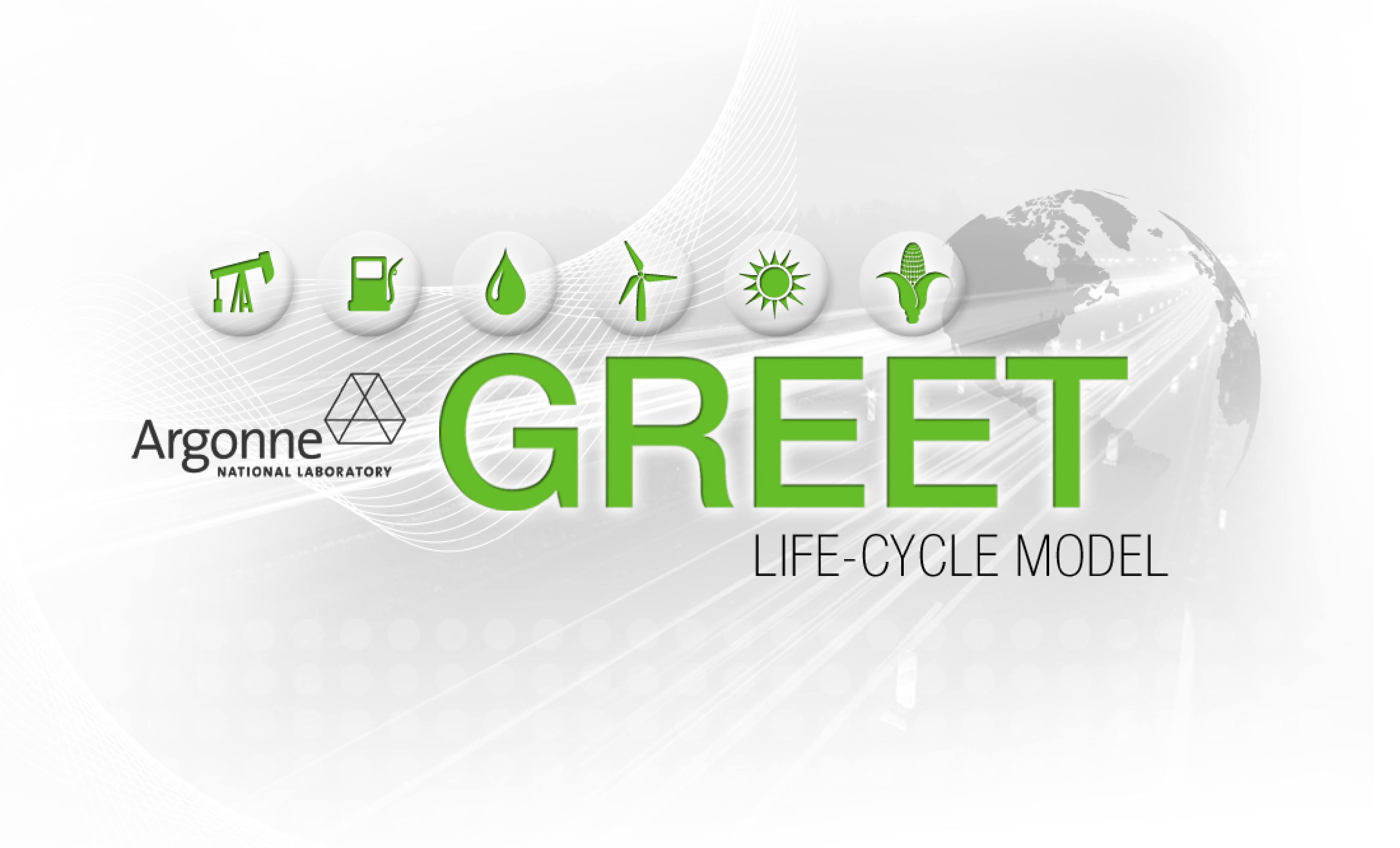


User Guide

Center for Transportation Research
Energy Systems Division
Argonne National Laboratory

December 21, 2012

Contents

List of Figures

1 Installation

GREET is available at `http://greet.es.anl.gov/greet/setup`.

Clicking the **Install** button will download the GREET installer. You do not need administrator privileges to install the software.

1.1 Installation Folder

The GREET executable file is stored in your personal folder, depending on which version of Microsoft® Windows® is used. We do not encourage you to try accessing directly that folder except for one particular reason: using the command line tool. This will be described in the Command Line section of this manual.

2 Starting GREET

2.1 First Use

To start GREET, click on the GREET shortcut located on your Desktop or in the Start menu.

When you start the software for the first time, a registration form will pop up. Please fill in the registration form and click Register. See Figure 1.

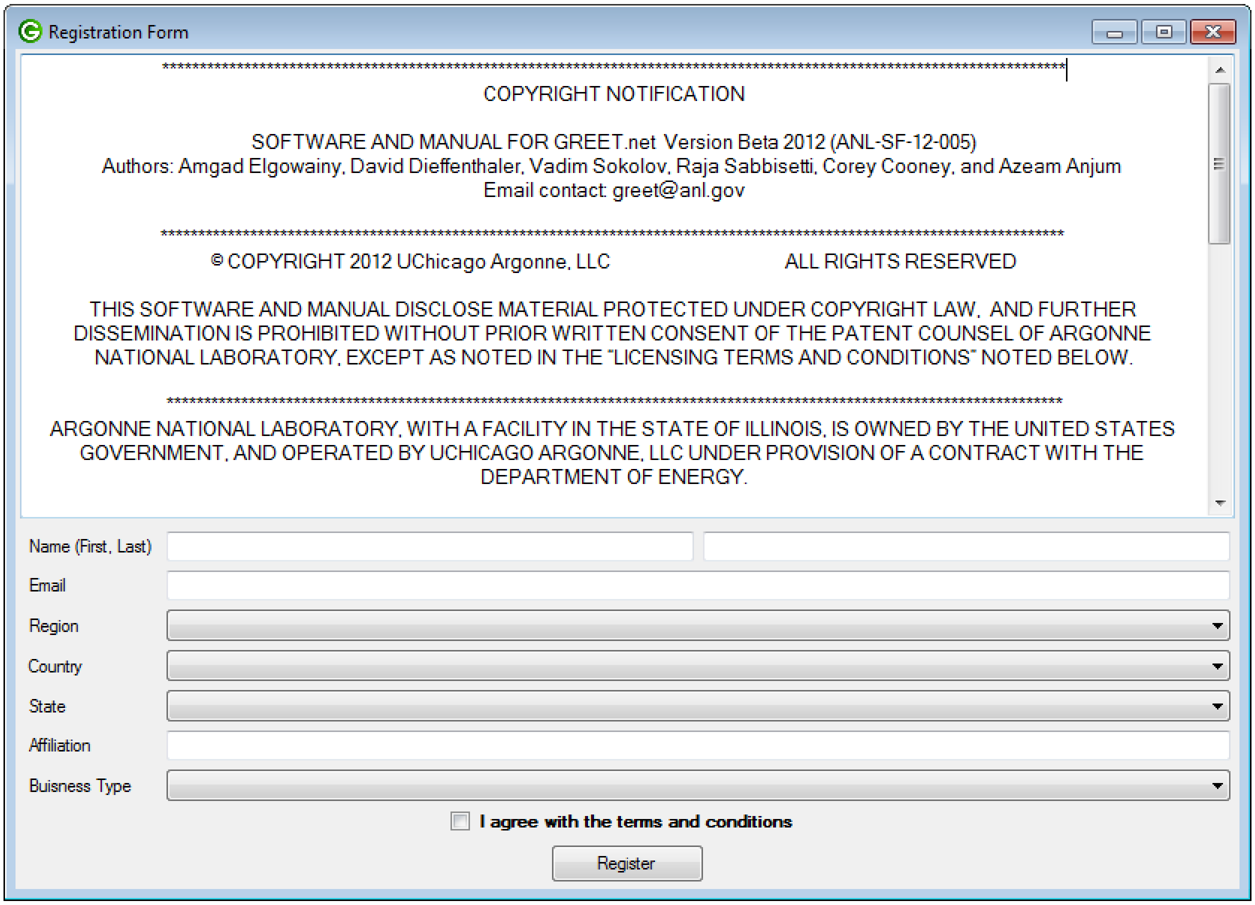


Figure 1: GREET Registration Form

After installation, GREET contains no initial database. A message box will offer you the opportunity to download the latest available data. See Figure 2.

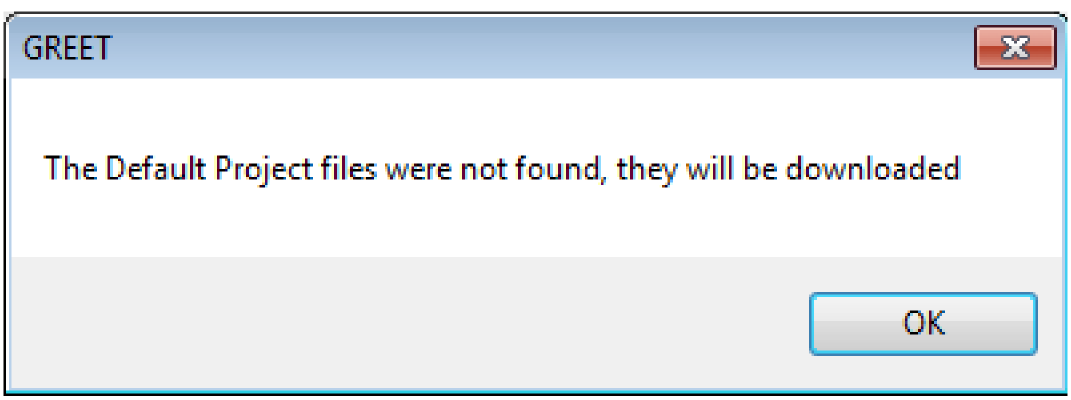


Figure 2: GREET Prompt for Downloading the Database

GREET is composed of a single file, which is by default named, Default.greet. This file will be downloaded and placed in your My Documents\Greet\Data folder.

Once the file is downloaded, GREET will load this default database.

3 GREET Usage

3.1 Loading a Project

By default, when GREET is started, it loads the default project file which is located at the default location: My Documents\Greet\Data\Default.greet. This behavior can be changed so that the latest used project file is loaded on start up by adjusting the preferences (see section 4.4.4).

To load another file, open the **File** menu and click **Open** a project. See Figure 3.

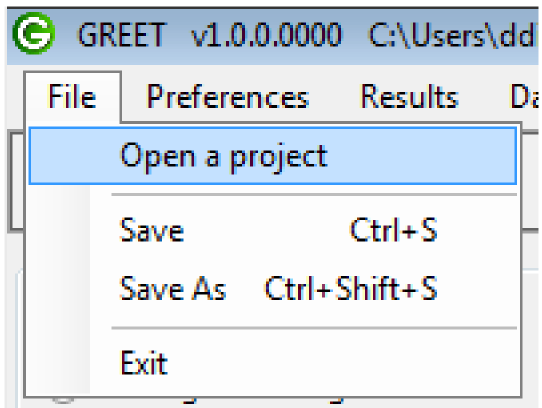


Figure 3: GREET File Menu

Double clicking a .greet file on your system will start the application and load the clicked project.

