADIAL DUCT
 OPTION - 11: LABY SEAL
 OPTION - 12: FACE SEAL
 OPTION - 13: COMMON FITTINGS & VALVES
 OPTION - 14: PUMP CHARACTERISTICS
 OPTION - 15: PUMP POWER PRESCRIPTION
 OPTION - 16: VALVE WITH GIVEN CV
 OPTION - 17: VISCOJET RESTRICTION
1
 ENTER LENGTH (IN), DIAMETER (IN), & ROUGHNESS FACTOR(E/D) OF
 BRANCH 12
10,0.5,0.005
 ENTER ANGLE WITH GRAVITY VECTOR (90 DEG FOR HORIZONTAL AXIS)
 FOR BRANCH NO. 12
100

The preceding screen prompts will be repeated until all of the branch information has been entered. One branch naming convention that has been found to be useful in modeling flow circuits, is to give the branch a number that is a combination of the numbers assigned to the nodes that the branch connects. For example, if there are two nodes, Node 1 and Node 2, that are connected by one branch, the name for that branch would be 12 according to this convention. This technique has been found to be helpful in facilitating the comprehension of the model results.

4.1.4 Boundary Conditions

In this portion of the current interactive session, the user is required to supply pressures, temperatures and concentrations at all of the boundary nodes. For a steady state model, the format of the dialogue is as follows:

ENTER PRESSURE (PSIA) & TEMPERATURE (DEG F) FOR NODE 1”

This screen prompt will be repeated for each boundary node that has been identified.

For transient calculations, the user is required to supply the filename containing the history data for each boundary node in the model. The user will be prompted to provide the following information:

ENTER FILENAME (20 CHARACTERS) FOR READING DATA HISTORY FOR
NODE 1

The format of the file is described in Example 8 of Chapter 5.

4.1.5 Additional Information

The following subsections discuss additional information that is required by the preprocessor that is dependent on which program options the user has incorporated into the current model. If the option has not been selected the associated prompts will not be displayed by the program.

4.1.5.1 Mass Source

The user will be prompted to declare whether or not there are internal nodes that have specified flow rates entering or leaving the node, excluding what the program calculates in the connected branches, with the following question:

HOW MANY INTERNAL NODES HAVE SPECIFIED FLOWRATES?”

GFSSP requires the specification of pressure at all of the boundary nodes. The code calculates the flow rates in all of the branches. The code, however, has been provided with the capability of accepting a mass source or sink in the internal nodes. The user will enter a 0 if there is no such mass source in the circuit. Otherwise, the actual number of internal nodes with mass flowing into, or out of, must be typed. The code then will ask user to provide the following information for the supplied number of internal nodes:

- a) The internal node number.
- b) The mass source (a positive number) or mass sink (a negative number) in lb/sec.

The format of the questions will be as follows (the responses to the prompts are in bold):

HOW MANY INTERNAL NODES HAVE SPECIFIED FLOWRATES?

1
 SPECIFY INTERNAL NODE NUMBER
2
 SPECIFY FLOWRATE(LBM/S) FOR NODE **2**
1.0

Depending on the number entered by the user to the first question, the last two questions will be repeated until the necessary information has been entered.

4.1.5.2 Heat Source

The program will prompt the user to declare how many of the internal nodes have heat added to or removed from them with the following question:

HOW MANY INTERNAL NODES HAVE SPECIFIED HEAT SOURCES?"

The user is prompted to supply the number of internal nodes where there are heat sources or sinks in the circuit. The user must enter a 0 if there is no such source in the circuit. Otherwise, the actual number must be typed. The heat source can be specified in either BTU/lbm or in BTU/sec. The user must select the option (See Table 4.1, program option variable HRATE). The code then will ask the user to provide the following information for the input number of internal nodes:

- a) The internal node number.
- b) The heat source flux (a positive number) or sink flux (a negative number) in appropriate units.

The format of the questions will be as follows (the responses to the prompts are in bold):

HOW MANY INTERNAL NODES HAVE SPECIFIED HEAT SOURCES?
1
HEAT SOURCE SPECIFICATION HAS TWO OPTIONS
 1. SOURCE IN BTU/SEC
 2. SOURCE IN BTU/LBM
SELECT YOUR OPTION
2
SPECIFY INTERNAL NODE NUMBER
3
SPECIFY HEAT SOURCE(BTU/LBM) FOR NODE **3**
1.0

Depending on the number entered by the user to the first question, the last three questions will be repeated until the necessary information has been entered.

4.1.5.3 Buoyancy

If buoyancy was activated earlier in the current interactive session, the following question will be asked:

SPECIFY REFERENCE NODE FOR DENSITY”

When buoyancy is activated, the gravitational term in Equation 2.1.2 is replaced by a buoyancy term that requires a density difference term instead of a density. The user must define the reference node for the density difference term. It is preferable to select a boundary node to be the reference node.

4.1.5.4 Rotation

If the rotational option was activated earlier in the current interactive session, the user will be prompted with the question:

HOW MANY BRANCHES HAVE THE ROTATING FLOWS?”

Once the user answers this question, the code will ask the user to provide the following information for the user supplied number of branches:

- a) The branch number.
- b) The upstream and downstream radius of the branch with respect to the axis of rotation.
- c) The rotational speed and the factor representing the extent of the rotation of fluid with respect to the solid boundary.

The format of the questions will be as follows (the responses to the prompts are in bold):

HOW MANY BRANCHES HAVE ROTATING FLOWS?
1
SPECIFY BRANCH NUMBER
12
INPUT UPSTREAM AND DOWNSTREAM RADIUS OF BRANCH 12
1.0,3.0
INPUT ROTATIONAL SPEED (RPM) AND KFACTOR OF BRANCH 12
30,000,0.25

Depending on the number entered by the user to the first question, the last three questions will be repeated until the necessary information has been entered.

