5. GRAPHICAL USER INTERFACE

This section introduces the Modeling Interface for GFSSP (MIG), the graphical user interface designed to simplify the model-building process for GFSSP. MIG allows the user to design GFSSP models by utilizing a 'point-and-click' workflow. This eliminates many of the more tedious and time-consuming aspects of model building such as the selection of unique ID numbers for nodes/branches and the specification of the upstream and downstream nodes for every branch. MIG also provides the ability to run models and view results with various tools for ease of analysis. Figure ____ shows the main MIG window which consists of menu and toolbar options and a blank canvas.

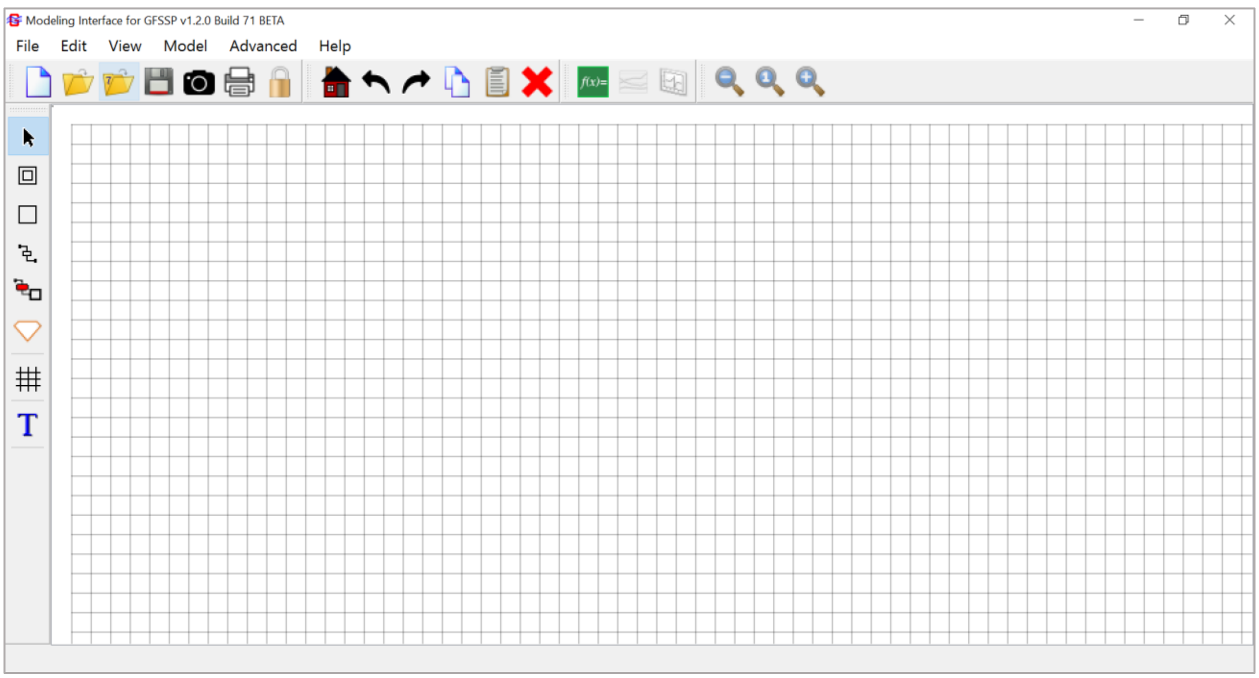


Figure: Main MIG window

5.1 Menus

5.1.1 File Menu

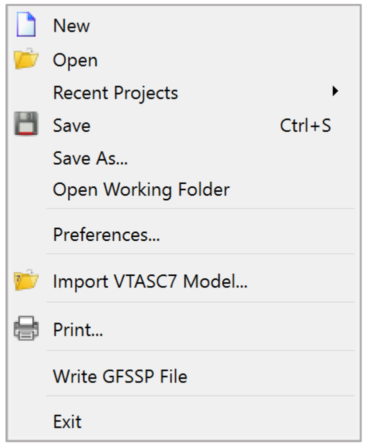


Figure: File dropdown menu in MIG

The file dropdown menu in MIG contains the necessary options to manage GFSSP model files and preferences. It contains options to create new models, open or import models, save models, and edit the application preferences.

New File

- Creates a new GFSSP model in MIG
- The window in Figure 3 prompts the user to enter a model title and analyst name

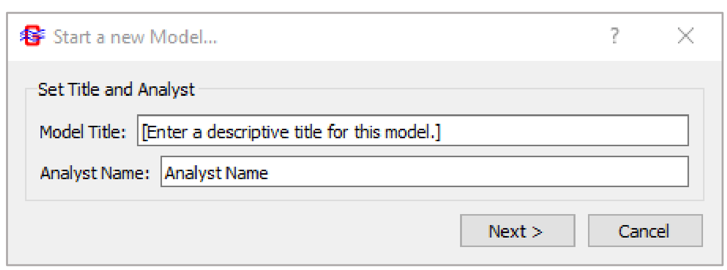


Figure: Start a new Model window

- The window in Figure 4 prompts the user to select the working folder (save location) for the model

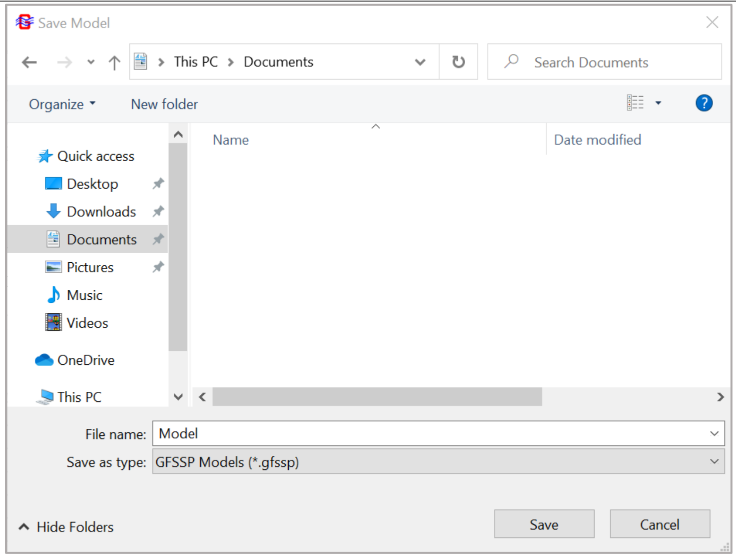


Figure: Windows file explorer used to select the model save location

- The window in Figure ____ prompts the user to set the model properties (detailed in section 5.2)

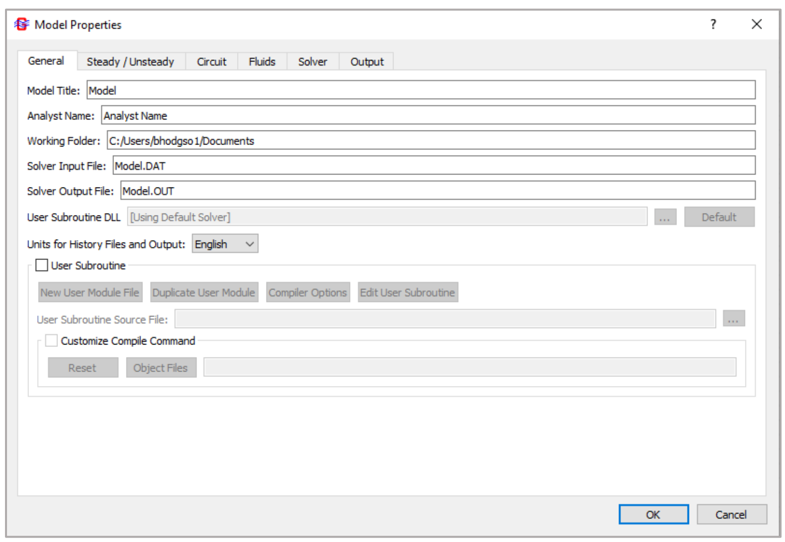


Figure: Model Properties window in MIG

- Toolbar shortcut: **New**

Open File	• Search the device file explorer to open GFSSP model files (*.gfssp) • Toolbar shortcut: **Open**
Recent Projects	• See the ten most recent models open in MIG by hovering the cursor over this option • Selecting one of the available projects will open the model in MIG
Save	• Saves the open GFSSP model

	• If the model has not previously been saved to the user's device, MIG will prompt the user to select a file name and save location • Toolbar shortcut: **Save** • Keyboard shortcut: **CTRL + S**
Save As	• Prompts the user to save the current GFSSP model under a new name and file location
Open Working Folder	• Opens the folder containing the current model in the device file explorer • Without a model currently open, an error popup will state *Working folder not set.*
Preferences	• Opens the application preferences window seen in Figure 6 • The user can edit general preferences such as analyst name, text editor, solver, and default project location • The Compiler Setup button can be used to identify the Fortran Compiler and select a preference for Free Format (*.f90) or Fixed Format (*.for) user subroutine files. • The Background Grid option allows the user to edit the canvas' background grid. This may also be accessed by right clicking the canvas and selecting Grid Setup. The user may set snap to grid, show grid, grid spacing, grid style, grid color, and grid opacity • The Custom Solver checkbox will display an additional dialog box on the General Tab of Model Properties for selecting a different executable (see Section 5.2.1) • The Copy Examples button will copy the Examples folder from the installation folder to the Default Project Location. (Windows does not permit Example models to run in the installation folder.) • The default units of quantities may be changed individually by using the dropdown menu for each unit • To change all units to the English or SI system, click either the Default English or Default SI button • Imported VTASC (.vts) model unit preferences are made with the VTS Import Units dropdown • Default history file and the output file units may be set by using the Default Units for History Files and Output dropdown • The user may edit the recent projects list using the adjustment and clear list widgets • To automatically load the most recent open project in MIG, select the Auto load most recent project on startup checkbox

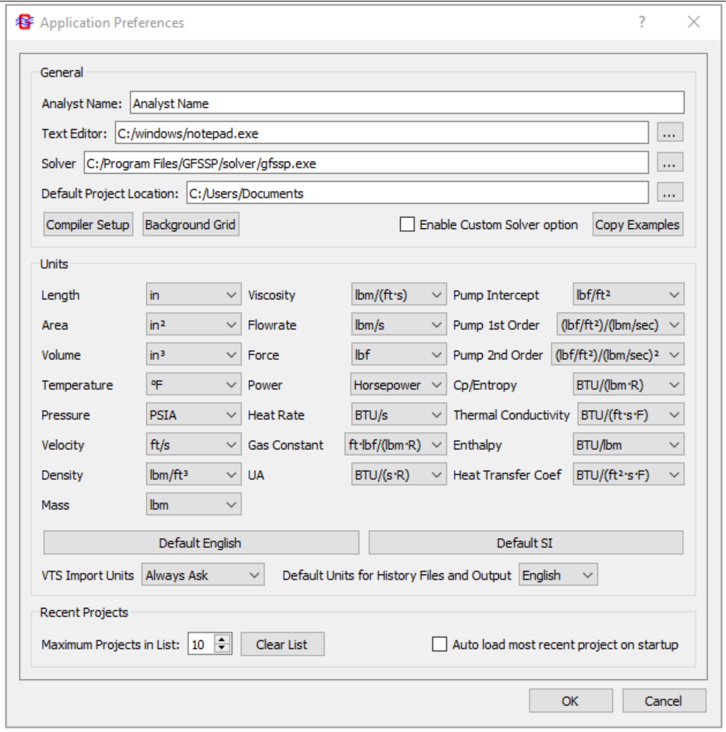


Figure: MIG Application Preferences window

Import VTASC7 Model	• Allows the user to import a VTASC model file (*.vts) into MIG • If the VTS Import Units are set to Always Ask in application preferences, MIG will prompt the user to pick the model units • The imported model may only be saved in the MIG format (*.gfssp) • Toolbar shortcut: Import VTASC7 Model
Print	• Opens the device print window to print out the model canvas • Toolbar shortcut: **Print**
Write GFSSP File	• MIG creates the GFSSP solver input file in the working folder • If the input file already exists, MIG will ask the user to overwrite the existing file
Exit	• Closes MIG

5.1.2 Edit Menu

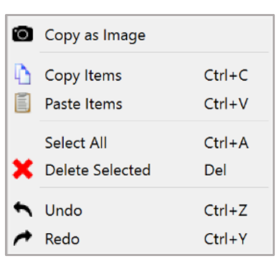


Figure: Edit dropdown menu in MIG

The edit dropdown menu in MIG provides operations to copy and paste items, select items, delete items, and undo and redo actions. It also allows the user to copy the MIG model as an image.