3.2 Saving a Project

To save a project you can either click **Save** or **Save As** from the **File** menu.

By default, you are not allowed to save over the Default.greet file for two major reasons. First, you'll alter the values that are provided by the Argonne research team; and therefore, the results can't be guaranteed to be accurate. Secondly, when data updates are available, the Default.greet file is overwritten with the newer version. Therefore, all of the local changes will be lost.

This is why we strongly encourage you to save your data in other files, rather than in the Default.greet file. If you still want to overwrite this file, check the **Show Advanced Features** option in the Preferences of GREET (see Section 4.4.4).

3.3 The .greet Files

GREET uses .greet files to store the database. By double clicking one of these files, GREET will automatically load it. See Figure 4.

Figure 4: Typical GREET Icon

A .greet file contains the following data sets:

- Resources
- Pathways
- Processes
- Technologies
- Transportation Modes
- Pollutants
- Pathway Mixes
- Locations
- Input Tables
- Vehicles
- Pictures
- Unit System Used
- Monitored Results

Data is stored in plain text, using XML; and thus, the data file can be opened by any text editor. However, the data might be compressed, depending on the options set in the preferences (see Section 4.4.4). A compressed file will not be readable by humans, so to programatically or manually alter this file, use the non-compressed version.

NOTE: Be aware that any modifications to this file might break relationships in the database, so this needs to be done really carefully.

3.4 Main Panes Selector

The main panes of GREET can be accessed using the four buttons represented in Figure 5.

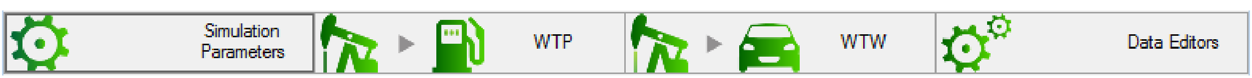


Figure 5: GREET Main Panes Selector

There are four main panes:

- Simulation Parameters
- WTP (Well-to-Pump)
- WTW (Well-to-Wheels)
- Data Editors

3.5 Simulation Parameters Pane

The Simulation Parameters main pane contains parameters that do not fit in our model (for more details on the model, see the *Mathematical Model* document [1]). It contains input tables and tabs. The tables are Microsoft® Excel-like objects and support similar in-cell calculations (see Figure 6).

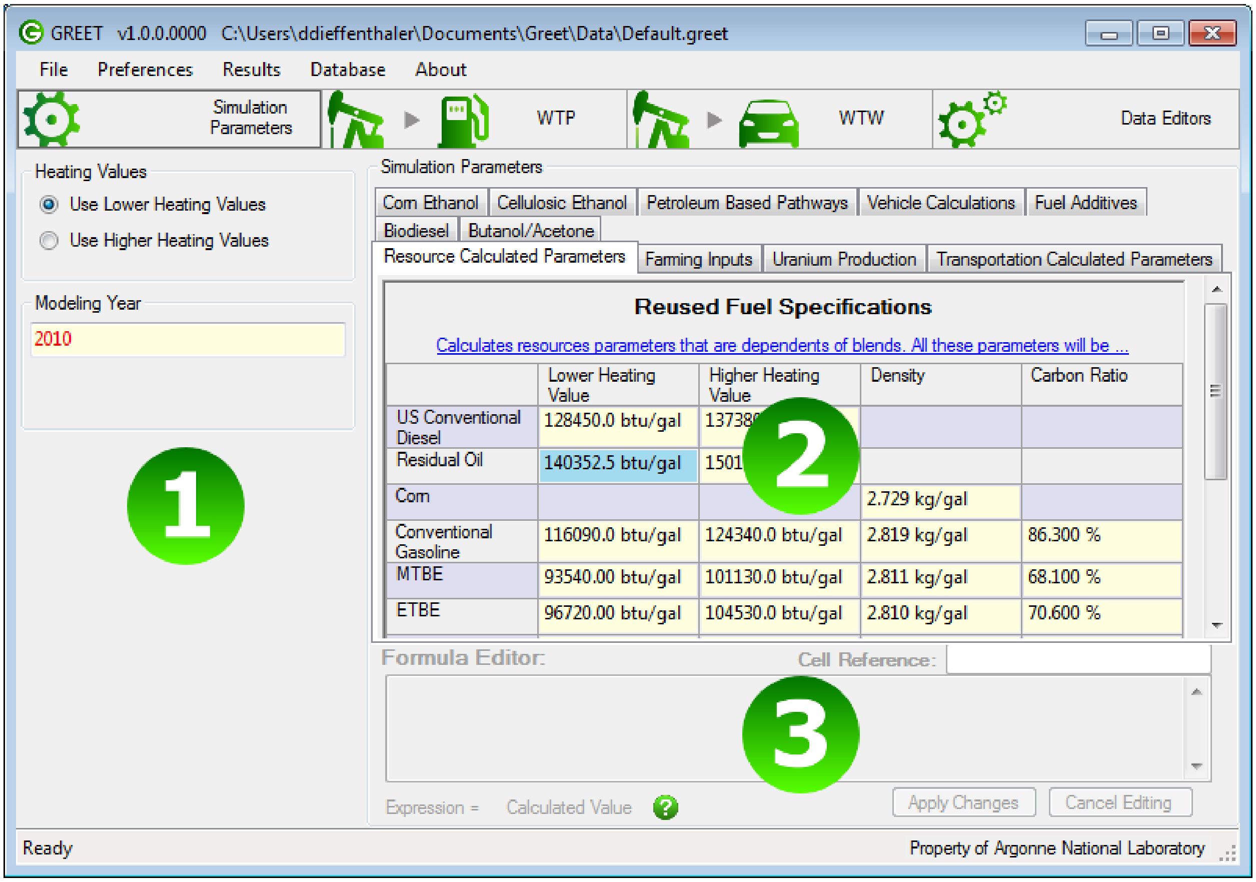


Figure 6: Global Parameters Main Pane

The Simulation Parameters main pane is composed of three main zones. See Figure 6.

Zone 1 shows the Other Options. It includes the selection between using lower-heating values (LHV) or high-heating values (HHV) for converting different quantities of material to energy quantities. The modeling year is used to select an appropriate parameter value, when an input parameter is given in a form of a time series.

Zone 2 shows the tabs that contain input parameters.

Zone 3 shows the area to change a formula used in a cell of an input table.

Then, a simulation parameter can be used in the model, by reference. A process, for instance, can refer to an efficiency parameter stored in one of the tables. References that can be specified use the following syntax: "[TABLE_NAME!COLUMN_LETTER ROW_NUMBER]". The same rules apply as in the formula editor (see Section 3.5.2).

3.5.1 Color Code for Cells

Different colors are used for cell text and cell background. The background color is used to distinguish constants from calculated values. Yellow is used for cells that store constants and blue is used for cells that store formulas. Furthermore, the color of the cell text is used to indicate whether the value is the default GREET value or if it is user-defined. See Figure 7.

Color Code	Explanation
	Yellow cells stores constants
	Blue cells stores formulas
15%	Black font represents GREET defaults
32%	Red font represents user value

Figure 7: Cell Color Codes

You can switch between the default GREET value and the user-defined value by right clicking a value cell and then clicking **Switch to default value** or **Switch to user value**. The font color will change between red and black, indicating which value is currently being used. See Figure 7.

3.5.2 Using Formulas and References

Cells can be edited in a Microsoft® Excel-like way. You can click on a cell and enter a value or use the Formula Editor on the bottom of the form to edit a formula (see Figure 8). References to other cells can be used in the formula editor. For example, [Uranium_Enrichement!C2] is a reference to the Uranium Enrichment table cell C2 which is the value in the second row, third column. In a more general way, the references obey the following syntax "[TABLE_NAME!COLUMN_LETTER ROW_NUMBER]".

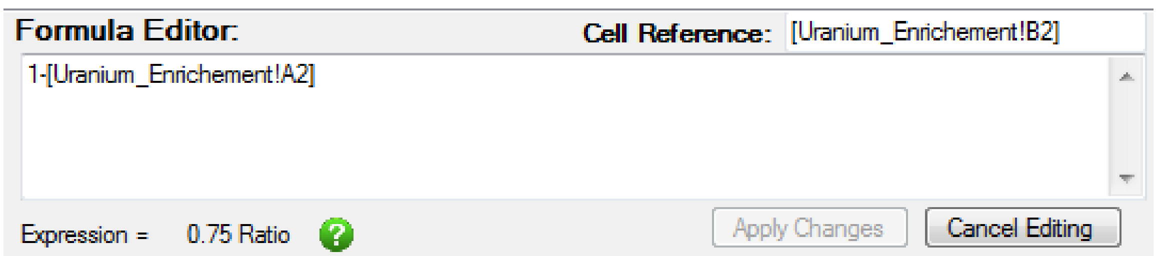


Figure 8: Formula Editor

While in the formula editor, you can use the pointing feature. Pointing allows you to click your mouse on another cell containing the data and add a reference to this cell. You will need to add at least one of the basic operators or mathematical functions to point and click another cell to add a reference.

Besides basic mathematical operations (subtraction, addition, division, multiplication) and references to other cells, you can use some of the following standard mathematical functions.

- **ln**(val): returns the logarithm of the value
- **e**(val): returns the exponential of the value
- **sin**(val): returns the sine of the value in radians
- **cos**(val): returns the cosine of the value in radians
- **tan**(val): returns the tangent of the value in radians
- **asin**(val): returns the arcsine in radians of a value which must be greater than or equal to -1, but less than or equal to 1
- **acos**(val): returns the arccosine in radians of a value

- **atan**(val): returns the arctangent in radians of a value
- **log**(val): returns the logarithm base 10 of the value which must be greater than or equal to -1, and less than or equal to 1
- **if**(cond1=cond2,val true, val false): returns val true if cond1 is equal to cond2 and val false otherwise

3.5.3 Units

Units can be changed in the cells by right clicking on the value and selecting Change Unit. A pop-up window will appear offering you a selection of units to choose from. See Figure 9.