4.1.5.5 Momentum Source

If the momentum source option was activated earlier in the current interactive session, the following questions will be asked (the responses to the prompts are in bold):

HOW MANY BRANCHES HAVE MOMENTUM SOURCES?
1
SPECIFY BRANCH NUMBER
34
INPUT MOMENTUM SOURCE IN LBF FOR BRANCH 34
1.0

Depending on the number entered by the user to the first question, the last two questions will be repeated until the necessary information has been entered.

4.1.5.6 Heat Exchanger

If the heat exchanger option was activated earlier in the current interactive session, the following questions will be asked (the responses to the prompts are in bold):

HOW MANY HEAT EXCHANGERS(HEX) ARE THERE?
1
SPECIFY HOT & COLD BRANCH FOR HEX 1
12,34
SELECT YOUR MODELING OPTION
1. LOG MEAN TEMP. DIF.(LMTD)
2. EFFECTIVENESS-NTU
SELECT 1 OR 2
2
SUPPLY EFFECTIVENESS OF HEAT EXCHANGER 1
EFFECTIVENESS BETWEEN 0 TO 1,GFSSP USES THE VALUE
EFFECTIVENESS GREATER THAN 1,GFSSP COMPUTES THE VALUE
1.5
SPECIFY TYPE OF HEAT EXCHANGER
1 : COUNTER FLOW
2 : PARALLEL FLOW
1

Depending on the number entered by the user to the first question, the last four questions will be repeated until the necessary information has been entered.

4.1.5.7 Moving Boundary

If the moving boundary option was activated earlier in the current interactive session, the following questions will be asked (the responses to the prompts are in bold):

' HOW MANY NODES HAVE MOVING BOUNDARY? '

1

' SPECIFY NODE NUMBER '

10

'SPECIFY AREA(IN**2) & VELOCITY (FPS) OF 10 '

1,10

Depending on the number entered by the user to the first question, the last two questions will be repeated until the necessary information has been entered.

4.1.5.8 Turbopump

If the turbopump option was activated earlier in the current interactive session, the following questions will be asked (the responses to the prompts are in bold):

HOW MANY TURBOPUMPS ARE IN THE CIRCUIT?

1

SPECIFY PUMP & TURBINE BRANCH FOR TPA 1

12,34

ENTER FILENAME(20 CHARACTERS) OF PUMP PERFORMANCE CURVE FOR
BRANCH 12

PCURVE.DAT

SPECIFY SPEED AND EFFICIENCY OF TURBINE FOR BRANCH 34

30000,0.85

Depending on the number entered by the user to the first question, the last three questions will be repeated until the necessary information has been entered.

4.1.5.9 Multidimensional Flow

If the multidimensional flow option was activated earlier in the current interactive session, the following questions will be asked (the responses to the prompts are in bold):

4.1.5.9.1 Transverse Momentum

The following preprocessor prompts request information needed by the program to calculate the transverse momentum component of the flow:

INPUT BRANCH FOR WHICH TRANSVERSE MOMENTUM WILL BE
 CALCULATED
12
 INPUT NUMBER (MAX 10) OF PARALLEL BRANCHES TO BRANCH 12
1
 INPUT NAME OF 1 BRANCH WHICH IS PARALLEL TO 12
34
 INPUT ANGLE BETWEEN 12 & 34
0
 INPUT NUMBER OF TRANSVERSE BRANCHES CONNECTING BRANCH 12
 AND 23 (MIN 2, MAX 5)
2
 INPUT NAME OF 1 TRANSVERSE BRANCH BETWEEN 12 & 34
31
 INPUT ANGLE BETWEEN 12 & 31
90
 INPUT NAME OF 2 TRANSVERSE BRANCH BETWEEN 12 & 34
42
 INPUT ANGLE BETWEEN 12 & 42
90

Depending on the number entered by the user to the first question, the last eight questions will be repeated until the necessary information has been entered.

4.1.5.9.2 Shear

The following preprocessor prompts request information needed by the program to calculate the shear component of the flow:

HOW MANY BRANCHES WILL HAVE SHEAR CALCULATED?
1
 INPUT BRANCH FOR WHICH SHEAR WILL BE CALCULATED
12
 INPUT NUMBER (MAX 10) OF PARALLEL BRANCHES TO BRANCH 12
1
 INPUT NAME OF 1 PARALLEL BRANCH TO 12
34
 INPUT ANGLE BETWEEN 12 AND 34
0
 INPUT SHEAR AREA (IN²) & DISTANCE (IN) BETWEEN 12 & 34
1.0,0.5
 INPUT NUMBER (MAX OF 10) OF SOLIDS/WALLS PARALLEL TO BRANCH
1

VELOCITY OF SOLID/WALL + ANGLE BETWEEN SOLID/WALL & 12
(ft/sec)

10.0,0.0

INPUT SHEAR AREA (in²) & DISTANCE (in) BETWEEN SOLID/WALL &
12

1.0,0.5

Depending on the number entered by the user to the first question, the last seven questions will be repeated until the necessary information has been entered.

4.1.5.10 Interactive Session Completion

At the end of the current interactive session, the user will be asked to provide a filename for use in naming the input and output data files for the current model and the preprocessor will also request the analyst's name. The questions that will be asked in sequence are:

ENTER FILENAME FOR WRITING THE INPUT DATA

ENTER FILENAME FOR WRITING THE OUTPUT DATA

ENTER ANALYST NAME

An example of the complete interactive session of creating a data file is provided in Appendix N.

4.2 VISUAL PREPROCESSOR

A Windows based preprocessor has been developed for GFSSP to allow for the building of a model using a menu driven format. This preprocessor was developed using Visual Basic 4.0 and is dubbed the Visual Preprocessor to distinguish it from the DOS based Command Line Preprocessor. The Visual Preprocessor has all of the features of the Command Line Preprocessor in an easy to use format. Additionally, the Visual Preprocessor allows the user to easily change input values and to open previously generated files for editorial changes. The requirements for the Visual Preprocessor are listed below in Table 4.2.1.

Table 4.2.1 - Visual Preprocessor Requirements

1)	A Personal Computer with Windows 95 or NT,
2)	1024 x 768 resolution monitor,
3)	15 Mb free hard disk space,
4)	8 Mb RAM (16 or more recommended)

The following is a step by step description of how to build a model using the Visual Preprocessor.