Copy as Image	• Copies an image of the model canvas to the device's clipboard • Toolbar shortcut**: Copy Model Image to Clipboard**
Copy Items	• Copies the selected model items to the device's clipboard • Toolbar shortcut: **Copy Items to Clipboard** • Keyboard shortcut: **CTRL + C**
Paste Items	• Pastes copied model items into the current model canvas • Toolbar shortcut: **Paste Items from Clipboard** • Keyboard shortcut: **CTRL + V**
Select All	• Selects all items in the model canvas • Keyboard shortcut: **CTRL + A**
Delete Selected	• Deletes all selected model items • Toolbar shortcut: **Delete Selected Items** • Keyboard shortcut: **DELETE**
Undo	• Undoes the previous action • Toolbar shortcut: **Undo** • Keyboard shortcut: **CTRL + Z**
Redo	• Redoes the previous action • Toolbar shortcut: **Redo** • Keyboard shortcut: **CTRL + Y**

5.1.3 View Menu

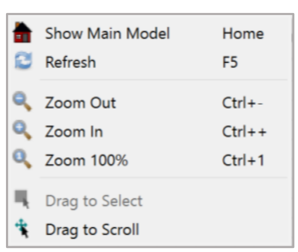


Figure: View dropdown menu in MIG

The view dropdown menu contains tools to edit the user's view of the model. It contains zoom, select, and scroll options with additional refresh and show main model options.

Show Main Model	• Returns the user to the main model screen if the user is viewing a Grid or SuperNode

	• Toolbar shortcut: **Show Main Model** • Keyboard shortcut: **Home**
Refresh	• Refreshes the MIG canvas
Zoom Out	• Zooms the model out • Toolbar shortcut: **Zoom Out** • Keyboard shortcut: **CTRL + -**
Zoom In	• Zooms the model in • Toolbar shortcut: **Zoom In** • Keyboard shortcut: **CTRL + +**
Zoom 100%	• Returns the zoom to the standard 100% setting • Toolbar shortcut: **Zoom 100%** • Keyboard shortcut: **CTRL + 1**
Drag to Select	• A rectangular selection box will appear upon a left click and drag on the model canvas
Drag to Scroll	• A click and drag by the user will pan around the model canvas

5.1.4 Model Menu

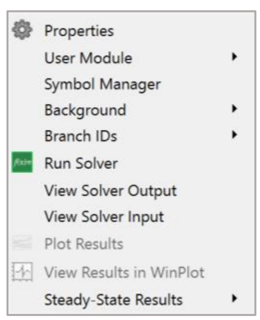


Figure: Model dropdown menu in MIG

The model dropdown menu contains important options for building, running, and viewing models. In this menu, the user can access the model properties window, manage symbols, branch IDs, and the model background. The user can run the built-in GFSSP solver and view the input and output files. There are two plotting options to plot unsteady results. MIG offers a built-in plotting feature but supports viewing results in WinPlot.

Properties	• Opens the model properties window where the user can edit various model properties as detailed in section 5.2 • Toolbar shortcut: **Properties**
User Module	• When hovered, allows the user to edit or compile user subroutines (detailed in sections 4.3 and 5.2.1)

<table>
<tr><td></td><td>Toolbar shortcuts: Edit User Subroutines and Compile User Module</td></tr>
<tr><td>Symbol Manager</td><td>Opens the symbol manager window where the user can add, edit, or delete symbols
Symbols are user defined global variables that can be assigned to quantities within the model components
When the value of the symbol is changed in the symbol manager, the values in all model components using the symbol will update</td></tr>
<tr><td>Background</td><td>The user may set a background image, adjust the background, remove the background, and set a background color by hovering the cursor over this option and selecting from the dropdown
Selecting Set Background Image will open the user’s device file explorer where the user can open an image from their device
Selecting Adjust Background allows the user to adjust the opacity and size of the background image using sliders. If there is no background image, MIG will prompt the user to select one from the device

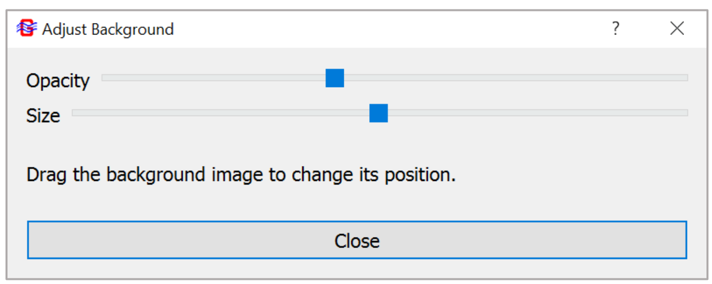


Figure: Background adjustment window

Selecting Remove Background will delete the background image set by the user
Selecting Set Background Color will open the Select Color window, allowing the user to set a background color</td></tr>
</table>

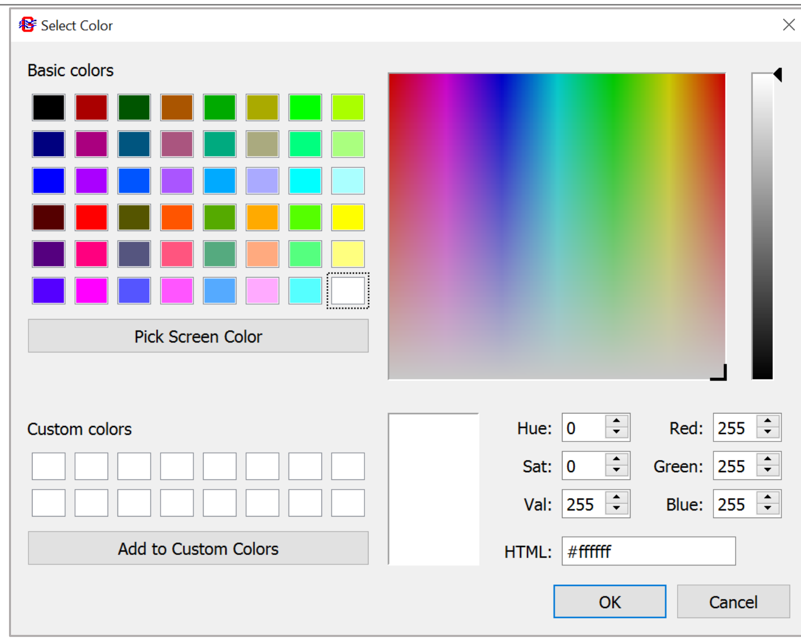


Figure: Background color selection window

Branch IDs	• When hovered, the user can enable or disable All Auto Position of branch IDs • After enabling All Auto Position and refreshing the model, MIG will find the optimal position for all branch IDs in the model
Run Solver	• Creates or updates the GFSSP input file • Runs the GFSSP solver to solve the governing equations • If the solver input file already exists, MIG will ask the user if they want to override the existing file • The GFSSP Run Manager (detailed in 5.5) opens • Toolbar shortcut: **Run GFSSP Solver**
View Solver Output	• Opens the files in MIG's text viewer from which the user may click Open in External Editor to open the output file in their preferred text editor
View Solver Input	• Opens the solver input file in the text editor selected in Application Preferences
Plot Results	• Opens the built-in MIG plotting tool to plot the results of an unsteady analysis • Toolbar shortcut: **Plot Results for Unsteady Model** • The MIG plotting tool is detailed in 5.5
View Results in WinPlot	• Opens WinPlot for plotting unsteady model results • WinPlot must be installed separately from GFSSP • Toolbar shortcut: **Launch WinPlot to Plot Unsteady Results**

Steady-State Results	• Enable, disable, and edit preferences for displaying steady-state results on the MIG model • Steady-state results display is detailed in 5.5.3

5.1.5 Advanced Menu

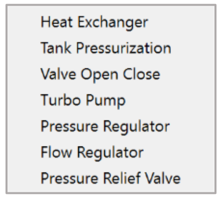


Figure: Advanced options dropdown menu

The advanced menu dropdown allows users to access the various advanced option settings in GFSSP. Each individual advanced option must be turned on in model properties before its settings can be accessed from the advanced menu. Each advanced option is detailed in 5.4.

5.1.6 Help Menu

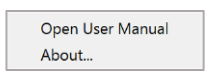


Figure: Help dropdown menu

In the help dropdown menu, the user may open the GFSSP user manual and click the about button to find MIG and GFSSP version information.

5.2 Model Properties

The Model Properties window contains six tabs. In most cases, the user will select the program inputs by working through the tabs from left to right.

5.2.1 General

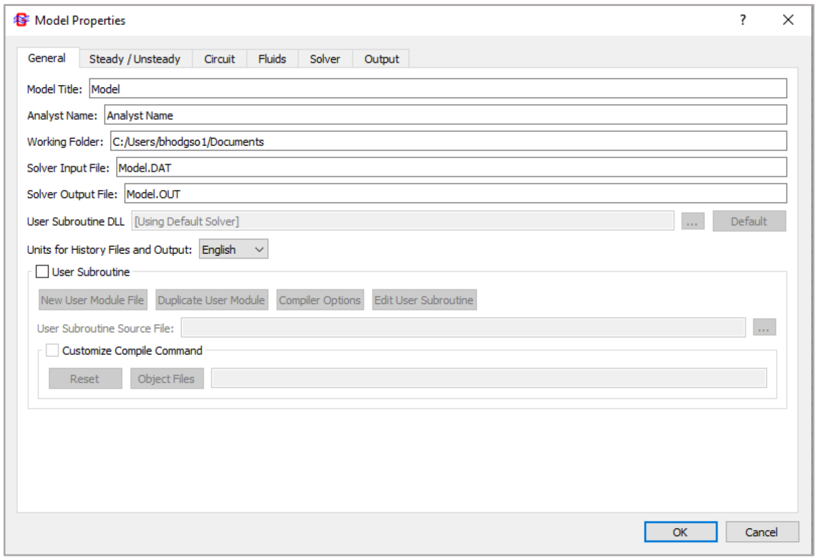


Figure: General model properties tab in the Model Properties window

Model Title	• Sets the name of the model • Can be seen in the upper left corner of the model canvas and is shown in the input and output files
Analyst Name	• Sets the name of the user performing the analysis • Shown in the input and output files • Will be pre-filled if the Analyst Name has been set in File/Preferences (see Section 5.1.1)
Working Folder	• Displays the folder containing the model files • Will only be displayed if a model is currently open • Cannot be edited. To change the working folder, move the *.gfssp file to another folder.
Solver Input and Output Files	• Displays the name of the GFSSP solver input and output files contained in the working folder • The user may edit the name of the files and click OK to save the change • Upon a name change, a new file will be created under the new name; old files will remain unchanged in the working folder • By convention, input files use the *.dat extension; output files use the *.out extension
User Subroutine DLL	• Allows the user to rename or select (the ellipse button) the user subroutine *.dll file • The Default option will use the default GFSSP solver without user subroutines upon running the simulation • The User option will use the current user subroutine *.dll to run the GFSSP solver with user modules

Custom Solver	• This option is only displayed if the Enable Custom Solver option is enabled in Application Preferences • Allows the user to specify the custom solver path
Units for History Files and Output	• Sets the default units used in history files and the output file
User Subroutine	• Enables user subroutines (detailed in section 4.3)

Options in Model Properties

- The New User Module File button creates a user subroutine file with Fortran template code
- The user subroutine file appears in the User Subroutine Source File text box
- Duplicate User Module copies the user subroutine source file under a new file name
- Compiler Options allow the user to select the compiler, the preferred Fortran format, and edit the compiler path
- Edit User Subroutine will open the built-in Fortran user module editor
- Right clicking on the toolbar in the main MIG window and selecting User Module will allow the user to open the editor by clicking Edit User Subroutines 
- Customize Compile Command allows the user to make manual edits to the compile command

User Module Editor

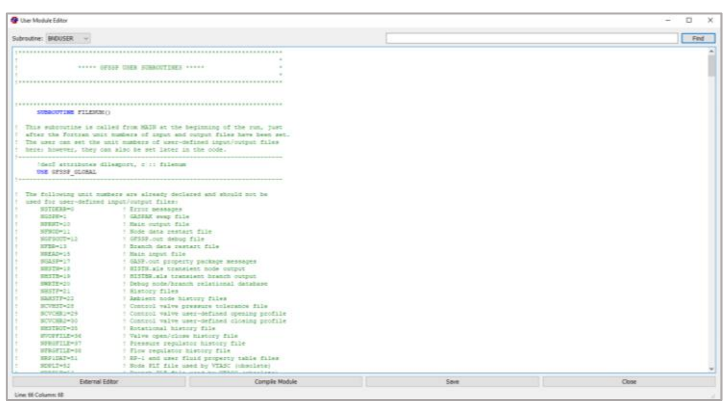


Figure: User Module Editor window in MIG

- The Subroutine dropdown menu allows the user to navigate to any of the subroutines detailed in section 4.3

	• The Find button will search the subroutine module for the exact text entered in the adjacent textbox • External Editor will open the user module in the text editor selected in Application Preferences • The Compile Module button in the editor or the Compile User Module button on the toolbar will compile the user subroutine module • The Save button will save the user subroutine source file • The Close button will exit the User Module Editor