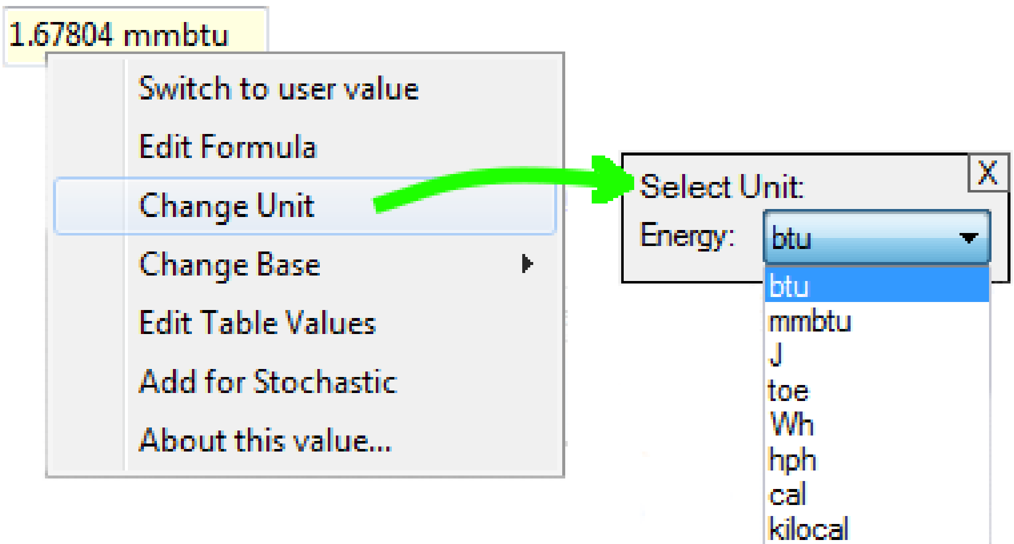


Figure 9: Changing the Unit of a Value in GREET

You can also enter a value and its unit directly. For example, 1 MJ or 947.817 Btu both represent the same quantity. However, the final value shown depends on the unit preferences (see Section 4.4.1).

NOTE: The units preferences are global. Thus, if you change a mass to be displayed in lb, all the masses will be displayed in lb in the graphical user interface (GUI). If the **Automatically add prefixes** option is checked in the preferences, it might happen that you'll see different prefixes in front of a unit, such as J, kJ or MJ, depending on the order of magnitude of the corresponding value.

3.5.4 Adding a New Table

To add a new table in a tab, right click in the background area of a Tab and select **Add table**. See Figure 10.

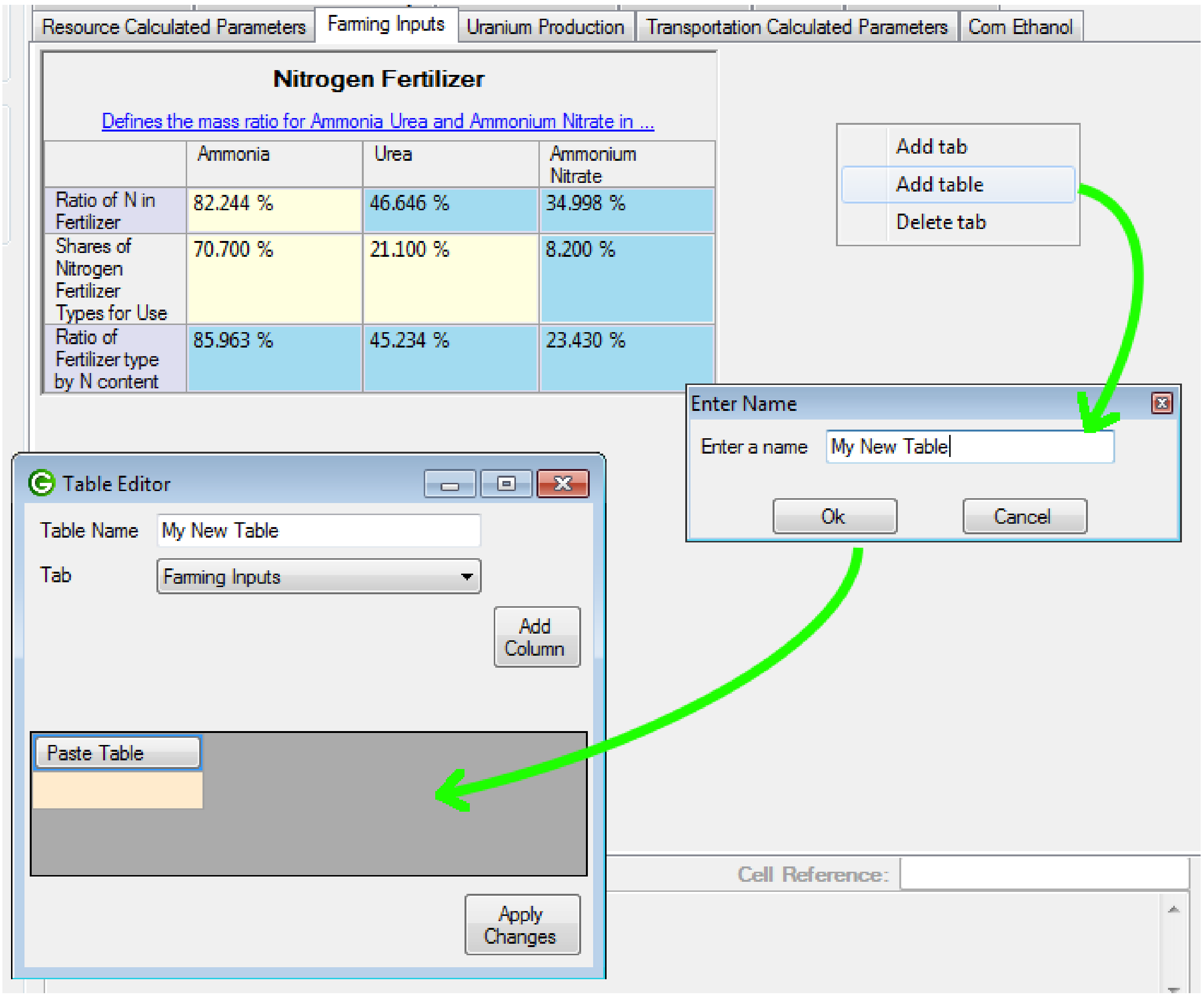


Figure 10: Adding Global Parameters Table

You will be asked to enter a table name. For now, there is no option to change the name once it is defined, so please enter this table name carefully.

The table editor will open in a new form and it will allow you to add rows and columns to a new table. The first row and first column of the table are reserved for naming what is in the table and they are represented by salmon colored cells. Values can be added into the white cells.

To add a column, press the **Add Column** button on the top right of the window (see Figure 10). To add a row, input a name for the current last row and a new row will be created automatically.

Filling in every cell is not required.

3.5.5 Editing an Existing Table

To edit a table from the simulation parameters, right click on the name of a table and select **Edit table**. The table editor will open in a new window and show you the structure of the table as well as the values. Values can be entered along with their units; for example, one can type "2500 btu/ton" to enter a quantity of energy per unit of mass. The tab on which that table is going to be located can also be changed. To change the table location, use the Tab selection box available just below the name and select another tab (see Figure 10).

3.6 Well-to-Pump (WTP) Pane

Clicking the **WTP** button in the main panes selector will open the Well-to-Pump main pane. The calculations will run automatically, if needed. A new window that shows the resources that are being calculated will pop up; this window is called the convergence monitor. See Figure 11.

3.6.1 Convergence Monitor

The convergence monitor shows the resources that are being calculated. The underlying algorithm in the GREET model is an iterative algorithm which converges to a fixed point [1]. The convergence monitor shows the convergence status for each resource. If neither Pathway Mix nor pathways are defined for a resource, the calculations do not run for this resource (to edit Pathway Mix, see Section 3.8.6).

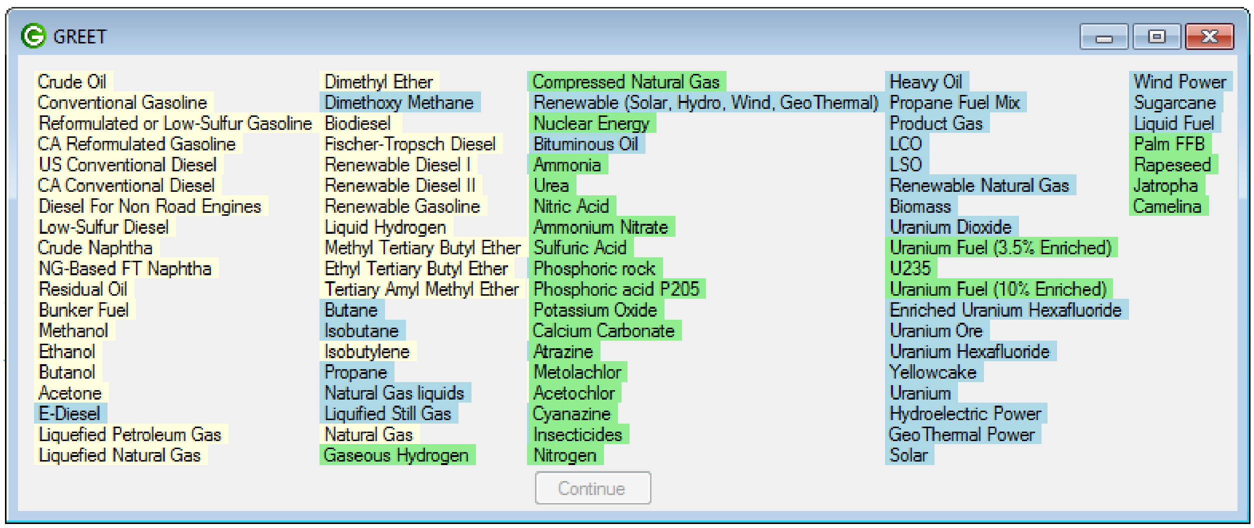


Figure 11: Convergence Monitor

Interpreting the color code:

- Green background: Resource already converged
- Yellow background: Resource being processed
- Blue background: Resource not calculated

A disabled button labeled **Continue** can be seen on the lower center of the convergence monitor. This button can be enabled in the general preferences so the program pauses between each iteration (see Section 4.4.2).

3.6.2 Exploring the Structure of Pathways and Pathway Mixes

To explore the results in GREET, browse through the Pathway-Mix tree to find a pathway or a Pathway Mix you are interested in.