From the start button, go to programs to select the GFSSP Visual Preprocessor. This will initiate the start sequence for the preprocessor. The first window that will appear is shown in Figure 4.2.1, which asks if a previously developed file is to be opened. If the answer is yes, another window, Figure 4.2.2, will prompt for the name of that file (including path). If the answer is no, then the main window of the Visual Preprocessor will appear (Figure 4.2.3) with default information listed. If a file is to be opened, the main window will appear with the required information already in place.

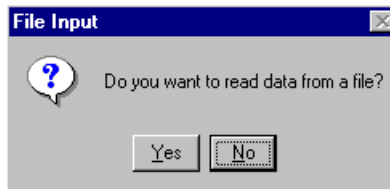


Figure 4.2.1 - Startup Window for the Visual Preprocessor

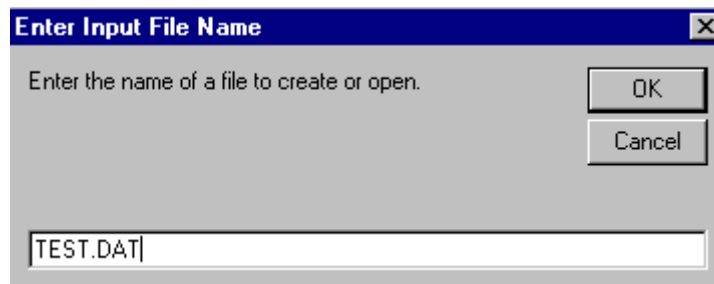


Figure 4.2.2 - File Name Window (For Opening or Writing to a File)

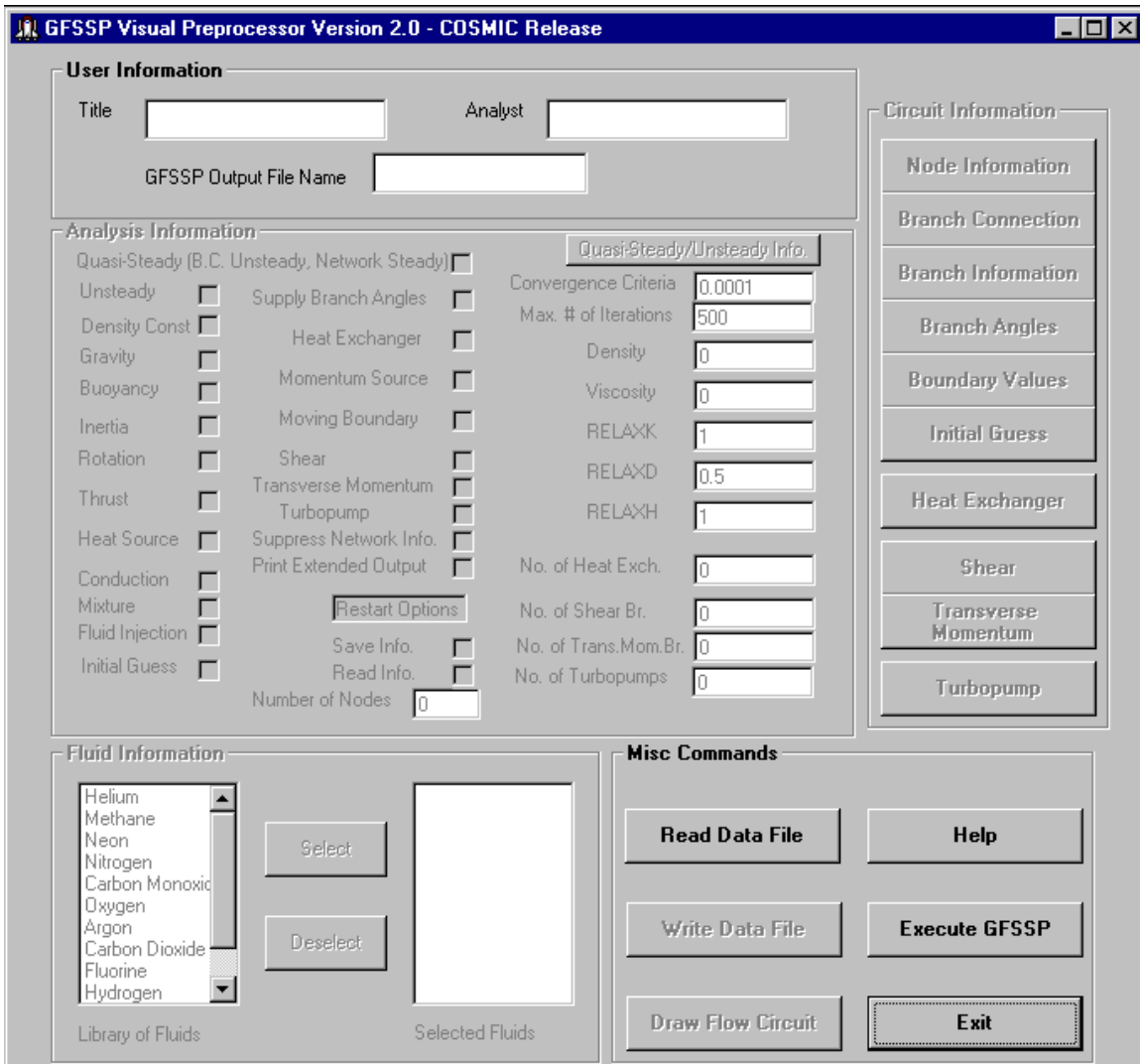


Figure 4.2.3 - Main Visual Preprocessor Window

The main menu requires information to be entered in a sequential manner similar to the Command Line Preprocessor. Only when the key information is entered will other portions of the main menu become active. Inactive menu items are indicated by a light gray text. The first bit of required information is the Title, Analyst and GFSSP Output File Name. After entering this information, the logical variables become active and the user can check off the desired logical variables along with entering control variable information (relaxation parameters, convergence criteria, etc.) and the number of nodes in the model. After the number of nodes in the model has been entered, the user must select a fluid from the library of fluids or choose the constant density option and enter the fluid density and viscosity.