5.2.2 Steady / Unsteady

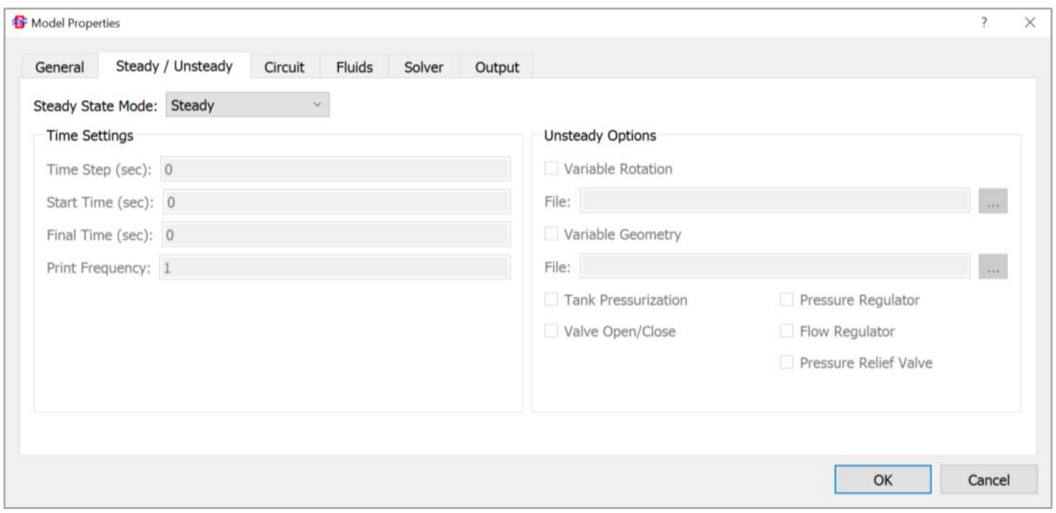


Figure: Steady/unsteady properties tab in the Model Properties window

Steady State Mode	• Select whether the model is steady state, quasi-steady, or unsteady • Quasi-steady models have boundary conditions that change with time, but a steady-state solution is solved in each time step. • All other options on this tab are for quasi-steady or unsteady models
Time Step (sec)	• Sets the time step of the unsteady model
Start Time (sec)	• Sets the initial time of an unsteady model • Does not have to be zero
Final Time (sec)	• Sets the ending time of an unsteady model
Print Frequency	• Sets the frequency at which the solver outputs branch and node solutions over the run time interval • The default value of 1 means the solver will output the branch and node solutions at each time step from the start time until the final time

	• Other values will result in the solver waiting the print frequency number of time steps between printing the branch and node solutions
Unsteady Options	• If applicable, the user may select the appropriate option from variable rotation, variable geometry, tank pressurization, valve open/close, pressure regulator, flow regulator, and pressure relief valve • Variable rotation and variable geometry can be controlled through a user subroutine or a data file • See Section 5.4 for discussion of the tank pressurization, valve open/close, pressure regulator, flow regulator, and pressure relief valve options

5.2.3 Circuit

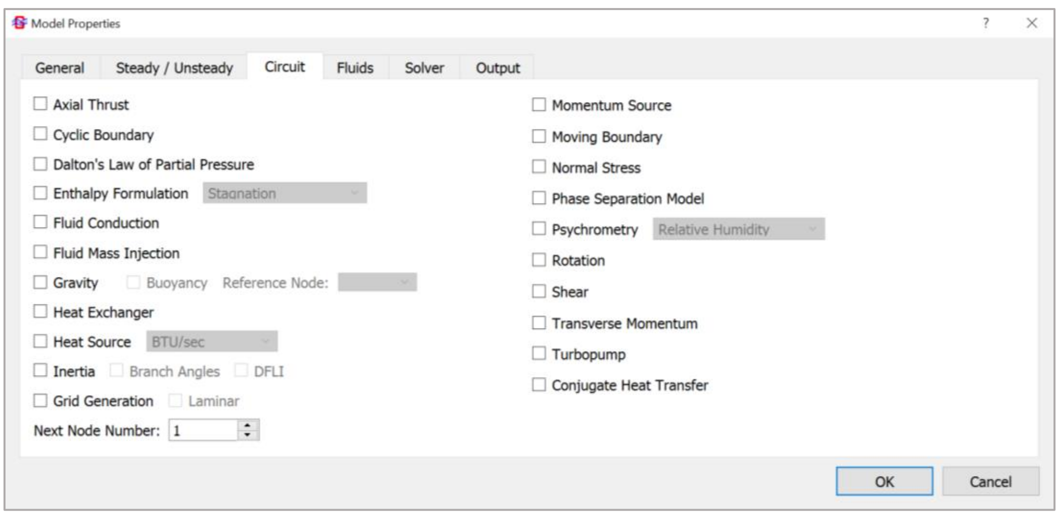


Figure: Circuit options tab in the Model Properties window

Axial Thrust	• Apply thrust in an area specified in the properties of any node
Cyclic Boundary	• User must select this option if creating a closed-loop circuit in a steady-state model (see Section 6.20) • User must designate an upstream node in the properties for the boundary node • The code will iterate on the temperature of the boundary node until it equals the temperature of the designated upstream node.
Dalton’s Law of Partial Pressure	• Apply Dalton’s law to evaluate properties at their partial pressures • Useful for gas mixtures when one or more of the gases would be a liquid if its properties were evaluated at the total mixture pressure

	• The psychrometry option requires Dalton's law to be used • GFSSP uses Amagat's law of partial volumes by default
Enthalpy Formulation	• Allows the user to specify stagnation (default) or static enthalpy in the energy equation
Fluid Mass Injection	• Allows the user to add a mass source that is specified in the properties of any node
Gravity	• Activates the gravity term in the momentum conservation equation (see 3.1.2) • User may indicate whether the model considers buoyancy (density change) effects in a gravity field • When the buoyancy option is selected, the user must select a reference node for density
Heat Exchanger	• Allows the user to model heat exchange between two branches • User must supply heat exchanger data by selecting Heat Exchanger from the advanced menu (see Section 5.4)
Heat Source	• Allows the user to add a heat source that is specified in the properties of any node • User may select the heat source units
Inertia	• Activates the inertia term in the momentum conservation equation (see 3.1.2) • User must select the inertia checkbox in each applicable branch • User may select Branch Angles to specify angles of branches for transverse inertia in multidimensional flow • User may choose to activate Differential Formulation for Longitudinal Inertia
Grid Generation	• Enables the user to generate multidimensional grids • Selecting the Laminar checkbox will require the flow to be laminar and shear stress to be calculated from the laminar viscosity. Otherwise, the flow is assumed to be turbulent
Next Node Number	• Sets the identifier number of the next node placed in the model
Momentum Source	• Allows user to apply a generic momentum source to a branch • User must select the momentum source checkbox in each applicable branch • The user must supply momentum source data in branch options
Moving Boundary	• Allows user to apply a force from a moving boundary (see 3.1.2) • User must select the Moving Boundary checkbox in each applicable node
Normal Stress	• Activates the normal stress term in the momentum conservation equation (see 3.1.2)
Phase Separation Model	• Activates the phase separation model

	• In phase separation model nodes, only the vapor phase is allowed to exit the node until the node is completely filled with liquid • User must select the Phase Separation model checkbox in applicable nodes
Psychrometry	• Activates psychrometric analysis • Psychrometric analysis requires Dalton's law of partial pressure to be used • The user must select air and water as the model fluids • The user must select one of the three input control parameters: Relative Humidity, Wetbulb Temperature, or Humidity Ratio
Rotation	• Activates the centrifugal term in the momentum conservation equation (see 3.1.2) • User must select rotation checkbox in applicable branches • User must supply rotation data to each branch with rotation
Shear	• Activates the shear term in the momentum conservation equation for all branches (see 3.1.2)
Transient Term Active	• Activates unsteady term in the momentum equation • Useful for fluid transient problems (e.g. waterhammer). Otherwise recommended to not activate the transient term
Transverse Momentum	• Activates the transverse inertia term in the momentum equation for all branches
Turbopump	• Allows the user to connect two branches as a pump and turbine (see Section 6.11) • User must supply turbopump data by selecting Turbopump from the advanced menu (see Section 5.4)
Conjugate Heat Transfer	• Activates conjugate heat transfer and related circuit options such as solid nodes, ambient nodes, and conductors • If unchecked, Conjugate Heat Transfer is removed from the model calculations, but solid nodes and conductors will still be visible on the model canvas

5.2.4 Fluids

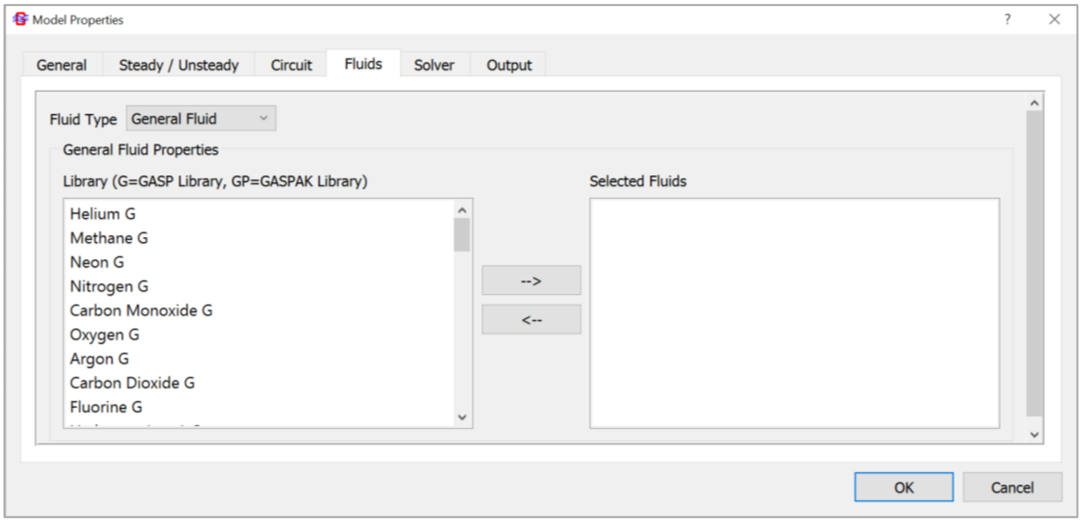


Figure: Fluid properties tab in the Model Properties window

Fluid Type

- The user can choose between using fluids from the built-in libraries (general fluid) or a constant property fluid
- General fluids may be transferred to or from the selected model fluids by selecting the fluid and using the arrow buttons
- Constant property fluids cannot be used in unsteady models

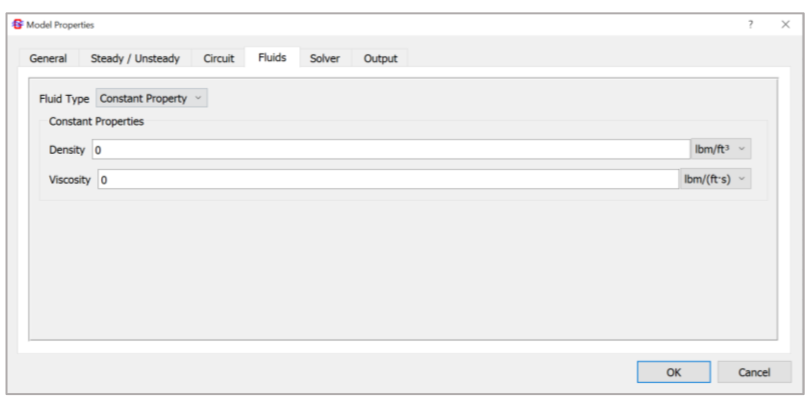


Figure: Constant property fluid tab in the Model Properties window

- The user must enter the density and viscosity of the fluid when using the constant property option

Fluid Library

- A list of all fluids available to the user

Built-in Fluids

- Contains fluids from the built-in GASP (identified by G) and GASPAK (identified by GP) libraries
- RP-1 (from tables), ideal gas, and user fluids are the other available fluid options
- The ideal gas fluid allows the user to enter various properties of an ideal gas, defaulting to the properties of air at room temperature

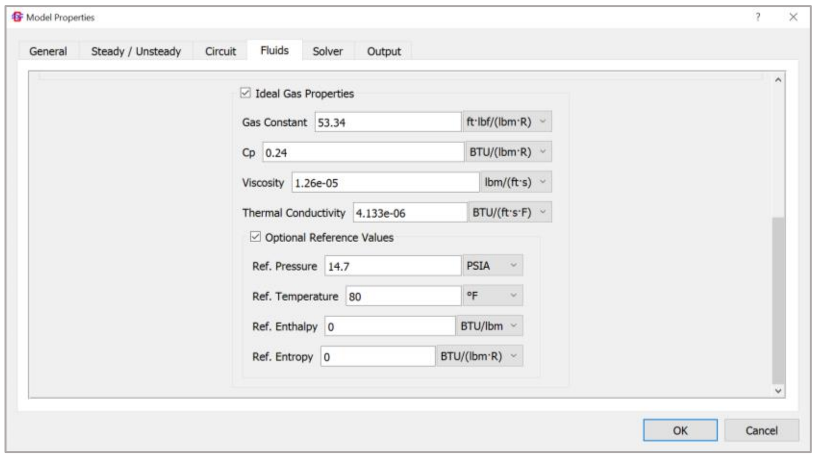


Figure: Ideal gas fluid properties editor in the Model Properties window

User Fluids

- There may be up to three user fluids in a model
- The user must provide property tables for the fluid
- Required properties are thermal conductivity, density, viscosity, specific heat ratio, enthalpy, entropy, specific heat, and fluid molecular weight
- An optional table may be added to support saturation and phase change properties
- A program with instructions for converting Refprop output into user fluid tables can be found in the GFSSP installation directory