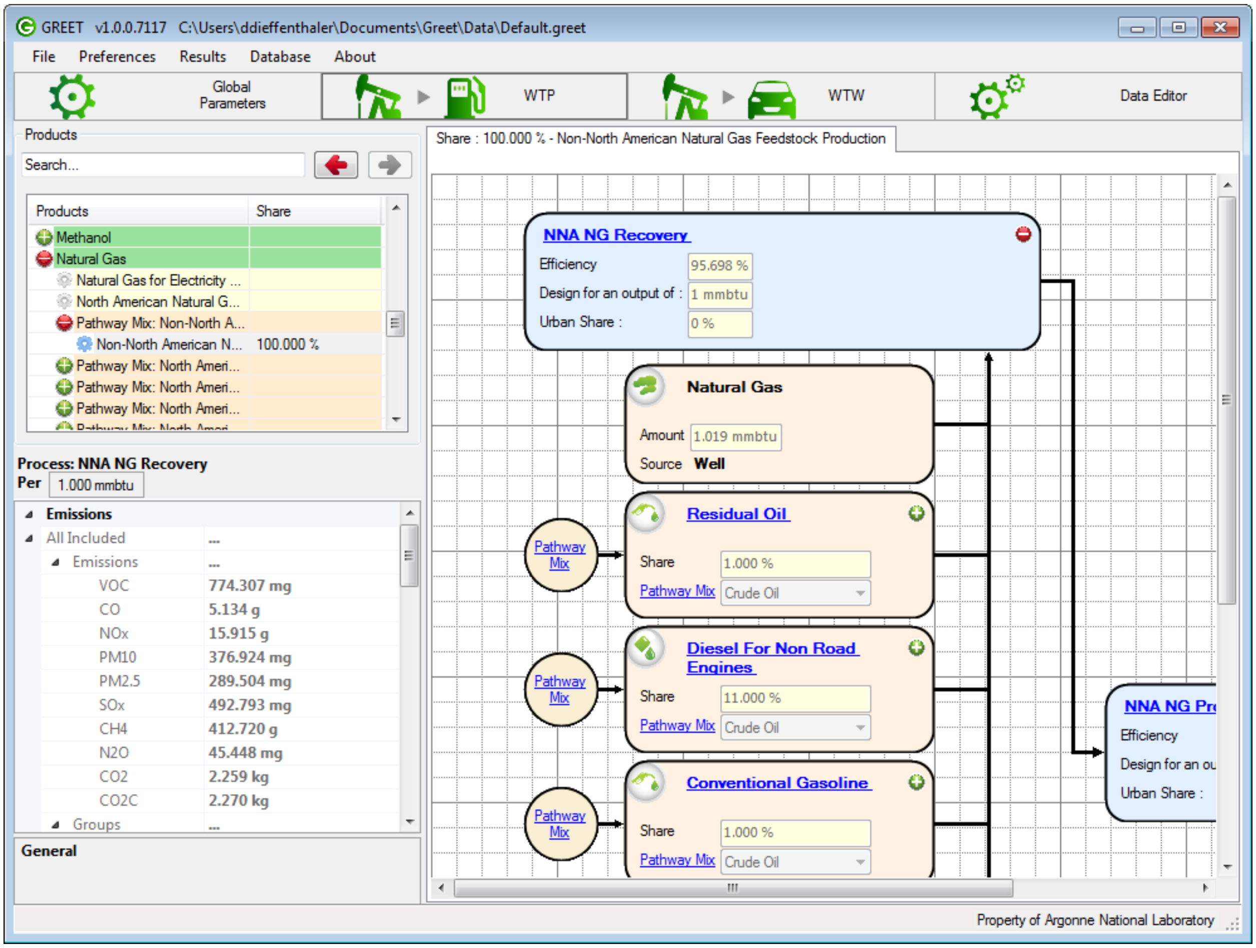


Figure 12: Well-to-Pump Pane Showing the Representation of a Pathway

The WTP pane has 3 zones, as shown on Figure 13. Zone 1 represents the Pathway-Mix tree. Zone 2 shows the results associated with the selected pathway. To select a pathway, expand a resource node or a Pathway Mix until you've reached an item that is not preceded by a plus or minus sign. In the example above, "Non-North American Natural Gas Feedstock Production" is selected and represented in Zone 3. Zone 3 shows either a representation of the pathway or a Pathway Mix. Figure 12 shows a pathway represented in the explorer and Figure 13 represents a Pathway Mix.

Getting to see more details: Say we would like to see the results for the Pathway Mix for U.S. electricity production. You can type electricity in the search box and expand the Electricity item in the list, then expand the Pathway Mix: U.S. as shown in Figure 13.

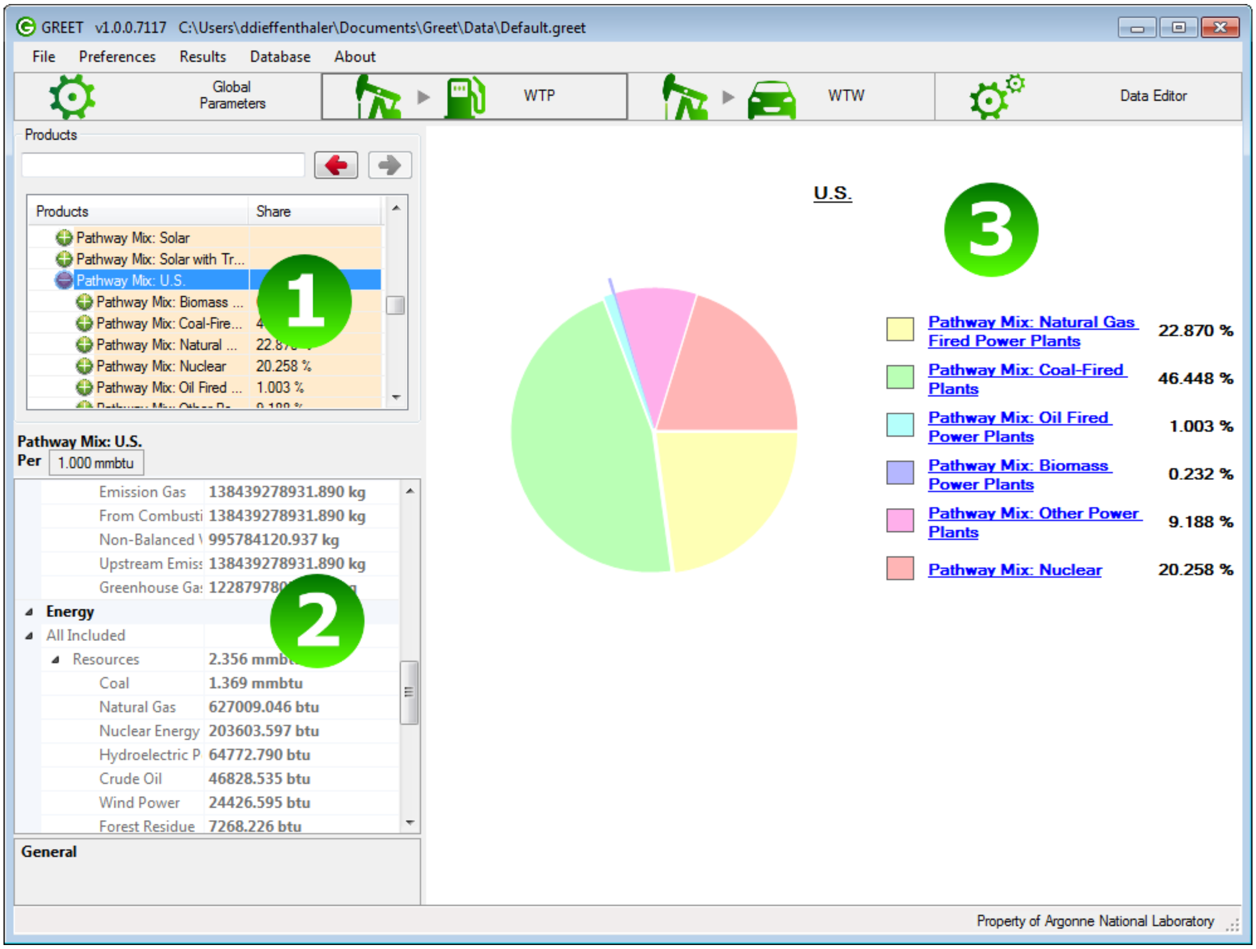


Figure 13: Electricity Produced in the United States Averaged before Distribution by Power Lines

The mix-pathway explorer on the right now represents a Pathway Mix. As it is assumed in the GREET model, electricity in the United States, on average, comes from many different types of power plants (natural gas, coal, oil, nuclear, etc.); all different types of plants are represented there as slices of the pie. To explore what each slice is composed of, click on the names in the legend to the right.

Figure 14 shows what is seen when you click on **From: Natural Gas Fired Power Plants**.

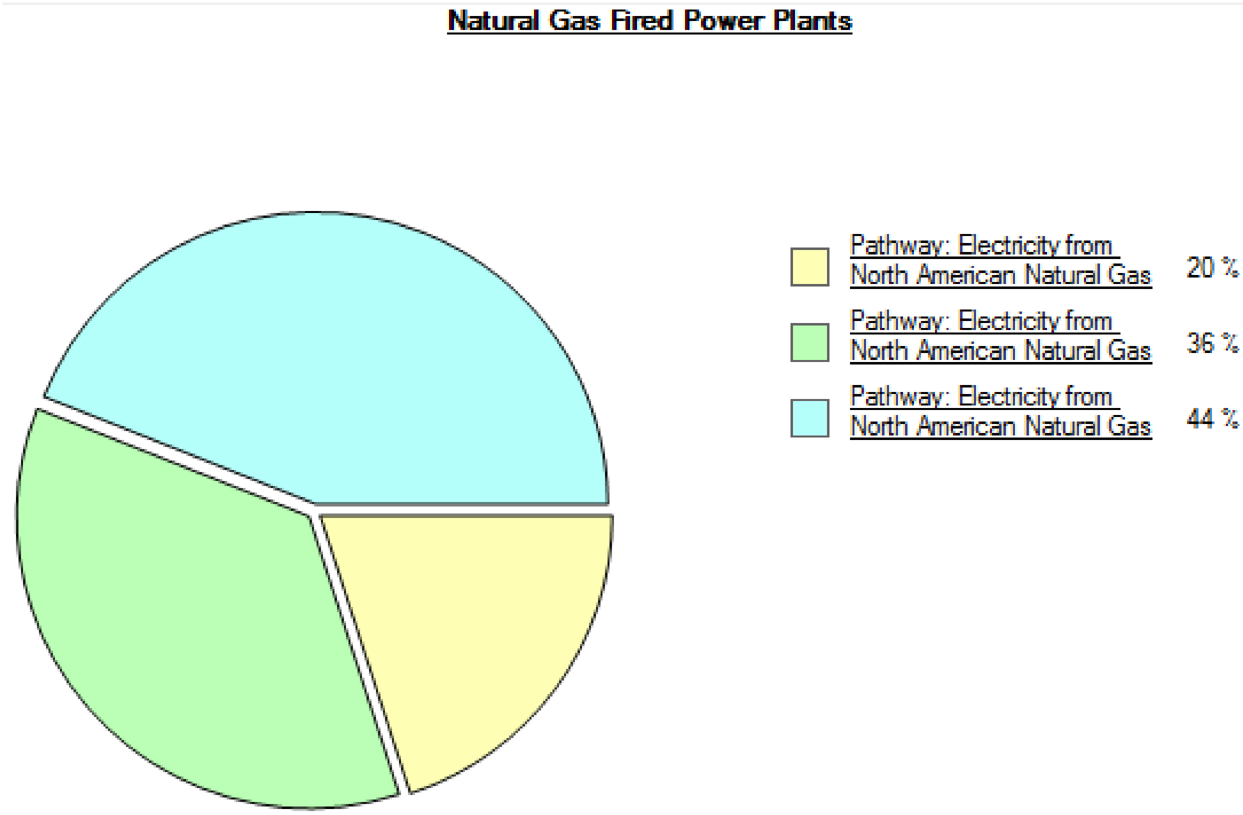


Figure 14: Natural Gas Fired Power Plant Mix

Because the Natural Gas Fired Power Plants is a weighted Pathway Mix of three different pathways, we still see a pie chart in Figure 14. Here, each slice represents a component of the Pathway Mix; pathways, in this case. To see what a pathway is, click on its name in the legend to the right. Figure 15 represents the pathway "Electricity from North American Natural Gas Power Production (Combined Cycle)."