Upon selection of a fluid, or entry of fluid properties, the button for Node Information becomes active. Pressing this button opens the node information window (Figure 4.2.4).

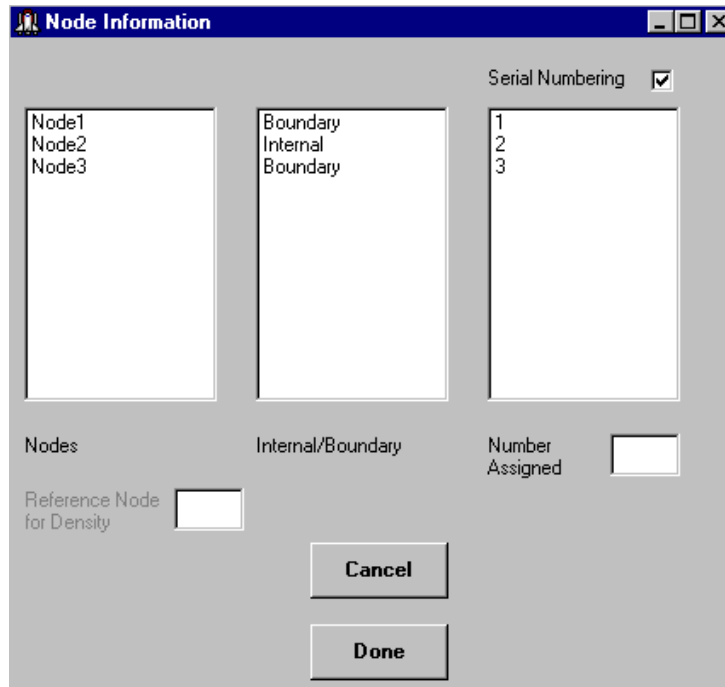


Figure 4.2.4 - Node Information Window

From this window, the user assigns the node numbers as well as the type of node (internal or boundary). In order to assign a node number, the user must highlight the desired node, then enter the node number in the text box corresponding to Number Assigned or if the user wishes, the user can select to serially number the nodes by checking the appropriate check box. The default node type is internal. To switch node type simply double click on the word Internal or Boundary corresponding to the desired node. When the user has completed inputting the node information, simply click on the Done button and return to the main menu where the Branch Connection button will become active.

The user will then click on the Branch Connection button and the node-branch connection window will open (Figure 4.2.5).

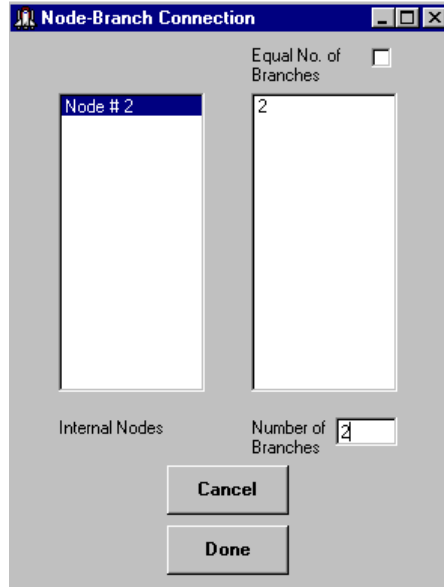


Figure 4.2.5 - Node/Branch Connection Window

For every internal node, the number of branches connected to it is required as well as the connecting branch numbers. In order to assign the number of branches connected to an internal node, highlight the internal node number on the left hand side of the window, then place the corresponding number of connecting branches in the lower right text box.

Upon entering the number of connecting branches for each internal node, highlight the number of branches corresponding to a given node to open the 'Branches for Node X' window (Figure 4.2.6), where X is the internal node number to which the listed branches are connected.

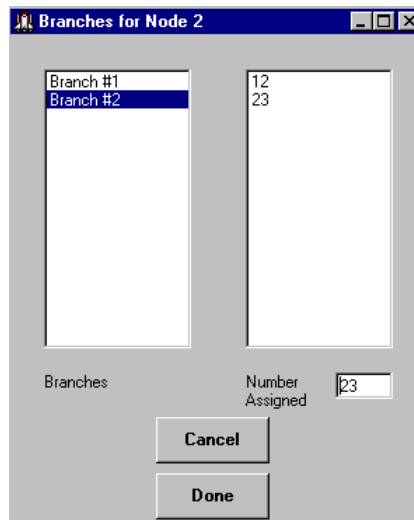


Figure 4.2.6 - Branch Numbering Window

Highlight each branch listed in the left hand text box in this window and assign a branch number to each branch in the lower right hand text box. When all branches have been assigned a branch number press the Done button to return to the Node-Branch Connection window. Highlight the number of branches corresponding to the next internal node to open the Branches for Node X window again and assign branch numbers for the corresponding internal node. This process is repeated until all of the internal nodes have been assigned a number of connected branches and each of these branches has been assigned a branch number. Note that a branch connecting two internal nodes must have been assigned the same branch number for each node (i.e. if branch 12 connects internal nodes 1 and 2, then both nodes 1 and 2 must list a branch with the number 12 assigned). Upon completion of specifying the number of branches for each internal node and branch numbers corresponding to these branches, press the Done button in the Node-Branch Connection window and return to the main menu window.