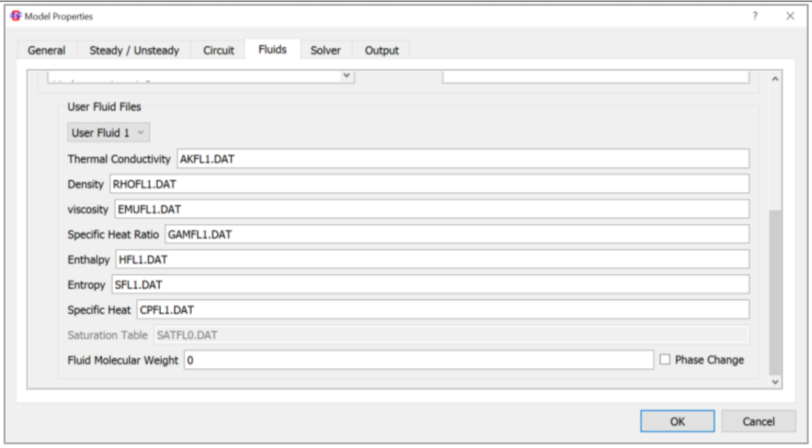


Figure: Required user fluid property table files

Selected Fluids	• A list of all fluids currently selected for use in the model

5.2.5 Solver

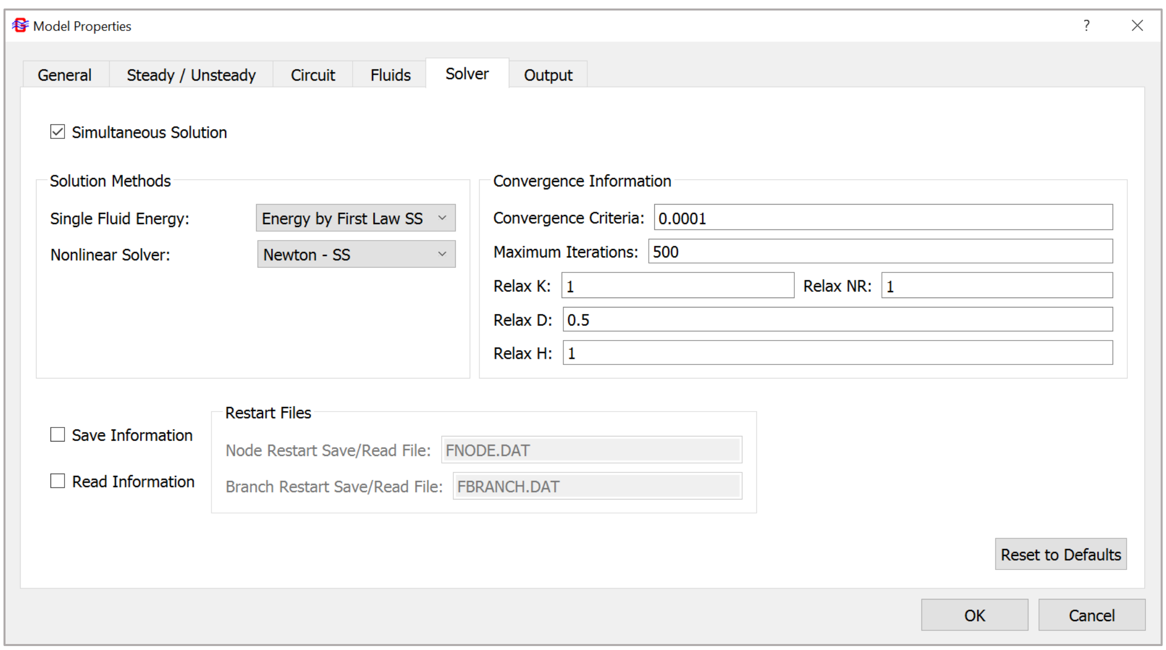


Figure: Solver properties tab

Simultaneous Solution	• The user may choose between a simultaneous solution scheme and a nonsimultaneous solution scheme • These solution schemes are detailed in 4.2.1 and 4.2.2
Single Fluid Energy	• The user may choose between the first law and second law of thermodynamics solution procedures for the energy equation

	• The first law solution can utilize successive substitution or Newton-Raphson methods • The energy equations are detailed in 3.1.3
Fluid Mixture Energy	• The user may choose how the energy equation is expressed for fluid mixtures • The energy equation can be expressed in temperature (Temperature) or mixture enthalpy (Enthalpy 1), or it can be solved for individual species (Enthalpy 2) • Unlike Temperature, the Enthalpy 1 and Enthalpy 2 options model phase change
Energy For Solid	• The user may choose between Newton-Raphson and Successive Substitution methods for solving the solid node energy equation
Differencing Scheme	• The user may choose between first order and second order differencing to evaluate time derivatives in conservation equations
Nonlinear Solver	• The user may select a nonlinear solver from Newton-Raphson with Successive Substitution, Broyden with Successive Substitution, or Newton-Raphson with Successive Substitution using a sparse matrix solver • Due to the sparse matrix solver, the Newton – SS (Sparse) option may run faster for large models • Due to no update of the Jacobian matrix in every iteration, the Broyden – SS method may be faster for long transient models
Convergence Criteria	• Sets the criteria for convergence • Based on the difference in variable values between successive iterations • Iteration is complete when the maximum relative change between iterations in solved-for variables such as pressure, flow rate, enthalpy, and resident mass is less than the convergence criterion • Default is 1.0×10^{-4}
Maximum Iterations	• Sets the maximum number of iterations that can be performed • Default is 500
Relax K	• Under-relaxes the change in resistance factor K_f • Used in the friction term of the momentum equation • Useful if a model has large swings in K_f • Lower values provide more relaxation • Default is 1.0
Relax NR	• Under-relaxes the Newton-Raphson solver • Used for the mass and momentum equations that are solving for pressures and flow rates • Lower values provide more relaxation

	• Default is 1.0
Relax D	• Under-relaxes the change in fluid density between iterations • Lower values provide more relaxation • Default is 0.5
Relax H	• Inertially relaxes the change in enthalpy between iterations, or under-relaxes the change in entropy between iterations if using the second law • Higher values provide more weight to the enthalpy from the previous iteration when using the first law • If using the second law, the value of Relax H applies a percentage of the correction (e.g 0.6 applies 60% of the correction) • Default is 1.0
Relax HC	• Under-relaxes the change in calculated convection coefficient between iterations • Lower values provide more relaxation • Default is 1.0
Relax TS	• Under-relaxes the change in solid temperature between iterations • Lower values provide more relaxation • Default is 1.0
Save Information	• Saves node and branch steady-state solutions to their respective restart files • User may specify node and branch save file names
Read Information	• The solver reads the information in the restart files and sets initial properties of nodes and branches to the values in the restart files

5.2.6 Output

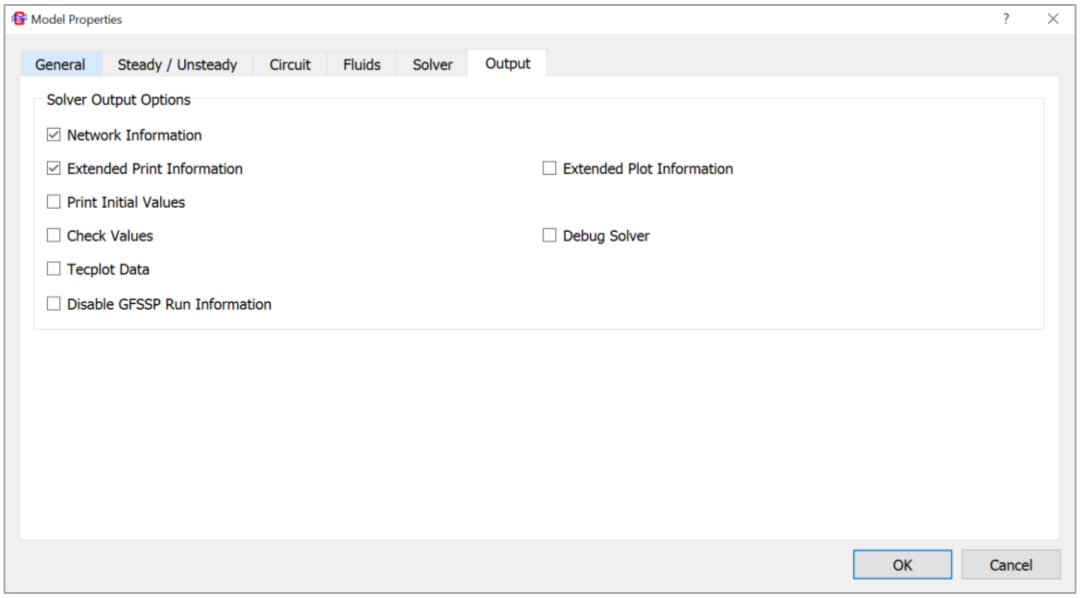


Figure: Output tab in the model properties window

Network Information	• Prints fluid network information in the output file • Includes internal node thrust area, mass source, and heat source designated by the user • Includes branch flow designation and resistance option information
Extended Print Information	• Includes the values of the enthalpy, entropy, viscosity, thermal conductivity, specific heat capacity, and heat capacity ratio of interior nodes in the output file
Print Initial Values	• Prints the initial guess for internal nodes and the trial solution for branches in the output file
Check Values	• Checks for unreasonably large or small values of pressure, temperature, and mass flow rate • Prints warnings of unreasonable values to the run manager
WinPlot Data	• Will create a WinPlot data file for unsteady models • The user may choose between a binary file and a CSV file for the data • Winplot binary files may be opened and plotted while the model is still running. • Winplot CSV files can only be opened when the model has completed. • Plot Frequency sets the timestep plot frequency (e.g a plot frequency of 2 plots every other timestep in WinPlot) • Binary Write Frequency sets the frequency at which timesteps are compressed and written to the binary file. Values of 20 or

	greater are recommended for improved speed and compression efficiency
Plot User Specified Values	• Enables user defined plot variables • Number User Variables specifies the number of user plot variables defined • User plot variables must me defined in the appropriate user subroutine
Tecplot Data	• Creates a Tecplot data file for creating contour plots of multidimensional flows
Disable GFSSP Run Information	• Disables run information in the Run Manager
Extended Plot Information	• Provides the values of the enthalpy, entropy, viscosity, thermal conductivity, specific heat capacity, and heat capacity ratio of interior nodes to the WinPlot data file
Debug Solver	• Creates a file in the working folder showing all iterations of the Newton-Raphson solver • This file can become very large for unsteady models, so it is recommended to use a low number of timesteps

5.3 Building Networks

This section describes the construction of the fluid network on the canvas, using the toolbar on the left side of MIG.

Select Item	• The user may select individual circuit elements by left clicking the element • The user may select multiple circuit elements by left clicking and dragging the selection rectangle over all elements to be selected • The user may select this tool by clicking the **Select Item** icon on the toolbar or by pressing **Esc** on the keyboard • To avoid accidentally dragging objects with the mouse, use the **Lock Workspace** option on the toolbar
Boundary Node	• The user may add a boundary node by clicking the **Add New Boundary Node** icon on the toolbar or by pressing (**1**) on the keyboard • Once this tool is selected, the user may left click anywhere in the model building space to place a boundary node **General Boundary Node Properties**

- The user may enter the properties of the boundary node by double-clicking the node with the left mouse button or by right clicking the node and selecting Properties
- The user may specify the node identifier number, which must be an integer
- The user may give the node a description and select the Show checkbox to show the description on the main model screen
- The user may open the symbol manager by clicking the Symbol Manager button

Steady-state Boundary Nodes

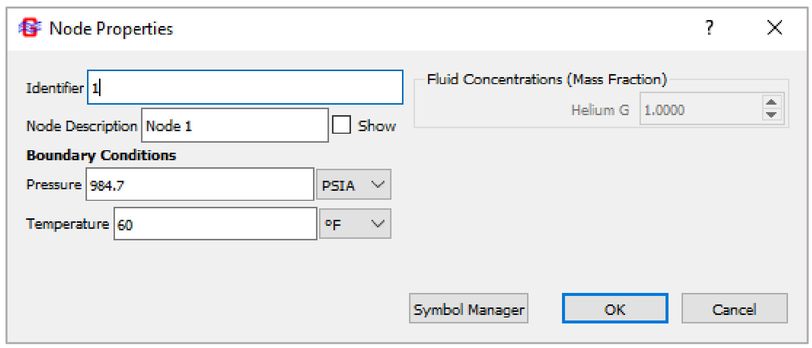


Figure: Boundary node properties window for steady-state models

- The user must specify the pressure and temperature boundary conditions of the node
- If there are multiple fluids selected, the user may specify the fluid concentrations at the node
- The user may toggle steady-state results display by right clicking the node and selecting Enable Results Display or Disable Results Display. This may also be done in the Model dropdown menu
- The user may change the steady-state results position around the node by right clicking the node and using the Results Position selector