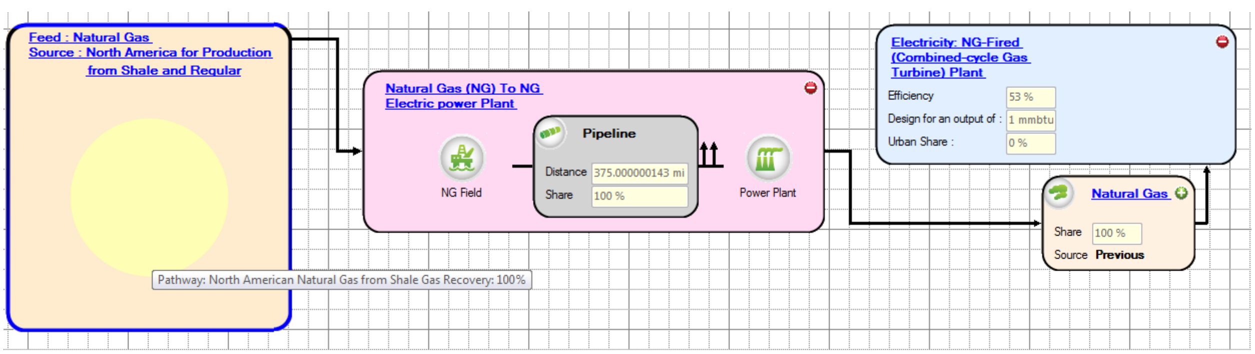


Figure 15: Natural Gas Combined Cycle Power Plants Pathway

A pathway can optionally have a Feed defined. The Feed is used to calculate the upstream emission and energy values. In Figure 15, the upstream for this pathway is the natural gas from North American Production averaged from Shale Gas and Regular Pathway Mix. It is followed by a pipeline transportation process and by the electricity generation at power plant process that has an input – natural gas – used for a Combined Cycle Emissions Turbine.

The processes in Figure 15 are completely expanded. To get to this expanded view, right click on the gray background area and select **Expand All**. The opposite action, **Collapse All**, will show fewer details.

3.6.3 Exploring the Results

At each level of the structure described above, different items can be selected. The user can select a Pathway Mix, a pathway, a process (stationary or transportation), transportation steps, a stationary process input, or a stationary process technology (set of emission factors). Each selection updates the Properties Display on the left of the Well-to-Pump setup pane shown in Figure 16.

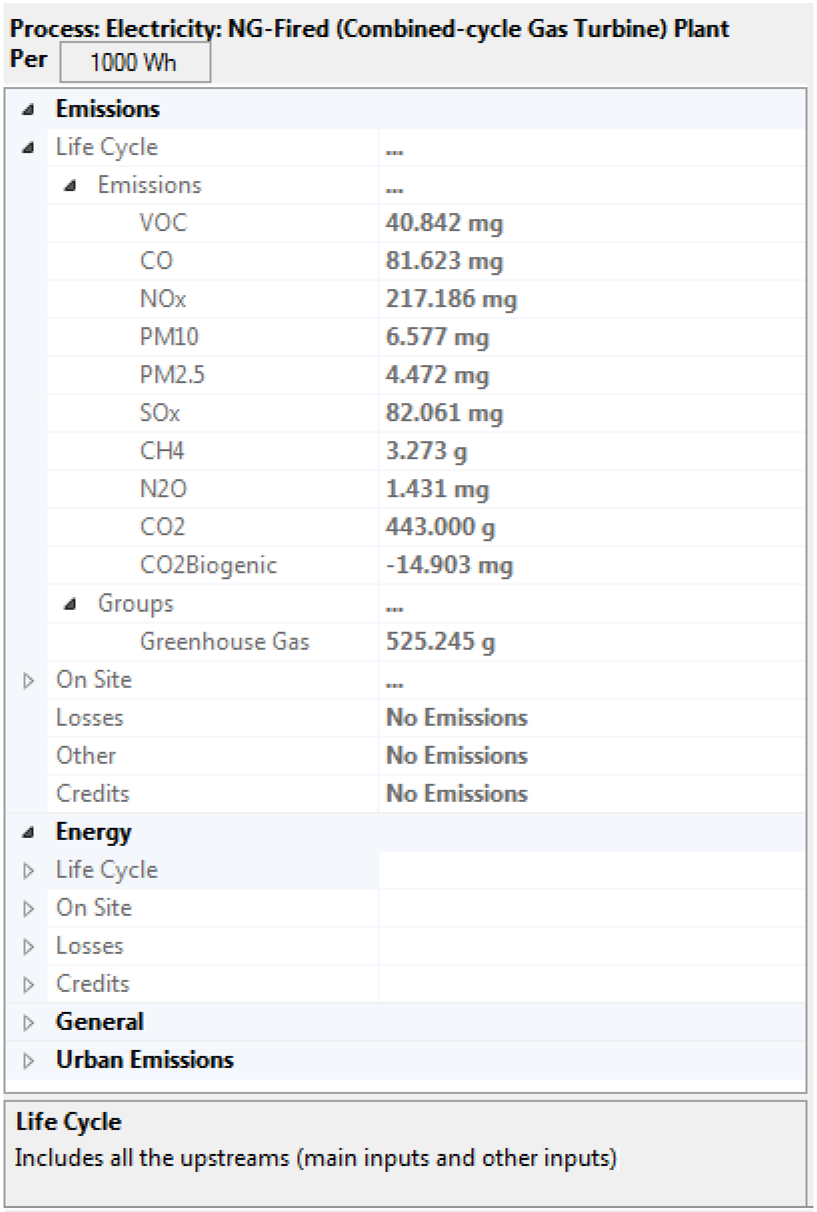


Figure 16: Properties Display for Combined Cycle Electricity Production from Natural Gas

The top of the property display shows the type and the name of the selected object. Below is the list of supported object types.

- Pathway Mix: When a Pathway Mix is selected in the Pathway-Mix tree (Zone 1 of Figure 12).
- Pathway: When a pathway is selected in the Pathway-Mix tree (Zone 1 of Figure 12).
- Process: When a process is selected in the pathway representation (Zone 3 of Figure 12).
- Step: When a transportation step is selected within a transportation process (Zone 3 of Figure 12).
- Input: When an input is selected in an expanded stationary process (Zone 3 of Figure 12).

- Technology: When a technology is selected in an expanded stationary process input (Zone 3 of Figure 12).

Figure 16 shows that a process has been selected and its name is "NG-Fired (Combined-cycle Gas Turbine) Plant."

The functional unit is shown below the name; in Figure 16, the functional unit is 1000 Wh.

Results organization: The results are organized into three categories.

- Emissions
- Energy
- General

Energy, Emissions, and Urban Emissions contain the results for a process. Those categories are further split into several sub categories.

- Life Cycle
- On Site
- Losses
- Other
- Adjustments
- Credits

Life Cycle results sum up all of the energy and emissions associated with the process inputs, including upstream. Energy and emission adjustments associated with co-products are not included.

On Site defines what amount of resources are being used and what emissions are created at the boundaries of the selected process. Neither any upstream for the inputs nor the co-product credits are included.

Losses sum up the results from the losses that might be defined for the output of the processes. Please refer to the Step Losses in section 3.8.4 of the transportation process.

Other is used in rare cases where some ratio emissions (see Section 3.8.3) are defined based on the input's amount. In a simple example, emissions of NO_x, N_2O, and CO_2 are associated with the use of fertilizers during corn production. These particular emissions are dependent on the mass of fertilizer used. This is a very rare case in the GREET model.

Credits section contains results associated with co-products defined for the process.

NOTE: The value shown there is positive; the credits are subtracted from the Life Cycle category.

Plotting the results: Some results can be plotted as bar charts or pie charts using the user interface. To do so, right click on a group of results such as — Energy, Emissions, or Groups — and then select **Plots** and click **Plot values in pie chart** or **Plot values in bar chart**.

The bar chart plotting window, shown in Figure 17, gives you an option to display the amounts on a logarithmic scale. To activate the logarithmic scale for the bar charts, check **Use Logarithmic Scale** box in the top left corner.

The pie chart shown in Figure 18 gives you an option to combine sections of the pie chart which are lower than 1%. To combine sections lower than 1%, check the box on the bottom left of the pie chart window.