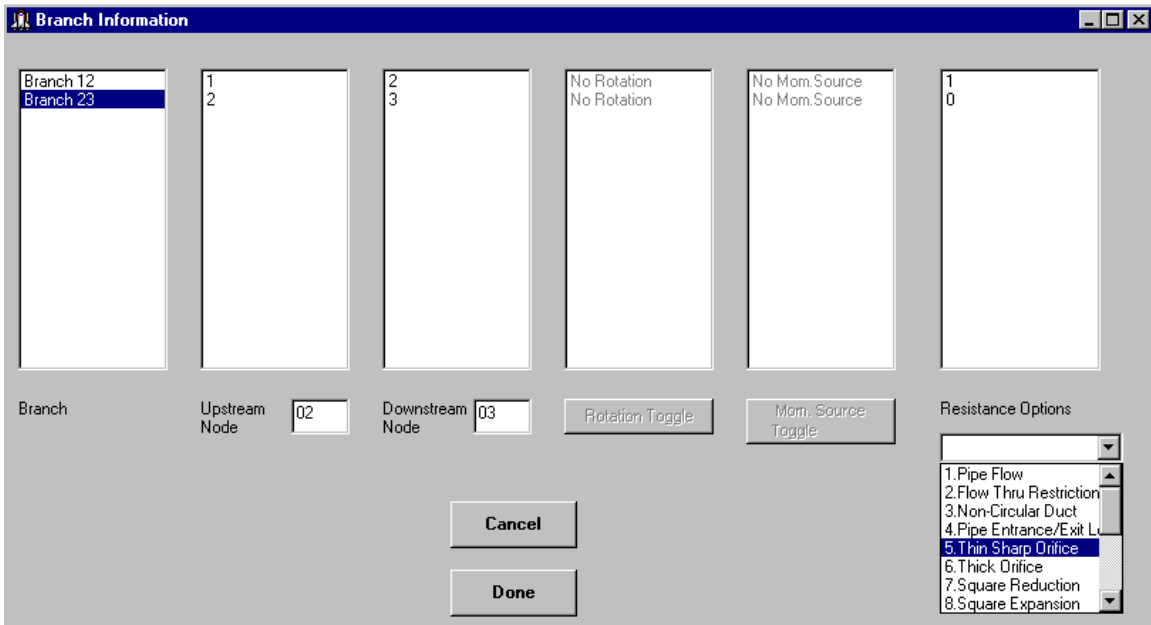


Figure 4.2.7 - Branch Information Window

For each branch, click on the branch, list the upstream and downstream nodes in the appropriate text boxes and choose a resistance option from the pull down menu on the lower right hand side of the window. If the rotation or the momentum source options were chosen on the main menu then their buttons would be active, allowing the user to toggle which branches would have these options. Once a resistance option has been chosen, the user should click on the option number in the right hand text box in order to pull up the appropriate window to input information required for the selected resistance option (Figures 4.2.8 is the Resistance Option 1 window).

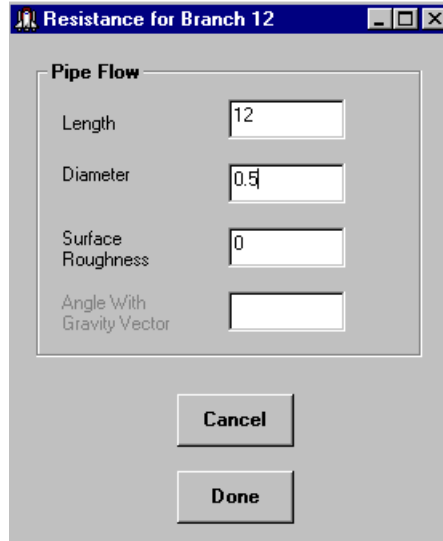


Figure 4.2.8 - Resistance Option 1 Window

The user will then place the required information into the appropriate resistance option window, followed by pressing the Done button. If either a momentum source or rotation is chosen for a particular branch, the user must click on the word(s) Rotation or Mom. Source to open a window in which to place the required information. When the user has completed inputting the required information for each branch, press the Done button on the Branch Information window to return to the main menu window.

The Boundary Values button on the main menu is now active as well as the Branch Angles button if the Supply Branch Angles logical flag has been activated. Click on the Boundary Values button to open the Boundary Values window. Figure 4.2.9 depicts the Boundary Values window for a steady state model.

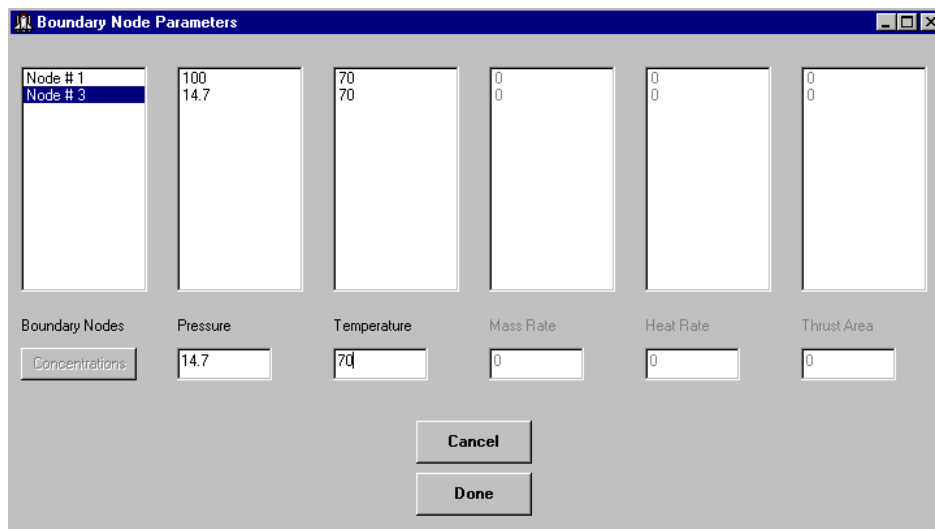


Figure 4.2.9 - Boundary Values Window (Steady)

In the steady state **Boundary Values** window, click on each boundary node and enter the appropriate information in the text boxes provided. For a transient model, Figure 4.2.10 depicts the transient **Boundary Values** window.