Unsteady Boundary Nodes

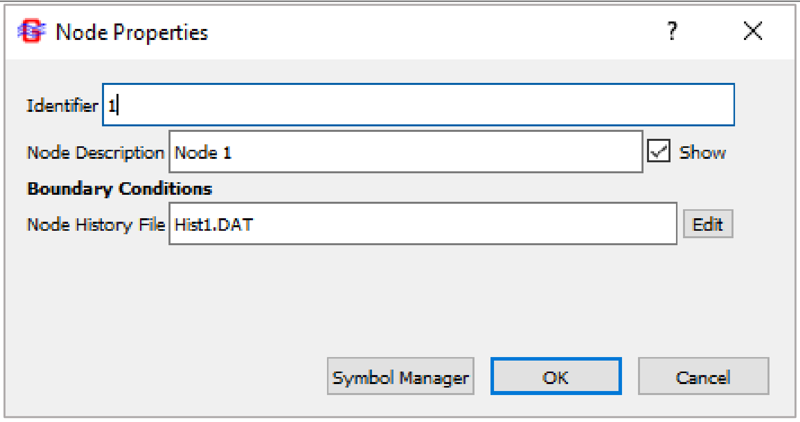


Figure: Boundary node properties window for unsteady models

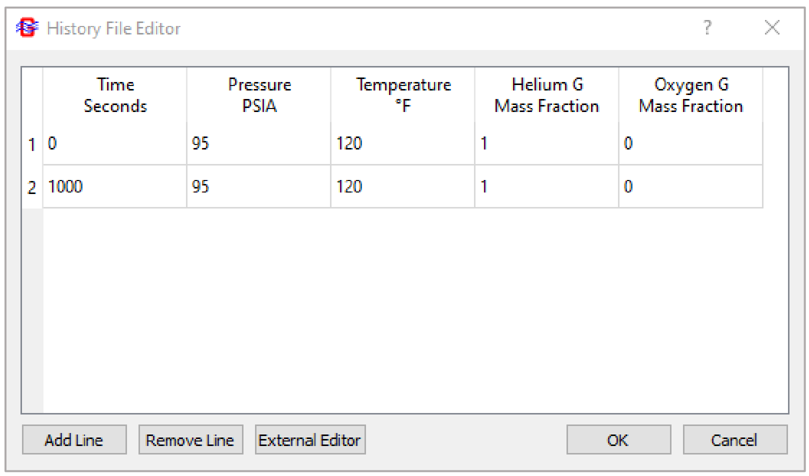


Figure: Example 10 boundary node history file

- The user must supply a history file for the boundary conditions by clicking the Edit button next to Node History File
- The user may edit the name of the node history file in the Node History File textbox
- The user must supply the pressure, temperature, and fluid concentrations at a minimum of two times
- The Add Line button adds another row to the history file while the Remove Line removes the selected row
- Pressing the Enter key when the cursor is in the right-most column will create a new row with the same properties as the previous row
- The user may choose to edit the history file in an external editor by clicking the External Editor button

Other Options

- The user may set the node icon to a custom image by right clicking on the node and selecting Set Custom Image
- The node may be moved into a SuperNode by right clicking the node and selecting the SuperNode or New SuperNode from the Move Items to SuperNode selector

Interior Node

- The user may add an interior node by clicking the **Add New Interior Node** ☐ icon on the toolbar or by pressing (**2**) on the keyboard
- Once this tool is selected, the user may left click anywhere in the model building space to place an interior node

General Interior Node Properties

- The user may enter the properties of the node by double-clicking the left mouse button or by right clicking the node and selecting Properties
- The user may specify the node identifier number, which must be an integer
- The user may give the node a description and select the Show checkbox to show the description on the main model screen
- The user may open the symbol manager by clicking the Symbol Manager button
- The properties of an interior node may be copied and pasted to other interior nodes by right clicking the node icon and selecting Copy Node Properties or Paste Node Properties. The properties may be copied to multiple interior nodes by selecting more than one node (by holding down CTRL or dragging a selection box around a group of nodes) before selecting Paste Fluid Node Properties

Steady-State Interior Nodes

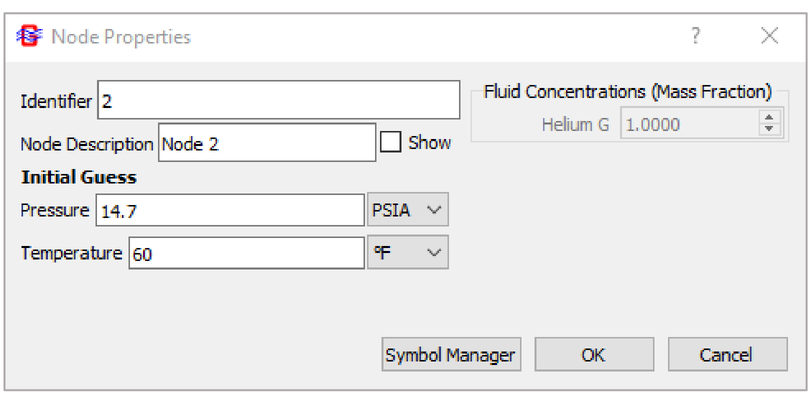


Figure: Interior node properties window for steady-state models

- The user may specify different initial guesses for the pressure and temperature of the node

- If there are multiple fluids selected, the user may specify the fluid concentrations at the node
- The user may enable steady-state results display by right clicking the node and selecting Enable Results Display or Disable Results Display. This may also be done in the Model dropdown menu
- The user may change the steady-state results position around the node by right clicking the node and using the Results Position selector

Unsteady Interior Nodes

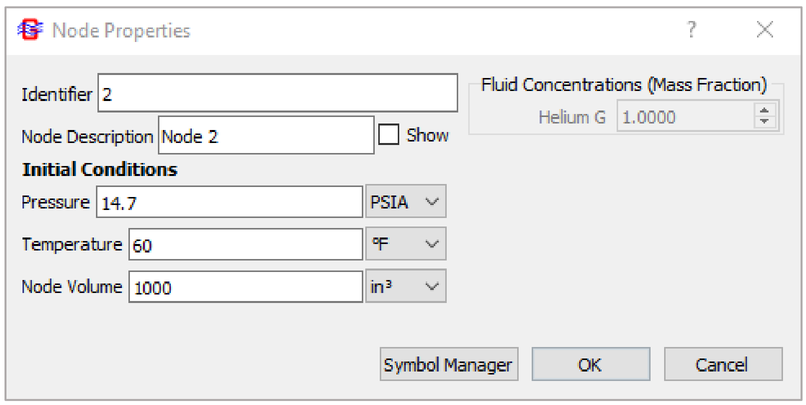


Figure: Interior node properties window for unsteady models

- The user must specify the initial pressure, temperature, and volume of the node
- The initial conditions can be overwritten from the restart file by selecting Read Information in the Solver tab of the Model Properties window
- Node volumes are optional. Nodes which are negligible compared to the overall system may be assigned a volume of zero. These nodes will react instantly to changes in the flow entering the node (zero capacitance).
- If the interior node is connected to a pipe branch, GFSSP will automatically assign half the pipe volume to that node, so the node volume input is usually zero. If a non-zero value is entered for the volume of a node connected to a pipe branch, this volume will be added to the volume from half of the pipe. MIG will issue a run-time warning to confirm that this is your intention

Other Options

- The user may set the node icon to a custom image by right clicking on the node and selecting Set Custom Image

	• The node may be moved into a SuperNode by right clicking the node and selecting the SuperNode or New SuperNode from the Move Items to SuperNode selector
Branch	• The user may add a branch between two nodes by clicking the **Add New Branch** icon on the toolbar or by pressing (**3**) on the keyboard • Once this tool is selected, the node borders will display small green dots. First left click the upstream node and then left click the downstream node; it is not necessary to click directly on the green dots • The branch restriction type must be selected by right clicking the branch icon ? and using the Set Branch Type dropdown menu • Once the branch type is selected, the Branch Properties can be opened by double left-clicking the branch icon or by right clicking the branch icon and selecting Properties • All the branch types and their properties are detailed in section 3.1.7 • The properties of a branch may be copied and pasted to a branch of equivalent type by right clicking the branch icon and selecting Copy Branch Properties or Paste Branch Properties. The properties may be copied to multiple branches by selecting more than one branch (by holding down CTRL or dragging a selection box around a group of branches) before selecting Paste Branch Properties • The branch ID position can be changed by right clicking the branch icon and using the ID Position dropdown menu • The branch icon can be set to a custom image by right clicking on the branch icon and selecting Set Custom Image • Steady-state results display may be toggled by right clicking the branch icon and selecting Enable Results Display or Disable Results Display. This may also be done in the Model dropdown menu • The steady-state results position can be changed by right clicking the branch and using the Results Position dropdown menu • The nodes upstream and downstream of the branch may be reassigned by right clicking the branch icon and using the Upstream Node or Downstream Node dropdown menu
Build Network	• After selecting the **Build Network** option, use this mode to construct “straight-line” models rapidly • Each left click on the modeling screen with this tool inserts an interior node connected by a branch to the selected or previously placed node • Hold down the SHIFT key while left-clicking to place a boundary node instead of an internal node. • Press ESC to exit this mode
SuperNode	• The user may add a SuperNode by clicking the **Add New SuperNode** icon on the toolbar or by pressing (**5**) on the keyboard

	• Once this tool is selected, the user may left click anywhere in the model building space to place a SuperNode • A SuperNode is an organizational tool that may contain parts (nodes and branches) of the greater fluid circuit • The user must connect upstream and downstream nodes to the SuperNode through branches • The user must select the appropriate upstream/downstream node inside the SuperNode for each branch connected to the SuperNode • To view the contents of a SuperNode either double left click the node or right click and select Show SuperNode Objects • Circuits may be constructed within the SuperNode just as they would outside of the node • To return to the main model screen, click the Show Main Model icon on the toolbar, the equivalent option in the edit menu, or press the Home button on the keyboard • The properties of the SuperNode can be accessed by right clicking the SuperNode and selecting Properties • The SuperNode may be given a description that can be shown on the main model screen by selecting the Show checkbox in the SuperNode properties window • The node icon may be set to a custom image by right clicking on the node and selecting Set Custom Image • A SuperNode may be moved into another SuperNode by right clicking the node and selecting the SuperNode ID or New SuperNode from the Move Items to SuperNode dropdown menu
Grid	• The user may add a grid by clicking the **Add New Grid** icon on the toolbar or by pressing (**6**) on the keyboard • Once this tool is selected, the user may left click anywhere in the model building space to place a grid • The grid displayed on the main model is a placeholder for the actual contents of the grid **Grid Properties** • The user may enter the properties of the grid by double-clicking the left mouse button or by right clicking the node and selecting Properties • NOTE: If the grid properties have already been set, double-clicking the left mouse button will open the grid model

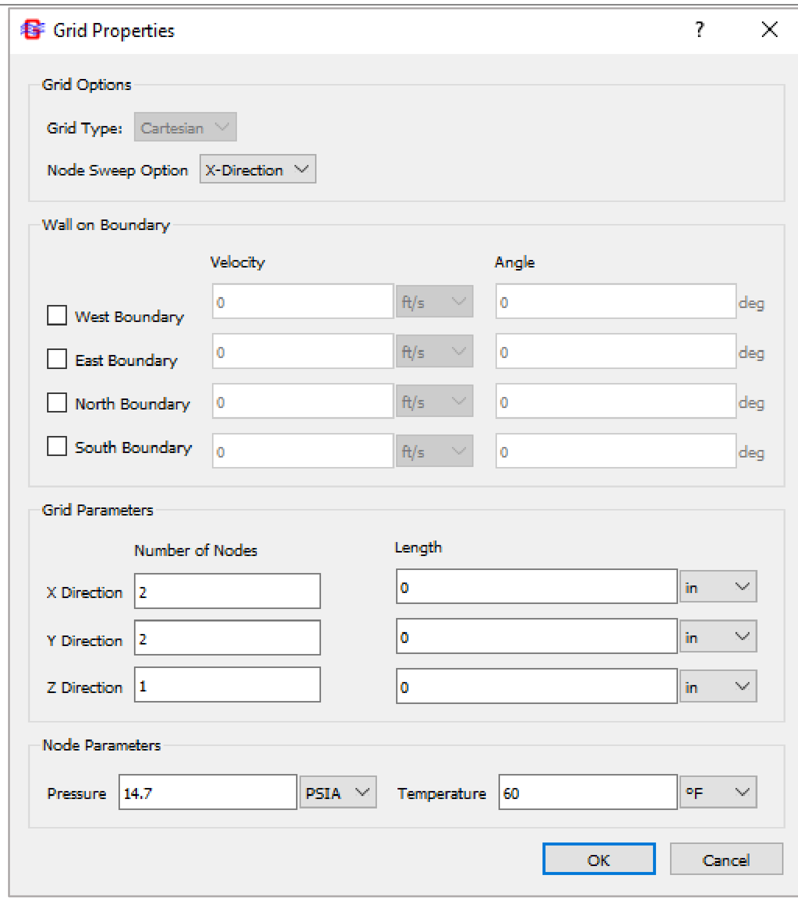


Figure: Grid properties window

- Grids may be generated with Cartesian (x,y) coordinates
- The grid type, node sweep, wall boundary information, dimensions, number of nodes in each dimension, default pressure, and default temperature must be specified by the user
- The Node Sweep Option specifies the initial direction of increasing node number
- For all walls, the velocity at the boundary, and the angle of the wall with the coordinate direction at the boundary must be given
- The Number of Nodes option specifies the number of nodes to distribute over the total given length of the corresponding direction
- To connect a grid to the rest of the network, right click a branch connected to the grid and use the Upstream Node or Downstream Node dropdown menus to select the connecting node in the grid