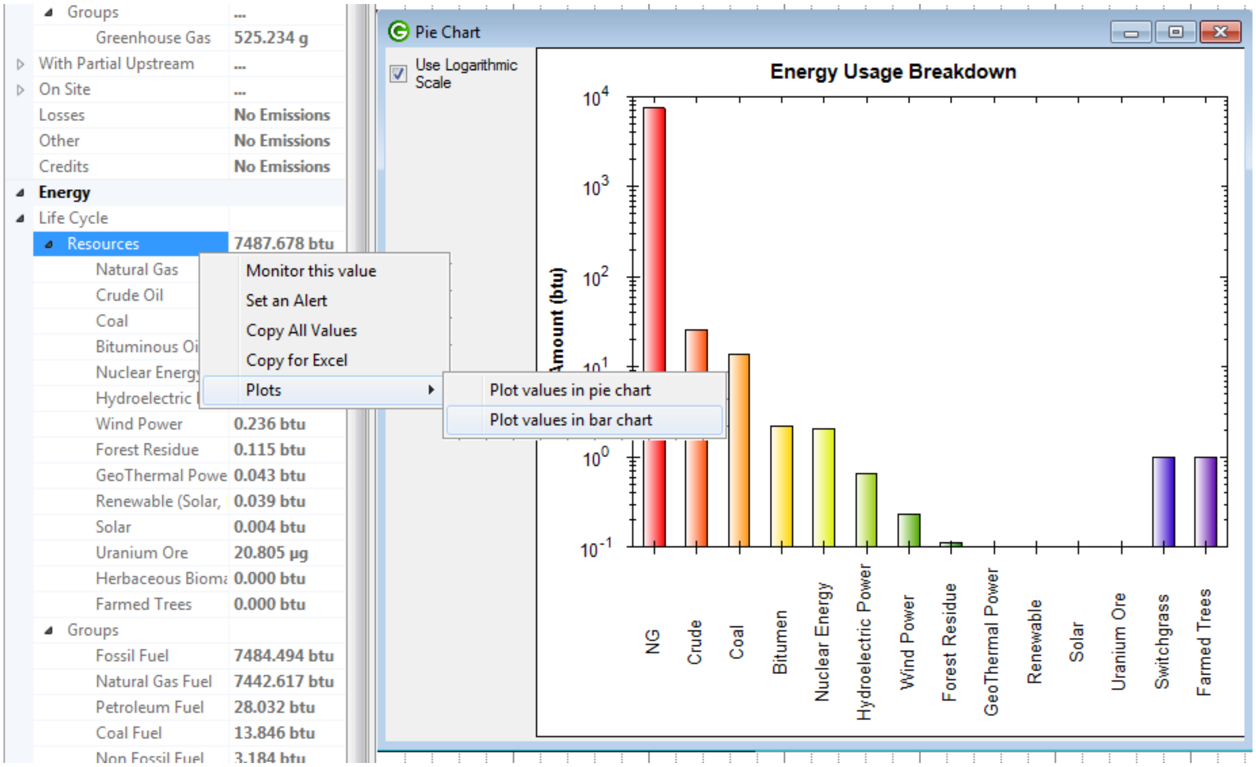


Figure 17: Plotting the Energy Used as a Logarithmic Bar Chart

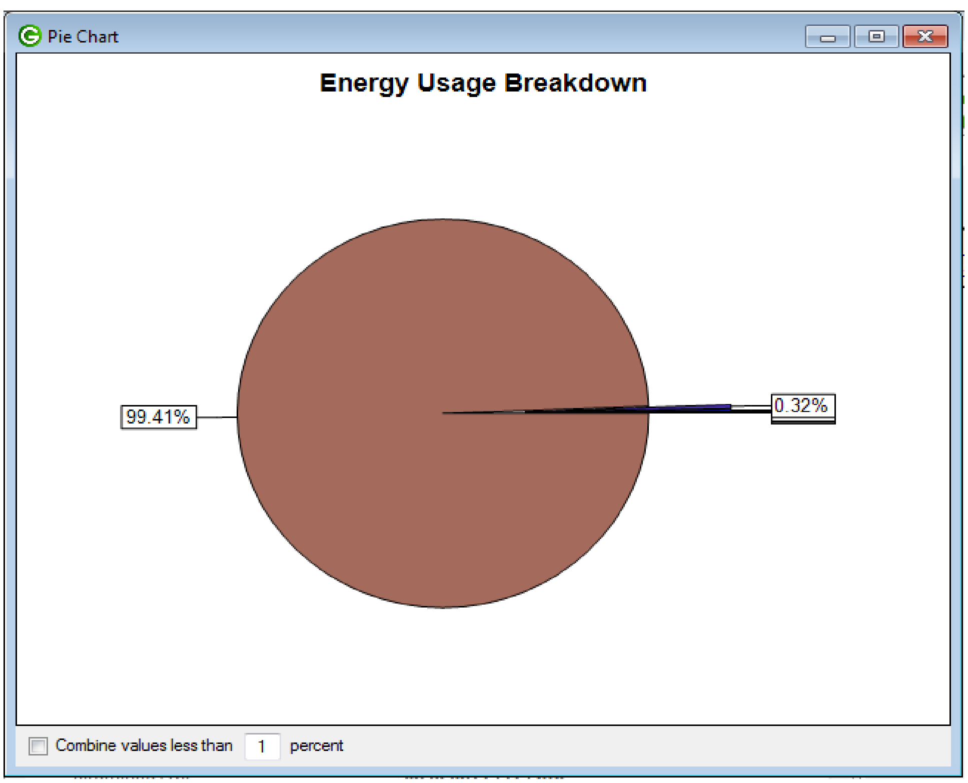


Figure 18: Plotting the Energy Used as a Pie Chart

More options are available for the plots by right clicking in the graph area. See Figure 19.

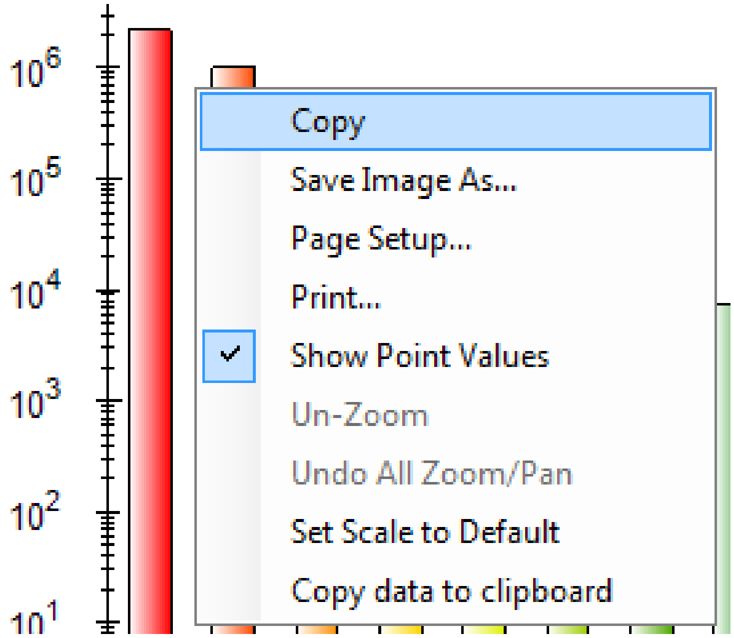


Figure 19: Bar and Pie Chart Right-Click Menu

The actions defined for each item in the menu are as follows:

- Copy: Will copy the graph as a bitmap image onto your clipboard which can then be pasted into any software that supports image pasting
- Save Image As...: Opens a saving dialog box and allows you to save the graph into a file
- Page Setup: Options for printing the graph
- Print...: Prints the graph
- Show Point Values: Show the values of the slices or bars when the mouse is positioned over a data area
- Un-Zoom: Reset to the zoom to view all the data
- Undo All Zoom/Pan: Fits the data to the available region
- Set Scale to Default: Automatically scale the axis to represent the data
- Copy Data to Clipboard: Copy the data as text to the clipboard. The data then can be pasted into Microsoft® Excel or any software that accepts text pasting.

Copying the results: The results from the properties display can be copied and pasted to other software. To copy the results, right click in the results/properties area and select **Copy All Values** or **Copy for Excel**. See Figure 20. The **Copy All Values** option will place the entire contents of the results/properties area into the clipboard. The **Copy for Excel** option takes the results which can be directly compared to the Excel GREET model.

NOTE: The Copy for Excel option is only displayed as an advanced feature (see Section 4.4.4).

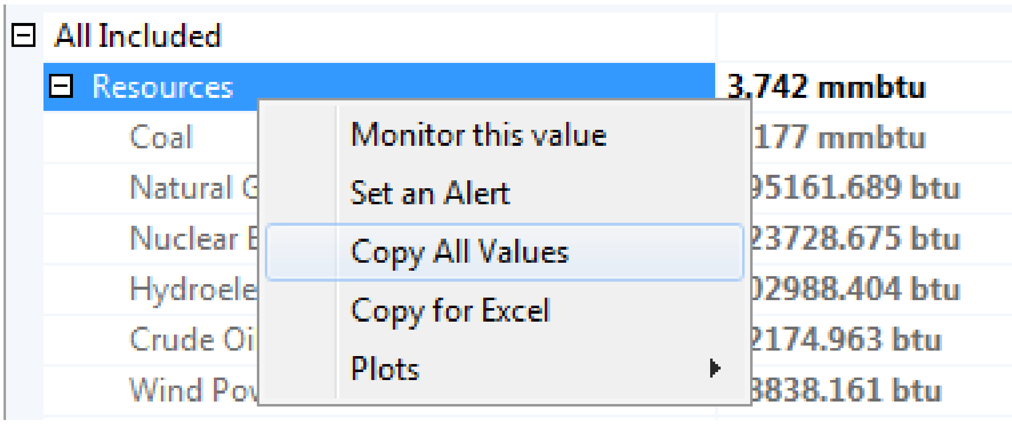


Figure 20: Copying all the Results to the Clipboard

Monitoring values: Monitoring results enable you to save and compare results between multiple simulation runs. At each run, the monitored values are also checked against their boundaries, if any are defined, to warn you if any of them are out of range. See Figure 21.

This feature can be used for two purposes. You can follow the changes of some results across multiple simulations or verify if some results stay within their limits. To monitor a result, right click on it in the properties display and select**Monitor This Value**. Now, every time the model is calculated, this value will be recorded.

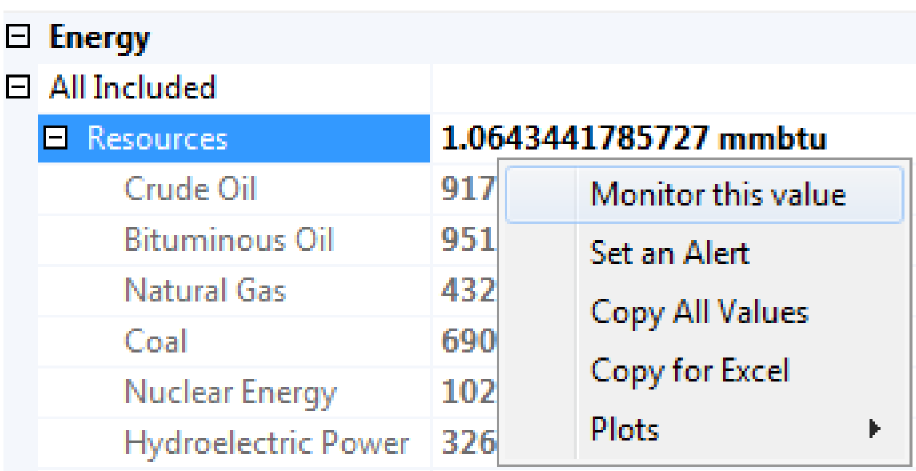


Figure 21: Monitoring the Total Energy Used

By pressing the F10 key, you will be able to see the results for multiple simulation runs as well as for the boundaries of this value. See Figure 22.

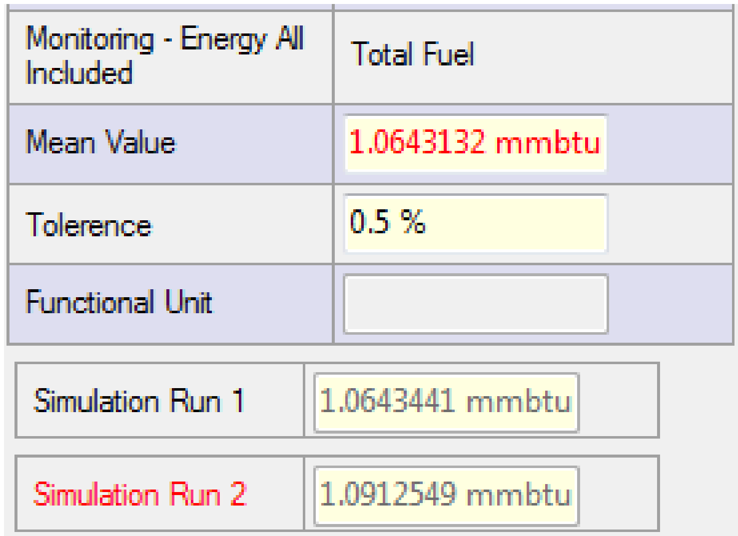


Figure 22: Results after Two Different Simulations, with a Recovery Efficiency Changed on the Second One

When boundaries are defined, a message will appear after the calculation, if one or more values are outside of their limits. Figure 23 shows the alert that appears after we modify the efficiency of the Crude Oil Recovery process. It impacts almost all of the monitored results as Crude is used as a Primary resource in many pathways.