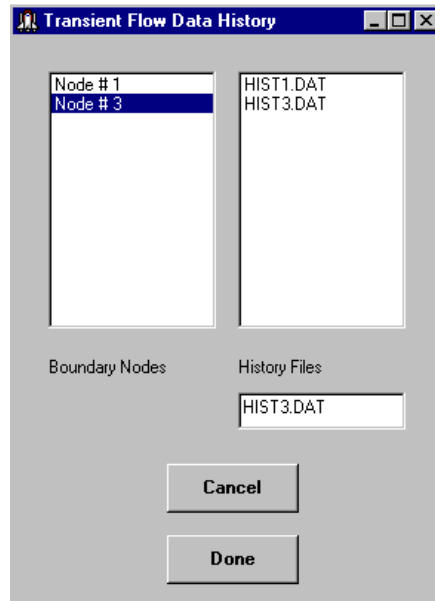


Figure 4.2.10 - Boundary Values Window (Unsteady)

In the transient **Boundary Values** window, click on each boundary node and enter the appropriate history file name in the text boxes provided. Upon exiting either of the **Boundary Values** windows, the **Initial Guess** button will become active if the user has chosen the **Initial Guess** logical variable check box.

If branch angles are to be supplied (for inertia calculations), press the **Branch Angles** button to open the **Branch Angles** window (Figure 4.2.11). For each branch, click on the branch and then click on each upstream branch and input the angle between the branches in the text box for upstream branches. Repeat this process for the downstream branches. Click the **Done** button when all of the required information has been supplied to return to the main menu window.

The Visual Preprocessor will supply a default initial guess for a steady state model that is based upon the boundary node conditions. If the user wishes to supply a different initial guess for steady state models, or initial conditions for a transient model, click on the **Initial Guess** button to

For each internal node, click on each node and enter the appropriate information in the text boxes provided. For a transient model, input any node volume that is in addition to any connected branch volumes (if the user does not have any additional volume, set this volume to zero). The program will calculate and assign one half of the attached branch volumes to each of the internal

nodes. When the required information has been supplied, press the Done button to return to the main menu window.

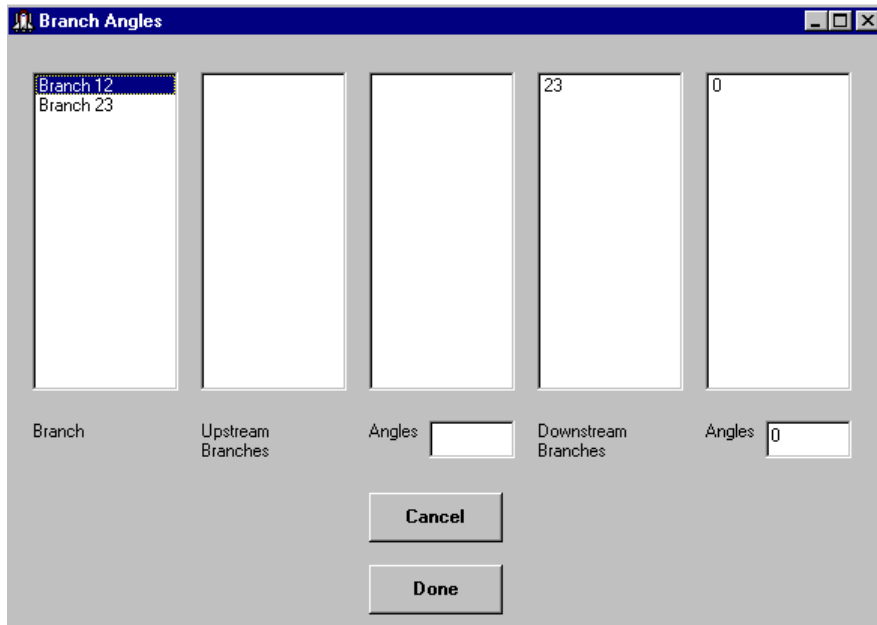


Figure 4.2.11 - Branch Angles Window

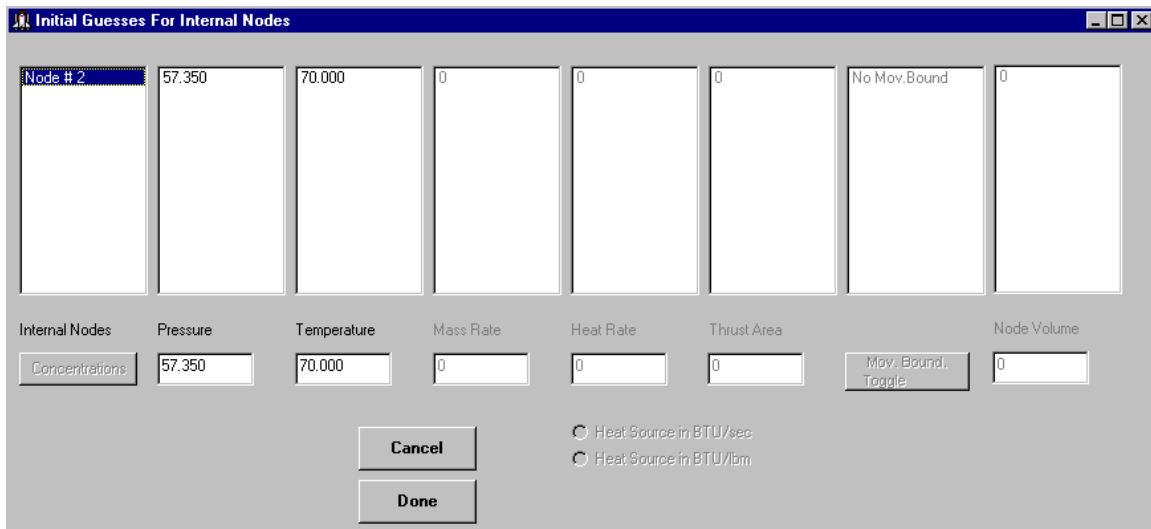


Figure 4.2.12 - Initial Guess Window

If a transient model is to be run (either quasi-steady or fully unsteady), the Quasi-Steady/Unsteady Info. button will become active. Press this button to open the Unsteady Information window

(Figure 4.2.13). Enter the required information and press the Done button to return to the main menu window.