Text	• The user may add text to the model by selecting the **Add Text** T tool on the toolbar or by pressing (**T**) on the keyboard

- Once this tool is selected, the user may left click anywhere in the model building space to place text

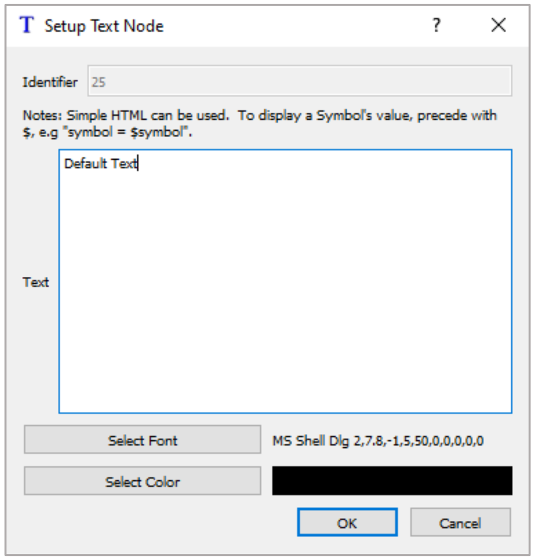


Figure: Text node setup window

- When the Setup Text Node window appears, the user may enter the desired text
- Simple HTML may be used in the text
- To display a Symbol's value, begin the symbol with a $ (e.g "symbol = $symbol")
- The font and color of the text may be edited through the respective buttons

Solid Node

- The Conjugate Heat Transfer option must be enabled in the Circuit tab of the Model Properties to use solid nodes
- The user may add a solid node by clicking the **Add New Solid Node** icon on the toolbar or by pressing (**Q**) on the keyboard
- Once this tool is selected, the user may left click anywhere in the model building space to place a solid node

Interior Node Properties

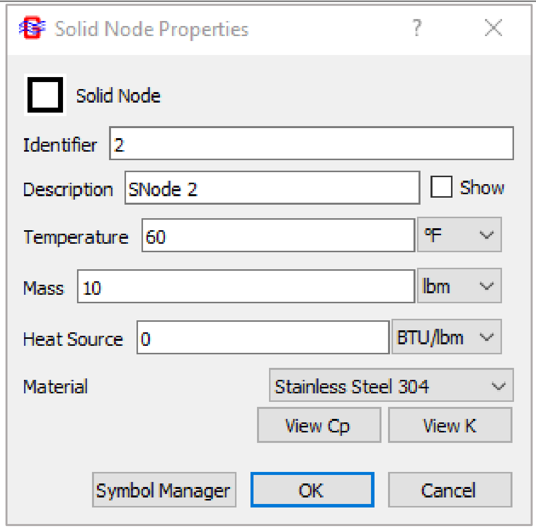


Figure: Solid node properties window

- The user may enter the properties of the node by double left-clicking the node or by right clicking the node and selecting Properties
- The user may specify the node identifier number, which must be an integer
- The user may give the node a description and select the Show checkbox to show the description on the main model screen
- The temperature, mass, and material of the solid node must be specified
- An optional heat source may be specified
- The symbol manager may be opened by clicking the Symbol Manager button

Other Options

- The properties of a solid node may be copied and pasted to another solid node by right clicking the solid node icon and selecting Copy Solid Node Properties or Paste Solid Node Properties
- The user may set the node icon to a custom image by right clicking on the node and selecting Set Custom Image

Ambient Node

- The Conjugate Heat Transfer option must be enabled in the circuit tab of the model properties to use ambient nodes
- The user may add an ambient node by clicking the **Add New Ambient Node**  icon on the toolbar or by pressing (**W**) on the keyboard
- Once this tool is selected, the user may left click anywhere in the model building space to place a solid node

Ambient Node Properties

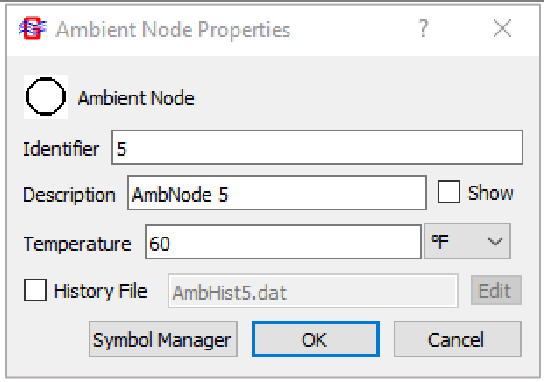


Figure: Ambient node properties window

- The user may enter the properties of the node by double left-clicking the node or by right clicking the node and selecting Properties
- The user may specify the node identifier number, which must be an integer
- The user may give the node a description and select the Show checkbox to show the description on the main model screen
- The ambient temperature must be specified
- For unsteady models, the user can use an optional history file to specify the ambient temperature over time

Other Options

- The user may set the node icon to a custom image by right clicking on the node and selecting Set Custom Image

Conductor

- The user may add a conductor between two nodes by clicking the **Add New Conductor** icon on the toolbar or by pressing (**E**) on the keyboard
- Once this this tool is selected, left click both nodes to be connected by the conductor
- The conductor type must be selected by right clicking the conductor icon and using the Set Conductor Type dropdown menu
- The Solid-Fluid Convection conductor type will be automatically selected when connecting a solid node and an interior node
- Once the conductor type is selected, the Conductor Properties can be opened by double left-clicking the conductor icon or by right clicking the conductor icon and selecting Properties
- The user must then enter the properties of the conductor
- The properties of a conductor may be copied and pasted to a conductor of equivalent type by right clicking the conductor icon and selecting Copy [Conductor Type] Properties or Paste [Conductor Type] Properties
- The conductor ID position can be changed by right clicking the conductor icon and using the ID Position dropdown menu

- The conductor description position can be set by right clicking on the conductor icon and using the Description Position dropdown menu
- The conductor icon can be set to a custom image by right clicking on the conductor icon and selecting Set Custom Image
- Steady-state results display may be toggled by right clicking the conductor icon and selecting Enable Results Display or Disable Results Display. This may also be done in the Model dropdown menu
- The steady-state results position can be changed by right clicking the branch and using the Results Position dropdown menu

Build Conjugate Heat Transfer Network

- After selecting the **Build Conjugate Heat Transfer Network** option or by pressing (**R**) on the keyboard, each left click on the modeling screen inserts a solid node connected by a conductor branch to the selected or previously placed node
- Hold down the SHIFT key while left-clicking to place an ambient node instead of a solid node
- Press ESC to exit this mode

5.4 Advanced Options

Heat Exchanger

- After the heat exchanger circuit option is enabled in Model Properties, the user may access the Heat Exchanger option from the Advanced dropdown menu
- Use the Add or Remove buttons to add or remove heat exchangers in the model
- NOTE: GFSSP supports a maximum of 20 heat exchangers in a single model

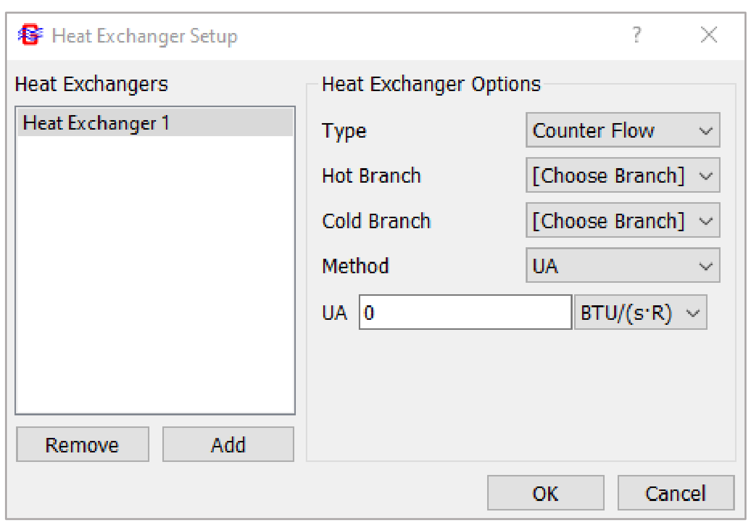


Figure: Heat exchanger advanced option properties window

- To edit the options of a heat exchanger, select the appropriate heat exchanger from the left menu
- The available heat exchanger options include the heat exchanger type, ID numbers of the hot branch and cold branch, and method
- Counter flow and parallel flow are the two available heat exchanger types available in the Type dropdown menu
- The available heat exchanger solution methods include UA and Effectiveness

Tank Pressurization

- After the Tank Pressurization option is selected in the Steady/Unsteady options in Model Properties, the user may access the Tank Pressurization option in the Advanced dropdown menu
- NOTE: GFSSP supports a maximum of 5 tanks in a single model

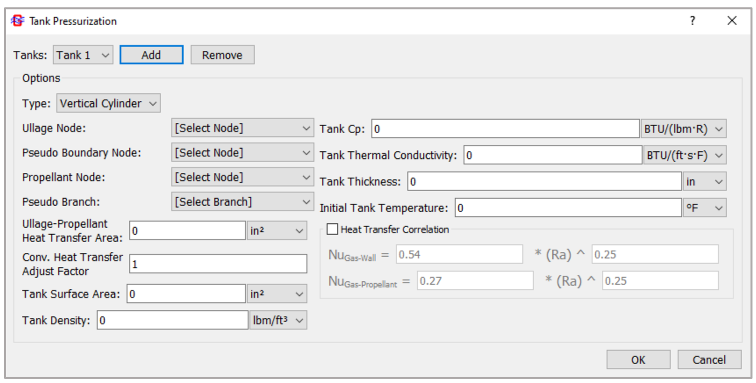
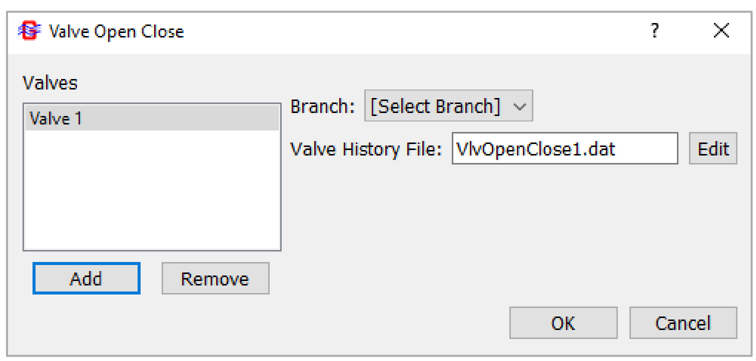


Figure: Tank pressurization advanced option properties window

- Use the Add or Remove buttons to add or remove tanks from the model
- The tank type, ullage node, pseudo boundary node, propellant node, pseudo branch, ullage-propellant heat transfer area, convection heat transfer adjustment factor, tank surface area, tank density, tank specific heat, tank thermal conductivity, tank thickness, initial tank temperature, and heat transfer correlation must be supplied
- Vertical cylinder and sphere are built in tank geometries, but there is an option to supply a user defined geometry
- The ID number of the internal node representing the tank ullage space must be selected using the Ullage Node dropdown
- The ID number of the pseudo boundary node representing the ullage pressure on the propellant surface must be selected using the Pseudo Boundary Node dropdown

	• The ID number of the pseudo branch representing the propellant surface must be selected using the Pseudo Branch dropdown • The ullage-propellant area for heat transfer to occur must be provided if the vertical cylinder is selected. If the sphere or user-defined tank is selected, the code will determine the ullage-propellant area based on the propellant depth. • The user may change the convection heat transfer adjustment factor if desired • The tank wall surface area initially exposed to ullage must be provided for the Tank Surface Area property if the vertical cylinder is selected. If the sphere or user-defined tank is selected, the code will determine the ullage-wall area based on the propellant depth. • The density, specific heat, thermal conductivity, thickness, and initial temperature of the tank material must be provided in the appropriate boxes • The user has the option to change the heat transfer correlation equations by selecting the Heat Transfer Correlation checkbox
Valve Open Close	• After the Valve Open/Close option is selected in Steady/Unsteady options in Model Properties, the user may access the Valve Open Close option in the Advanced dropdown menu • NOTE: GFSSP supports a maximum of 50 Valve Open/Close options in a single model  *Figure: Valve open close advanced option properties window*

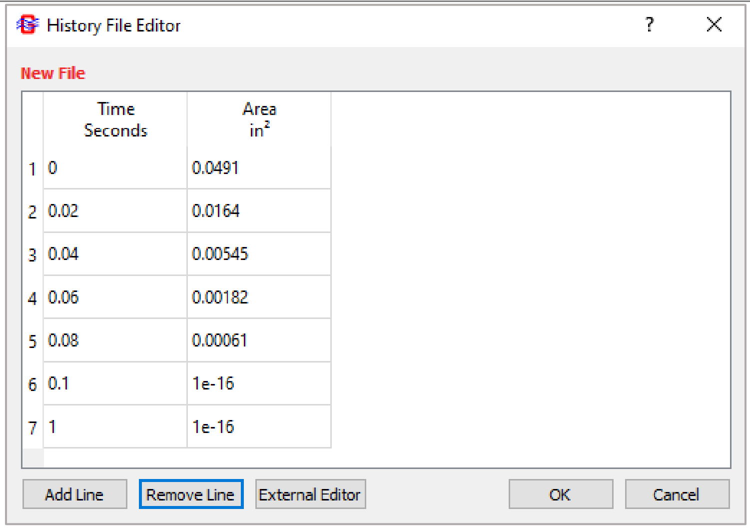


Figure: Tutorial 2 valve Open/Close history file

- Use the Add or Remove buttons to add or remove valves from the model
- The valve branch ID number must be specified using the Branch dropdown menu
- The user must supply a valve history file containing the open area of the valve over time. A closed valve should be given a small but non-zero area, such as 1×10^{-16} in^2.