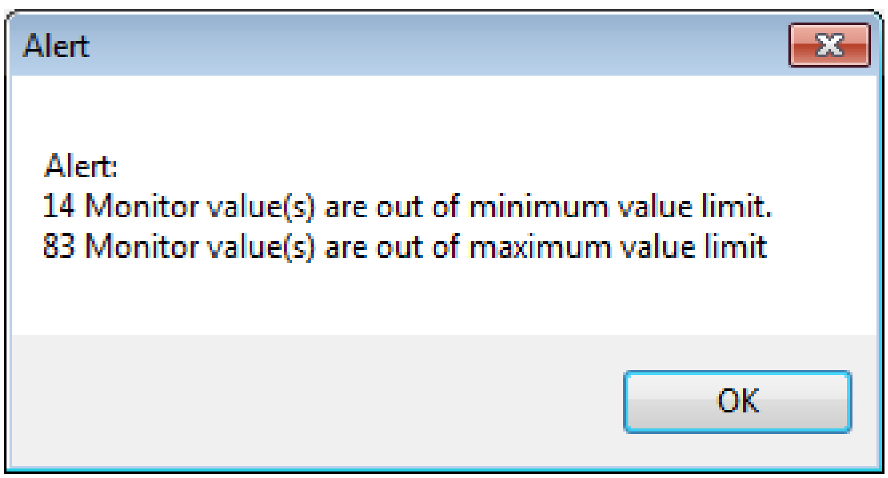


Figure 23: Alert Window

For more information about the monitored result viewer, see the GREET specific features in section 4.3.

3.6.4 Functional Unit

In Figure 16, the results for 1000 Wh are shown, i.e., 1000 Wh is a functional unit. The functional unit can be changed for each process and results can be seen per gallon or per pound as you prefer, provided that required physical properties are defined for the resource. To access the functional unit editor, click on the button showing *Per: 1000 Wh*.

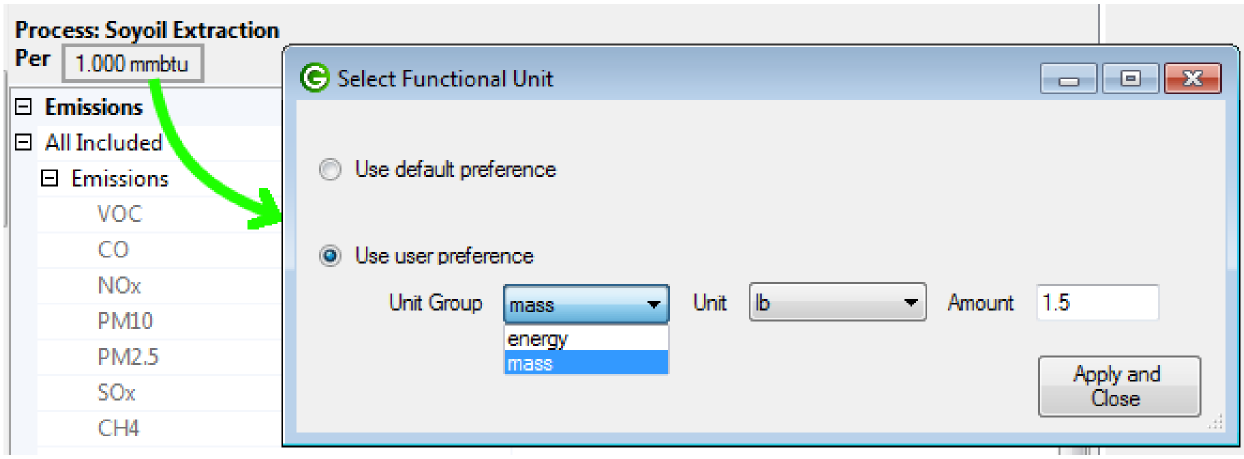


Figure 24: Changing the Functional Unit from the Default 1mmBtu to 1.5 lb

The functional unit editor opens in a new window. Depending on the physical properties available for this resource (see Resources Editor Section 3.8.1) different conversion options will be offered. In the example in Figure 24, the functional unit can be a mass or an energy unit. When mass is selected in the first **Unit Group** drop-down selector, the **Unit** drop-down selector will be updated with different mass units. By selecting **lb** as a unit and setting the amount as **1.5**, the functional unit has just been changed to 1.5 lb instead of the 1mmBtu. Now, the results for this process are shown per 1.5 lb of product. See Figure 25.

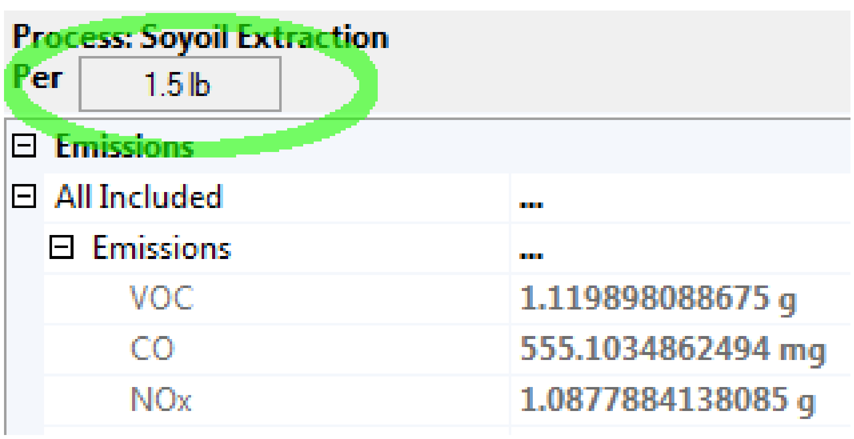


Figure 25: The Results Are Now Displayed Per 1.5 lb of Oil for This Process

Using this feature, neither are the results limited using a single functional unit, nor does the model need to be modified or recalculated.

This feature allows you to build a model of the process with the data you obtained without having to perform any prior conversions. If the data you have for a process is per 123456 Joules of output, just create your processes using this data for the inputs and outputs. However, the user interface will show all the results uniformly for 1mmBtu or the functional unit of your choice. (This assumes that enough of the physical properties are defined for the resources used.)

3.7 Well-to-Wheels (WTW)

The Well-to-Wheels main pane allows you to browse through the vehicles technologies available in the database and explore the Well-to-Wheels results. See Figure 26. The Well-to-Wheels results represent what the life-cycle impact is of a vehicle technology in terms of energy and emissions.

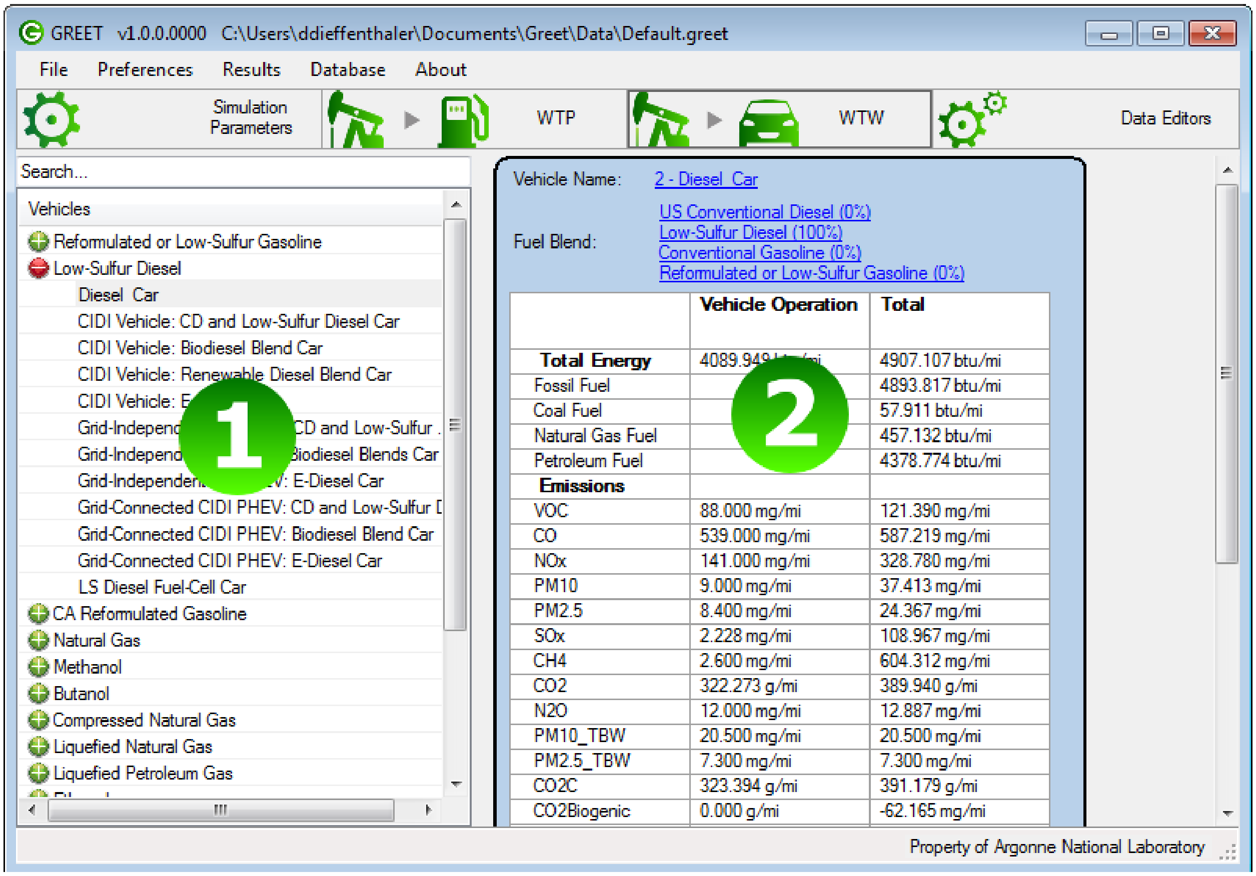
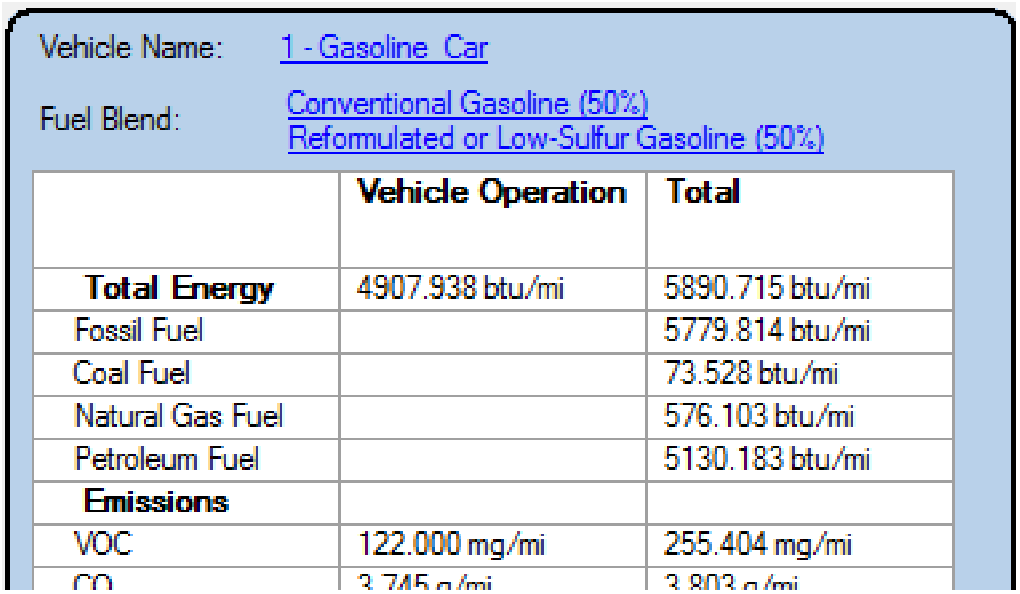


Figure 26: Well-to-Wheels Pane

The Well-to-Wheels editor is composed of two main zones. Zone 1 on the left is where vehicles can be selected. Vehicles are categorized by fuel used. If a blend of fuels is used to power the vehicle, it can be selected from any of the fuels used in the blend. Zone 2 on the right shows the results for a selected vehicle.