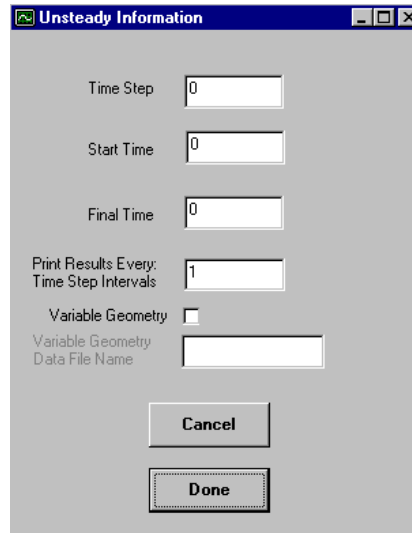


Figure 4.2.13 - Unsteady Information Window

After the user has completed inputting all required information, the model is saved by pressing the Write Data File button on the main menu window. When this button is pressed, the File Name window (Figure 4.2.2) will open and the user will supply the input data file name. Once the file has been saved, the user can then press the Execute GFSSP button to begin execution of GFSSP to run the model.

The user can open a previously built model by pressing the Read Data File button from the main menu window. The File Name window (Figure 4.2.2) will then open and the user must supply the name of the file to be opened, including the path if the file is in another directory.

When the user is ready to exit the Visual Preprocessor, press the Exit button on the main menu window. A prompt will ask if the user wishes to save the model to a file. If there are no changes to the model since it was last saved, there is no need to resave.

4.3 DESCRIPTION OF THE INPUT DATA FILE

The previous sections describe how to create a GFSSP model of a fluid flow network using the available GFSSP preprocessors. The following section describes the structure of the GFSSP input data file that is created by the preprocessors. The data in the GFSSP input data file can be classified into the following 18 sections. The data file from Example 1 was used as an example within this section to explain the content of the different sections of the input file:

1. Version Number:

The code prints a three digit version number:

```
GFSSP VERSION
  200
```

2. Analyst's Name:

The code prints the analyst's name with a limit of 40 characters:

```
ANALYST
ALOK MAJUMDAR
```

3. Input Data File Name:

The code prints the input data file name with a limit of 40 characters:

```
INPUT DATA FILE NAME
EX1GP.DAT
```

4. Output Data File Name:

The code prints output data file name with a limit of 40 characters:

```
OUTPUT FILE NAME
EX1GP.OUT
```

5. Title:

The user can specify a model title of 80 characters or less:

```
TITLE
PUMP-SYSTEM CHARACTERISTICS
```

6. Logical Variables:

The settings for the twenty six program logical variables are provided in next four lines:

```
DENCON  GRAVITY  ENERGY  MIXTURE  THRUST  STEADY  TRANSV  SAVER
  F      T      T      F      F      T      F      F
HEX  HCOEF  REACTING  INERTIA  CONDX  PRINTI  ROTATION
  F      F      F      F      F      F      F
BUOYANCY  HRATE  INVAL  MSORCE  MOVBNB  TPA  VARGEO  TVM
  F      F      F      F      F      F      F      F
SHEAR  PRNTIN  PRNTADD
  F      T      T
```

Table 4.1.1 contains a listing of the GFSSP logical variables and their options.

7. Node, Branch and Fluid Information:

NNODES - Number of nodes.
 NINT - Number of internal nodes.
 NBR - Number of branches.
 NF - Number of fluids.

NNODES	NINT	NBR	NF
4	2	3	1

8. Relaxation Parameters:

RELAXK - Under relaxation parameter for resistance (Recommended value = 1.0).
 RELAXD - Under relaxation parameter for density (Recommended value = 0.5).
 RELAXH - Under relaxation parameter for enthalpy (Recommended value = 1.0).

RELAXK	RELAXD	RELAXH	CC	NITER
1.000	0.500	1.000	0.100E-03	500

9. Index Number for Fluids:

The index number for each fluid is printed in this line. Table 4.1.2 shows the fluids that are available in GFSSP.

```
NFLUID(I), I= 1,NF
11
```

10. Nodes and Indices:

In this section, the node numbers, NODE(I), and their indices, INDEX(I), are printed. INDEX(I) = 1 implies an internal node and INDEX(I) = 2 indicates a boundary node.

NODE	INDEX
1	2
2	1
3	1
4	2

11. Node Information:

In this section, pressure, temperature, mass source, heat source and node areas are printed sequentially. Node areas are required only when the axial thrust calculation option is activated.

For transient calculations, the code reads pressure, temperature and concentrations from a file defined in this part of the data file.

For STEADY = .TRUE.

NODE	PRES (PSI)	TEMP(DEGF)	MASS SOURC	HEAT SOURC	THRST AREA
1	0.1470E+02	0.6000E+02	0.0000E+00	0.0000E+00	0.0000E+00
2	0.1000E+02	0.6000E+02	0.0000E+00	0.0000E+00	0.0000E+00
3	0.4803E+02	0.6000E+02	0.0000E+00	0.0000E+00	0.0000E+00
4	0.1470E+02	0.6000E+02	0.0000E+00	0.0000E+00	0.0000E+00

For STEADY = .FALSE.

NODE	PRES (PSI)	TEMP(DEGF)	MASS SOURC	HEAT SOURC	THRST AREA
1	0.1000E+03	0.8000E+02	0.0000E+00	0.0000E+00	0.1728E+05

hist2.dat

12. Branch Connection:

The number of branches connected to every internal node is defined within this portion of the input file. Also included in this section are the names of the connected branches.

INODE(I) - Internal node number.

NUMBR(I) - Number of branches connected with the Ith internal node.

NAMEBR(I,J), J = 1, NUMBR(I) - Name of the branches connected with the Ith internal node.

INODE	NUMBR	BRANCH 1	BRANCH 2	BRANCH 3	BRANCH 4	BRANCH 5	BRANCH 6
6							
	2	2	12	23			
	3	2	23	34			

13. Branch Information:

Branch information is provided in this section. In the first part of this section, the branch number, upstream node, downstream node and selected resistance option are printed. In the second part, the required input parameters of every branch are printed in the same order as in the first part. A header is printed for every branch describing the required input parameters.