Turbo Pump

- After the Turbopump option is selected in Circuit options in Model Properties, the user may access the Turbo Pump option in the Advanced dropdown menu
- NOTE: GFSSP supports a maximum of 10 turbopump assemblies in a single model

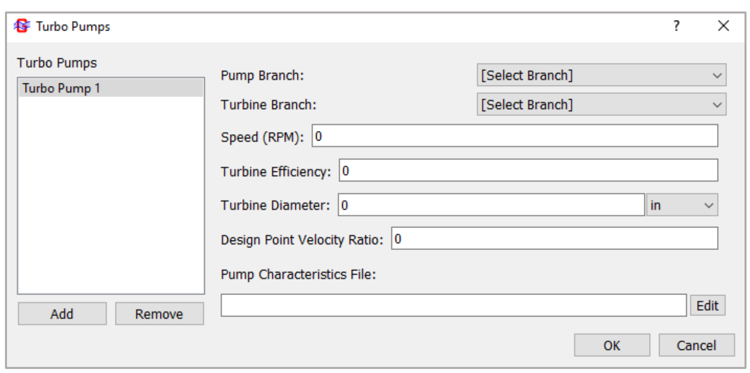
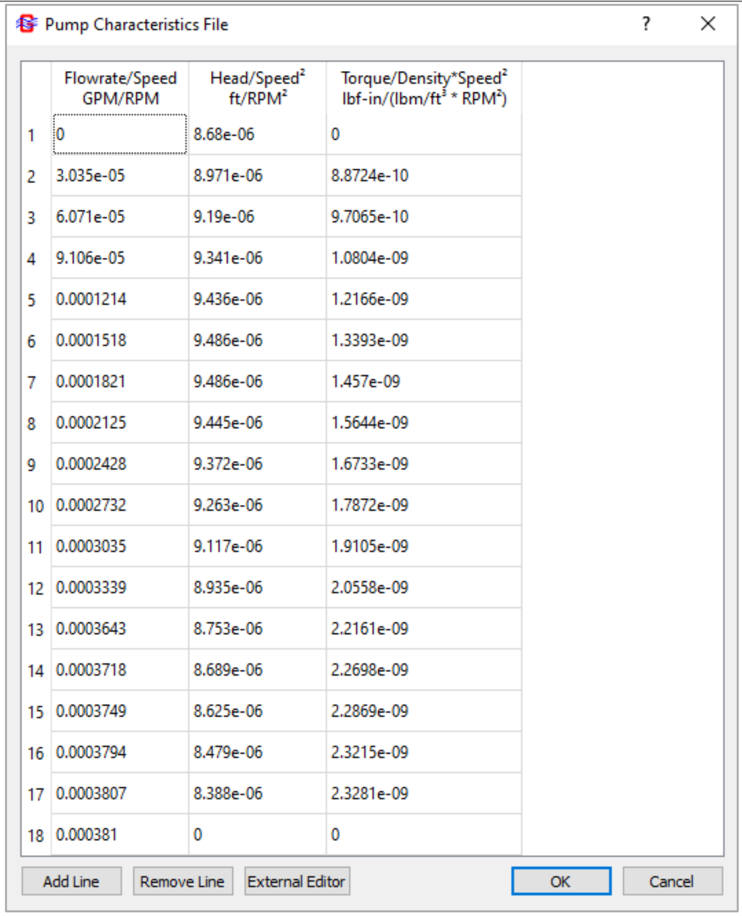


Figure: Turbopump advanced option properties window


Pump Characteristics File

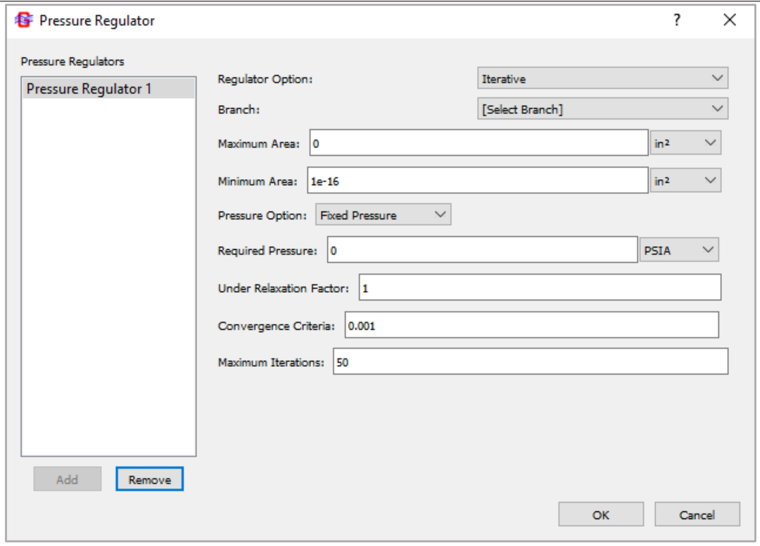
	Flowrate/Speed GPM/RPM	Head/Speed^2 ft/RPM^2	Torque/Density*Speed^2 lbf-in/(lbm/ft^3 * RPM^2)
1	0	8.68e-06	0
2	3.035e-05	8.971e-06	8.8724e-10
3	6.071e-05	9.19e-06	9.7065e-10
4	9.106e-05	9.341e-06	1.0804e-09
5	0.0001214	9.436e-06	1.2166e-09
6	0.0001518	9.486e-06	1.3393e-09
7	0.0001821	9.486e-06	1.457e-09
8	0.0002125	9.445e-06	1.5644e-09
9	0.0002428	9.372e-06	1.6733e-09
10	0.0002732	9.263e-06	1.7872e-09
11	0.0003035	9.117e-06	1.9105e-09
12	0.0003339	8.935e-06	2.0558e-09
13	0.0003643	8.753e-06	2.2161e-09
14	0.0003718	8.689e-06	2.2698e-09
15	0.0003749	8.625e-06	2.2869e-09
16	0.0003794	8.479e-06	2.3215e-09
17	0.0003807	8.388e-06	2.3281e-09
18	0.000381	0	0

Figure: Example 11 turbopump characteristics file

- Use the Add or Remove buttons to add or remove turbopump assemblies from the model
- The ID numbers of the pump branch and turbine branch, speed, turbine efficiency, turbine diameter, design point velocity ratio, and pump characteristics file must be supplied by the user
- The pump characteristics file (shown in Figure XXX) requires the flowrate divided by the pump speed, the head divided by the square of the speed, and the torque divided by the product of the density and the square of the speed

Pressure Regulator

- After the Pressure Regulator option is selected in Steady/Unsteady options in Model Properties, the user may access the Pressure Regulator option in the Advanced dropdown menu
- NOTE: GFSSP supports a maximum of 10 pressure regulators in a single model

	 *Figure: Pressure regulator advanced option properties window* • Use the Add or Remove buttons to add or remove pressure regulators from the model • The regulator model algorithm, ID number of the branch representing the regulator, maximum area, minimum area, pressure option, required pressure, under relaxation factor, convergence criteria, and maximum number of iterations must be supplied by the user • The available regulator model algorithms are iterative and forward-looking and can be chosen using the Regulator Option dropdown • The iterative algorithm allows only one pressure regulator per model. Each time step is repeated with varying flow area until the desired pressure is reached (within the tolerance of the convergence criteria) • The forward-looking option has the flexibility of using multiple regulators and runs faster. The flow area of the regulator is adjusted just once per time step. • A pressure regulator may be applied to a Restriction or Compressible Orifice branch using the Branch dropdown • Pressure Option gives the user the option to model a fixed pressure or use a time-varying pressure history file • The fixed pressure or pressure history file can be edited in the Required Pressure or Pressure History File text box • If the Iterative pressure regulator option is selected, the under-relaxation factor, convergence criteria, and maximum number of iterations must be specified
Flow Regulator	• After the Flow Regulator option is selected in Steady/Unsteady options in Model Properties, the user may access the Flow Regulator option in the Advanced dropdown menu

- NOTE: GFSSP supports a maximum of 10 flow regulators in a single model

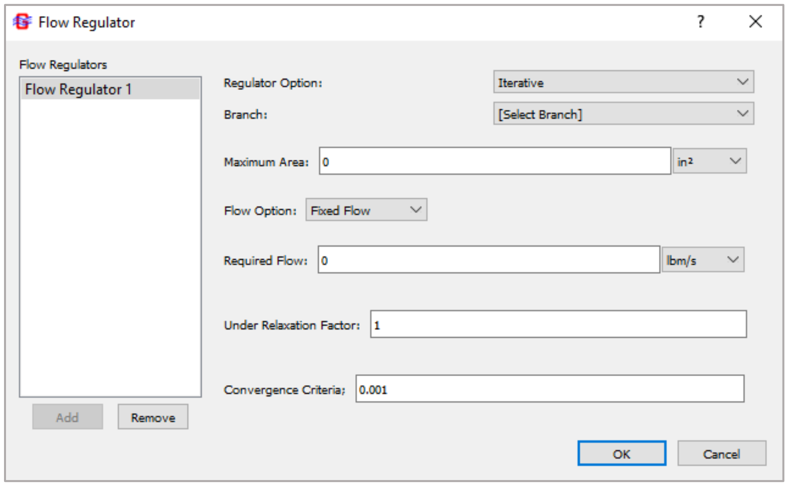


Figure: Flow regulator advanced option properties window

- Use the Add or Remove buttons to add or remove flow regulators from the model
- The regulator model algorithm, ID number of the branch, maximum area, flow option, required flow, under relaxation factor, and convergence criteria must be supplied by the user
- The available flow regulator model algorithms are iterative and marching and can be chosen using the Regulator Option dropdown
- The iterative algorithm allows only one flow regulator per model. Each time step is repeated with varying flow area until the desired flow rate is reached (within the tolerance of the convergence criteria).
- The marching option has the flexibility of using multiple regulators and runs faster, however, it may require relaxation for numerical stability. The flow area of each regulator is adjusted just once per time step.
- A flow regulator may be applied to a Restriction or Compressible Orifice branch using the Branch dropdown
- Flow Option gives the user the option to model a fixed flow or use a time-varying flow history file
- The fixed flow or flow history file can be edited in the Required Flow or Flow History File text box
- If the Iterative flow regulator option is selected, the convergence criteria option must be specified

Pressure Relief Valve

- After the Pressure Relief Valve option is selected in Steady/Unsteady options in Model Properties, the user may access the Pressure Relief Valve option in the Advanced dropdown menu