BRANCH	UPNODE	DNNODE	OPTION		
12	1	2	14		
23	2	3	13		
34	3	4	1		
BRANCH	OPTION -14:	PUMP CONST1,	PUMP CONST2,	AREA	
12	30888.00000	-0.00081	201.06177		
BRANCH	OPTION -13:	DIA, K1, K2,	AREA		
23	6.00000	1000.00000	0.10000	28.27431	
BRANCH	OPTION -1:	LENGTH, DIA,	EPSD, ANGLE,	AREA	
34	18000.00000	6.00000	0.00500	95.74000	28.27431

14. Inertia Information:

In order to account for the inertial effects in the fluid flow model, the velocity in the upstream branch is required along with the angle between the branches. During the course of the calculation, if the flow rate becomes negative, the designated downstream branch becomes the upstream branch. Therefore, in this section, all of the upstream and downstream branches, for each branch in the flow circuit, are defined. In the first part of this section, the number of upstream branches and their designated numbers are listed. In the second part, the number of downstream branches and their designated numbers are listed. Finally the information about the angle the branch makes with its upstream and downstream neighbors are printed. The default values of the angle are set to zero. If the user wants to modify these angles, the user must use a text editor to alter the data file.

```
BRANCH  NOUBR  NMUBR
    12      0
    23      1      12
    34      1      23
BRANCH  NODBR  NMDBR
    12      1      23
    23      1      34
    34      0
BRANCH
    12
    UPSTRM BR.  ANGLE
    DNSTRM BR.  ANGLE
    23      0.00
BRANCH
    23
    UPSTRM BR.  ANGLE
    12      0.00
    DNSTRM BR.  ANGLE
    34      0.00
BRANCH
    34
    UPSTRM BR.  ANGLE
    23      0.00
    DNSTRM BR.  ANGLE
```

15. Rotation Information:

When the option ROTATION is set to TRUE, this section provides the related information. First, the number of rotating branches is printed. This is followed by a table of related data:

BRANCH : Designated branch number.
RADU: Radial distance to the upstream node from the axis of rotation, in units of inches.
RADD: Radial distance to the downstream node from the axis of rotation, in units of inches.

RPM: Rotational speed of the branch in units of rpm.
 AKROT: Empirical factor representing the ratio of the fluid and the solid surface speeds.

A portion of the data file from Example 6 is shown here to illustrate the use of the rotation option.

NUMBER OF ROTATING BRANCHES					
9					
BRANCH	UPST RAD	DNST RAD	RPM	K ROT	
23	0.125000E+01	0.225000E+01	0.500000E+04	0.867100E+00	
34	0.225000E+01	0.362500E+01	0.500000E+04	0.815800E+00	
45	0.362500E+01	0.468750E+01	0.500000E+04	0.763000E+00	
56	0.468750E+01	0.537500E+01	0.500000E+04	0.725200E+00	
67	0.537500E+01	0.550000E+01	0.500000E+04	0.707600E+00	
89	0.550000E+01	0.537500E+01	0.500000E+04	0.712900E+00	
910	0.537500E+01	0.468750E+01	0.500000E+04	0.734900E+00	
1011	0.468750E+01	0.362500E+01	0.500000E+04	0.782400E+00	
1112	0.362500E+01	0.265000E+01	0.500000E+04	0.837600E+00	

16. Heat Exchanger

If the heat exchanger option is activated (HEX = .TRUE.), the data file contains the following variables:

NHEX: Number of heat exchangers in the circuit
 IBRHOT: Branch carrying hot fluid
 IBRCLD: Branch carrying cold fluid
 ITYPHX: Type of heat exchanger
 ARHOT: Heat transfer area on hot side
 ARCOLD: Heat transfer area on cold side
 UA: Product of overall heat transfer coefficient and area
 HEXEFF: Heat exchanger effectiveness

A portion of the data file from Example 5 is shown here to illustrate the use of the heat exchanger option.

NUMBER OF HEAT EXCHANGERS						
1						
IBRHOT	IBRCLD	ITYPHX	ARHOT	ARCOLD	UA	HEXEFF
23	67	1	0.00000	0.00000	1.10375	1.50000

17. Moving Boundary

If the moving boundary option is activated (MOVBND = .TRUE.), the data file contains the following variables:

NMNODE: Number of nodes with moving boundary

AREAN: Surface area of the node moving in to or away from the field

VBOUND: Velocity of the moving boundary (positive when moving into, negative when moving away from)

A portion of the data file from Example 7 is shown here to illustrate the use of moving boundary option.

```
NUMBER OF NODES WITH MOVING BOUNDARY
18
NODE      AREAN      VBOUND
 2  0.777126E+00  0.256180E+00
 3  0.777126E+00  0.484598E+00
 4  0.777126E+00  0.660503E+00
 5  0.777126E+00  0.764832E+00
 6  0.777126E+00  0.786280E+00
 7  0.777126E+00  0.722522E+00
 8  0.777126E+00  0.580468E+00
 9  0.777126E+00  0.375510E+00
10  0.777126E+00  0.129861E+00
11  0.777126E+00 -0.129861E+00
12  0.777126E+00 -0.375510E+00
13  0.777126E+00 -0.580468E+00
14  0.777126E+00 -0.722522E+00
15  0.777126E+00 -0.786280E+00
16  0.777126E+00 -0.764832E+00
17  0.777126E+00 -0.660503E+00
18  0.777126E+00 -0.484598E+00
19  0.777126E+00 -0.256180E+00
```

18. Restart

Two additional data files are required if the user has activated the restart option. If SAVER = .TRUE., the code saves node and branch information in two files. One file contains the node data and the other file contains the branch information. The user can select any twenty character name for these two files. In the example listed below the names of these files are FNODE.DAT and FBRANCH.DAT. If INVALID = .TRUE., the code reads node and branch information from these two data files for initial field values. This section of the data file appears as:

```
NODE DATA FILE
FNODE.DAT
BRANCH DATA FILE
FBRANCH.DAT
```