- NOTE: GFSSP supports a maximum of 10 relief valves in a single model

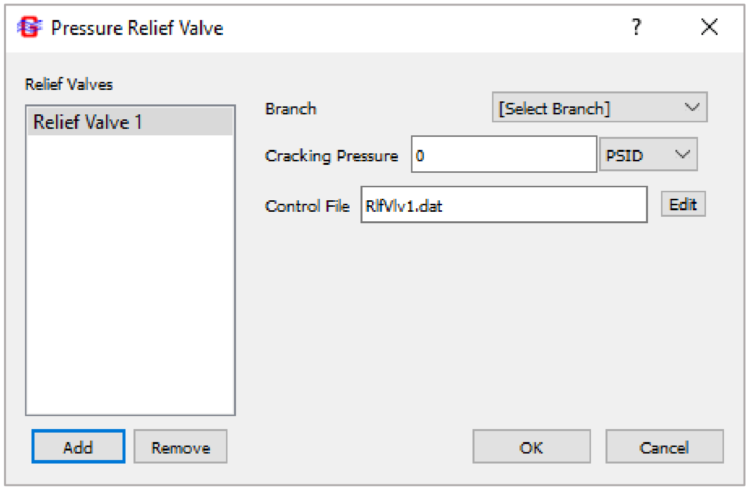


Figure: Pressure relief valve advanced option properties window

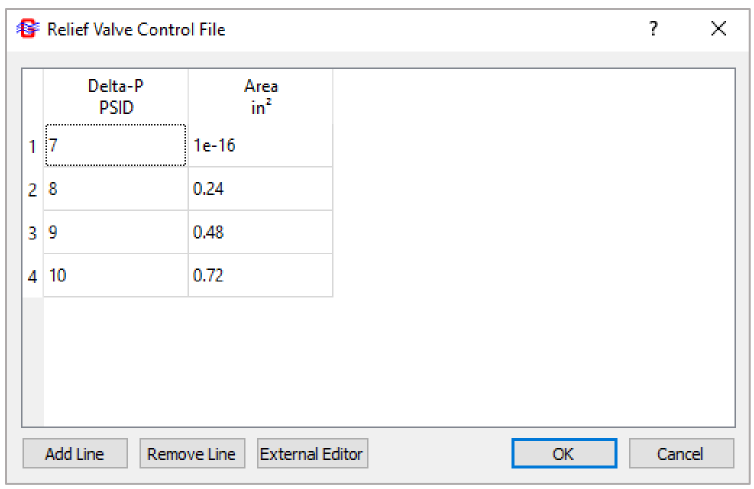


Figure: Example 24 pressure relief valve control file

- Use the Add or Remove buttons to add or remove relief valves
- The ID of the Restriction or Compressible Orifice branch, cracking pressure, and control file for the relief valve must be supplied
- The relief valve control file specifies the open area of the valve for given pressure differentials (seen in Figure XXX)

5.5 Execution and Results

5.5.1 Run Manager and Output File

After clicking Run GFSSP Solver on the toolbar or selecting Model > Run Solver, MIG writes the input text file before executing the GFSSP solver. If the input text file already exists, MIG will ask if the user would like to overwrite the file. After this, the GFSSP run manager opens.

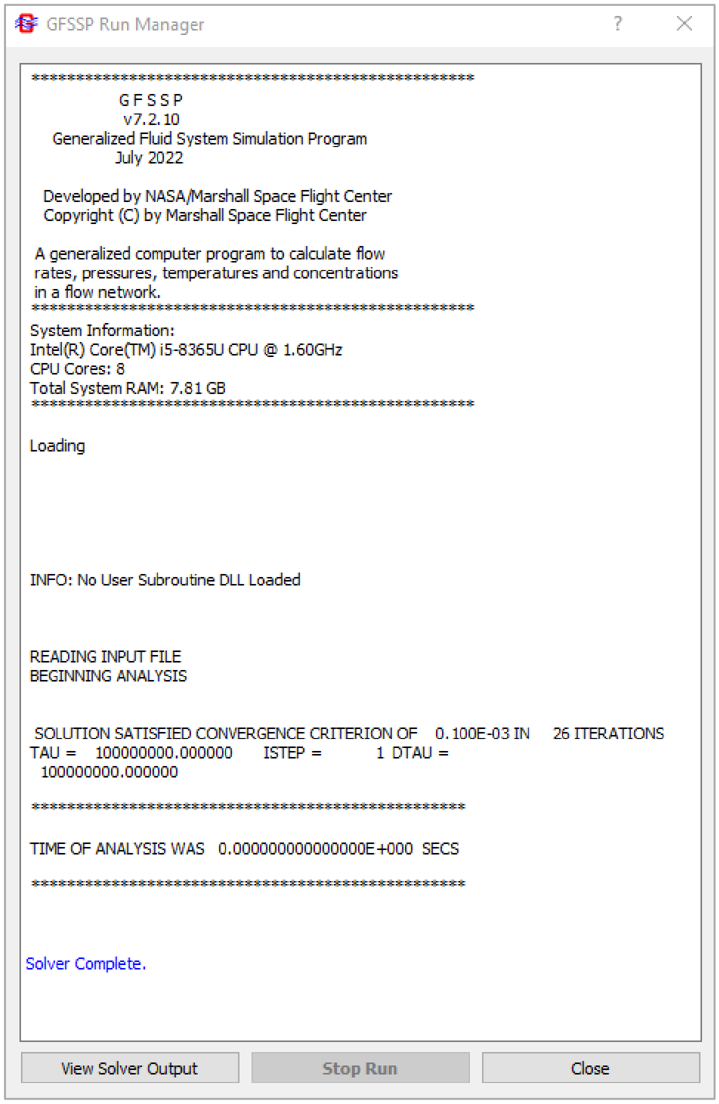


Figure: GFSSP Run Manager for steady-state models

The GFSSP solver prints out information about iterations, convergence, and analysis time to the run manager over the course of the simulation. The user may plot binary WinPlot data during the runtime of an unsteady model by selecting Open WinPlot. To stop the simulation, click the Stop Run button. After the simulation is finished running, the user may click View Solver Output to open the output file generated by GFSSP containing the solutions for the various quantities solved by GFSSP. Clicking the Close button will exit the user out of the run manager, returning to the main model screen.

5.5.2 Steady-State Results Display

MIG offers the option to display steady-state simulation results on the model canvas. To set up the results to display, navigate to Model > Steady-State Results > Preferences or click the Results Display Preferences icon on the toolbar. Results can be displayed on all nodes and branches by selecting Model > Steady-State Results > Enable All or on select nodes and branches by right-clicking a node or branch and using the options discussed in section 5.3.

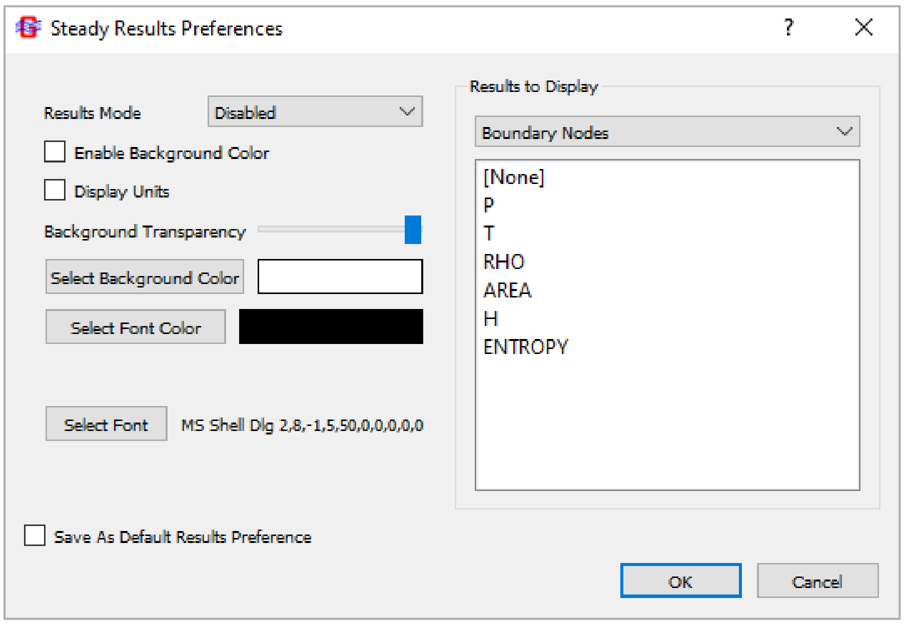


Figure: Steady-state results display preferences window

The Steady Results Preferences window in Figure ____ allows the user to customize the result display. Results Mode allows the user to choose whether results are always displayed, displayed on hover, or disabled. Clicking the **Show Results on Hover** icon on the toolbar will set this option to Show on Mouse Hover. The user may set a background color and adjust its transparency using the Enable Background Color, Background Transparency, and Select Background Color options. The font style and color can be changed using the Select Font and Select Font Color buttons. The Results to Display section allows the user to choose which quantities to display. Use the dropdown menu to select the appropriate model element and left click all the quantities to display. To show the units of the displayed results, select the Display Units checkbox. The settings selected in the preferences window can be saved as the default preferences for results display be selecting the Save As Default Results Preference.

5.5.3 MIG Plotter

MIG offers a built-in plotting tool for viewing unsteady model results saved in a Winplot binary (*.WPL) file. The user can plot any desired parameter solved by GFSSP, customize the zoom of the plot, and copy an image of the plot. The tool features two different ways of viewing results: temporal and profile view.

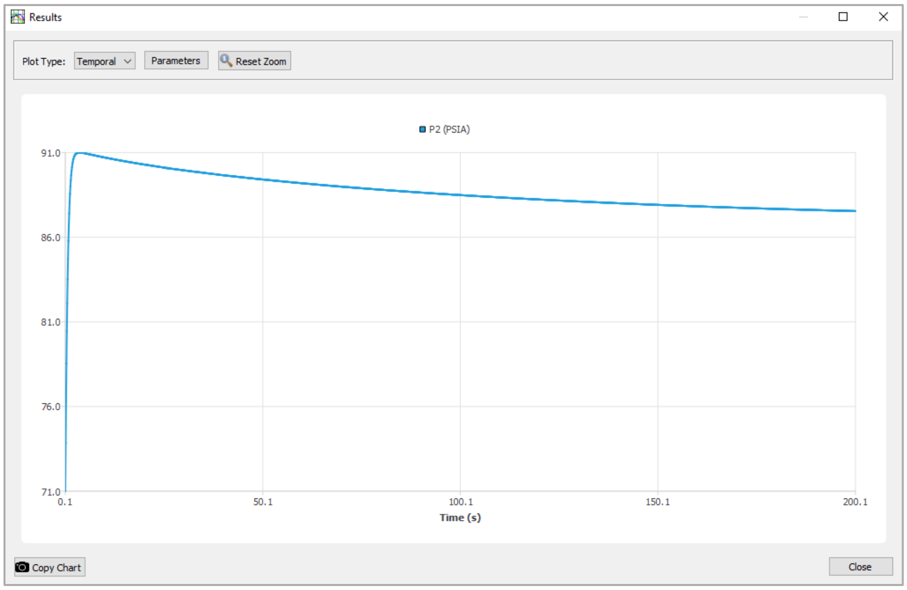


Figure: Temporal view of the pressure in node 2 of example 10

Temporal view allows the user to view the change in a quantity at a desired node or branch over the time of the simulation. Figure ____ shows a temporal view of the pressure in node 2 of example 10.

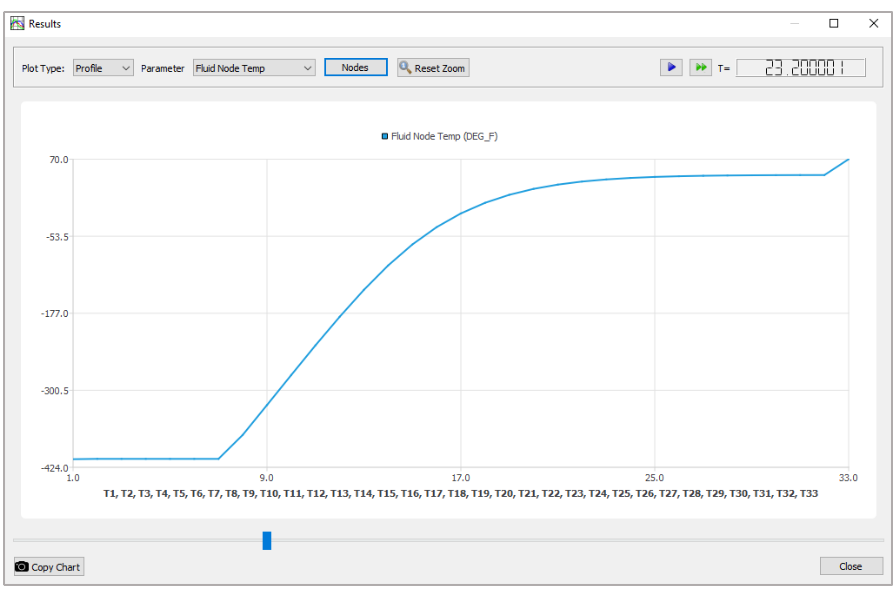


Figure: Profile view of the temperature of all nodes in example 14

Profile view allows the user to view the value of a quantity at all nodes or branches simultaneously. It is most useful for “straight-line” models with evenly spaced nodes. The user can select and sort the nodes to plot in the profile view by clicking the Nodes button. The horizontal axis of the plotter represents each node/branch while the vertical axis represents the value of the quantity. The user may use either the scroll bar at the bottom or the pause/play and fast forward buttons to view how the quantity changes at each node/branch with time. The box in the upper right corner contains the current simulation time. Figure ____ shows a profile view of fluid node temperature in example 14.

To zoom into a section of the graph, left click and drag the mouse cursor over the desired rectangular zoom area. The zoom setting can be reset to its default value by clicking the Reset Zoom button. The Copy Chart button will copy a picture of the graph to the device clipboard. To exit the MIG plotter, press Close.