Vehicle Name: 1 - Gasoline Car

Fuel Blend: Conventional Gasoline (50%)
Reformulated or Low-Sulfur Gasoline (50%)

	Vehicle Operation	Total
Total Energy	4907.938 btu/mi	5890.715 btu/mi
Fossil Fuel		5779.814 btu/mi
Coal Fuel		73.528 btu/mi
Natural Gas Fuel		576.103 btu/mi
Petroleum Fuel		5130.183 btu/mi
Emissions		
VOC	122.000 mg/mi	255.404 mg/mi
CO	3.745 g/mi	3.803 g/mi

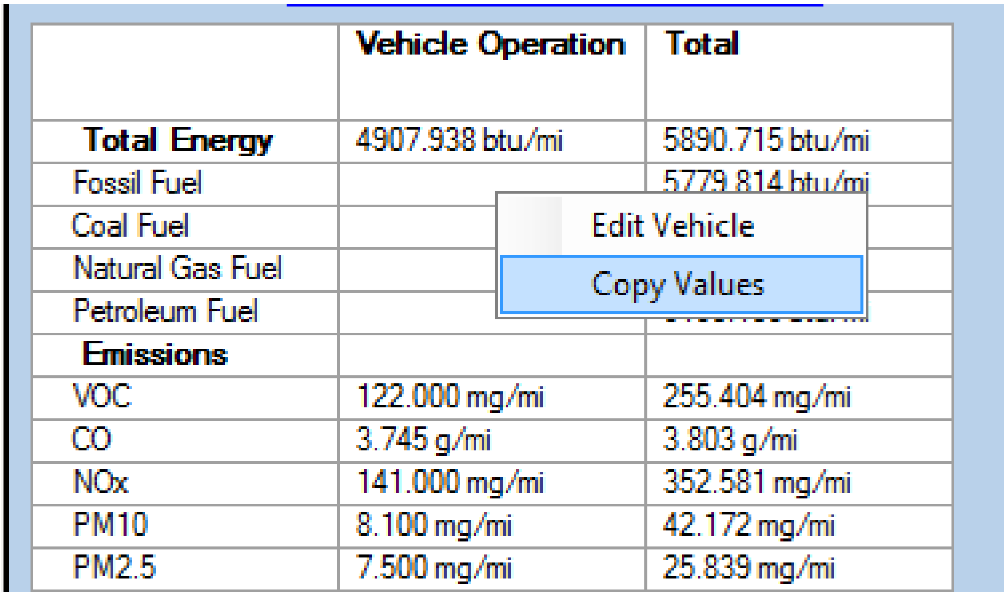
Figure 27: Well-to-Wheel Results for Conventional Gasoline Car

The Vehicle Operation column (Figure 27) shows the total energy and emissions associated with vehicle operation only; those results do not include any upstream energy or emissions. To check or edit the parameters used to calculate those values, please refer to the Vehicles Editor in section 3.8.8.

The Total column shows the vehicle operation and the upstream associated with the fuel production added together; these are the Well-to-Wheels results.

Clicking one of the fuel links will open the pathway or the Pathway Mix that was used to produce that fuel.

All the results for a vehicle can be copied for use in other software. To copy the results, right click on the results table and select **Copy**. See Figure 28.



	Vehicle Operation	Total
Total Energy	4907.938 btu/mi	5890.715 btu/mi
Fossil Fuel		5779.814 btu/mi
Coal Fuel		[illegible]
Natural Gas Fuel		[illegible]
Petroleum Fuel		[illegible]
Emissions		
VOC	122.000 mg/mi	255.404 mg/mi
CO	3.745 g/mi	3.803 g/mi
NOx	141.000 mg/mi	352.581 mg/mi
PM10	8.100 mg/mi	42.172 mg/mi
PM2.5	7.500 mg/mi	25.839 mg/mi

Figure 28: Copying the Results of a Vehicle

In order to access the vehicle parameters one can click the **Edit Vehicle** item after a right click on the results.

3.8 Data Editor

3.8.1 Resources Editor

The resource editor allows you to add new resources or edit existing ones. To access it click on the **Data Editor** button of the main pane selector, then open the resources, and either click **Add Resource** or **Modify Resource**.

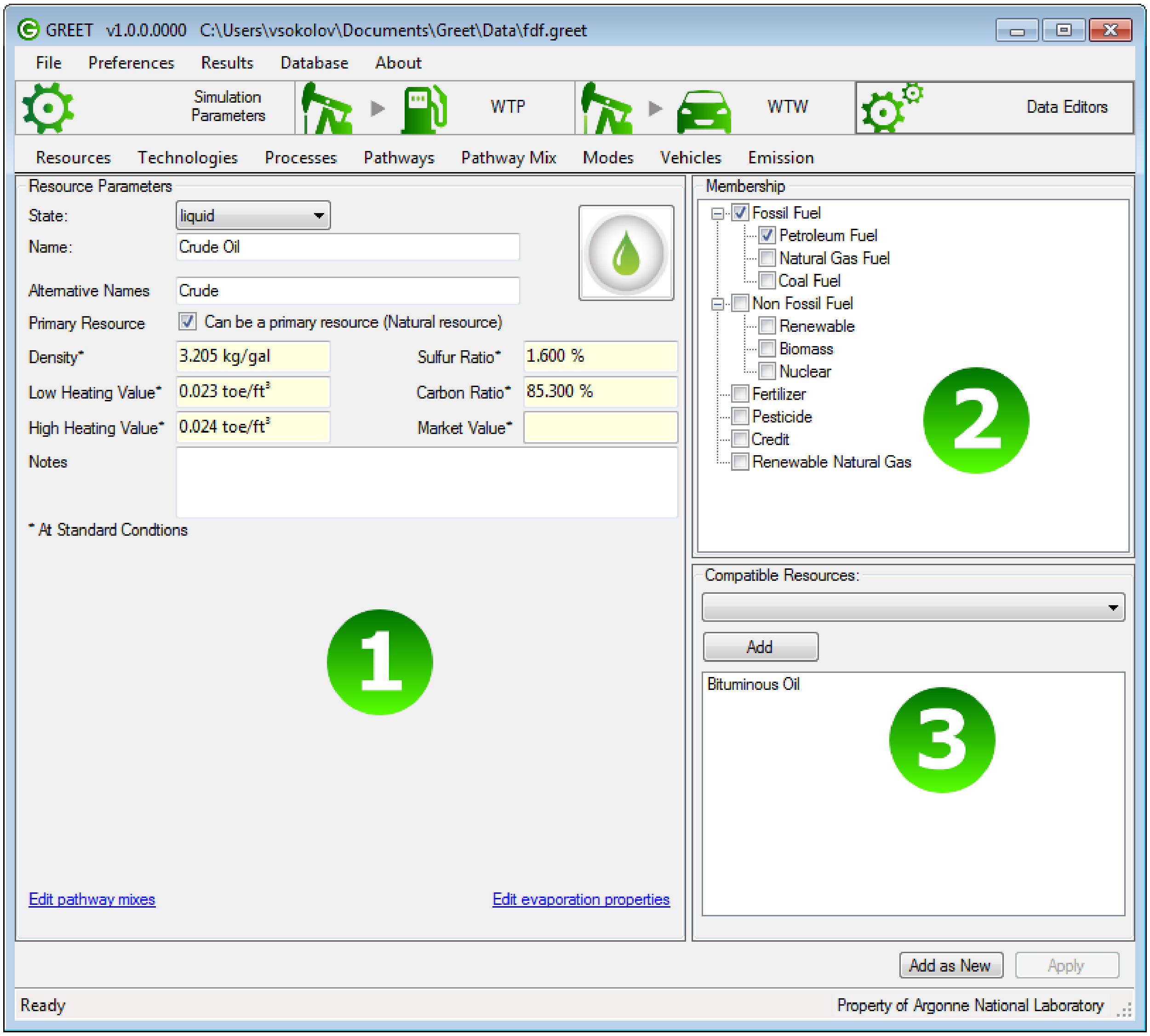


Figure 29: Crude Oil in the Resource Editor

Figure 29 shows the resource editor loaded with the Crude Oil resource. The editor is made up of three zones. Zone 1 represents the properties of the resources such as the physical state, name, heating values, density, and a picture to associate with the resource. Zone 2 represents the group memberships for this resource; this control only appears if we are editing a primary resource. The last zone, marked number 3, shows the list of resources that can be blended with the edited resource.

NOTE: Depending on whether the resource can be a primary resource, Zone 2 can be hidden.

Resource parameters: The state of the material defines its physical state for the given properties. It is mostly used for calculating the energy intensities for some transportation modes and for setting up the right units for the parameters below.

The alternative names allow you to add more names for the same material. An example would be natural gas which is abbreviated as NG. All the abbreviated names should be separated by commas. The alternative names can be used in the search boxes in the software to provide easier access to the database for those used to a different nomenclature.

A picture can be chosen for the resource. To select a picture, click on the picture selector on the right side of the name input box. A picture selector will open. It allows you to select the appropriate picture from the database or to load a new file from your hard drive. Formats such as png, jpg, and bmp are supported.

The **Primary Resource** check box defines if we can assume that this resource is coming from our environment without any upstream and can be used in the process as a resource from well (without any upstream). A few examples of primary resources are crude oil, natural gas, wind, and sunlight; counter examples would be soy oil, compressed natural gas, or gasoline. This parameter decides if the group memberships panel will show or not.

Density, low- and high-heating values, sulfur ratio, carbon ratio, and market value are all used for calculating the results. All of those properties are optional. Physical properties are used in several cases: dimensions normalization (conversion between mass/volume/energy), calculation of co-product allocation ratios, and sulfur and carbon balancing [1].

Edit Pathway Mixes: Further down the link, Edit Pathway Mixes for this resource provides you with access to the Pathway Mix. This is described in more detail in the Pathway Mix section 3.8.6.

Copy properties: Copy properties from another resource will prompt you with a dialog box offering many other resources. By selecting one of them, you will copy all of the physical properties of this resource into the current one.

Evaporation parameters: The Edit evaporation properties link is quite important for the losses. Each time losses are defined in the model, they are defined as a percentage of the quantity displayed. That lost amount might then be converted into some pollutants when it evaporates.

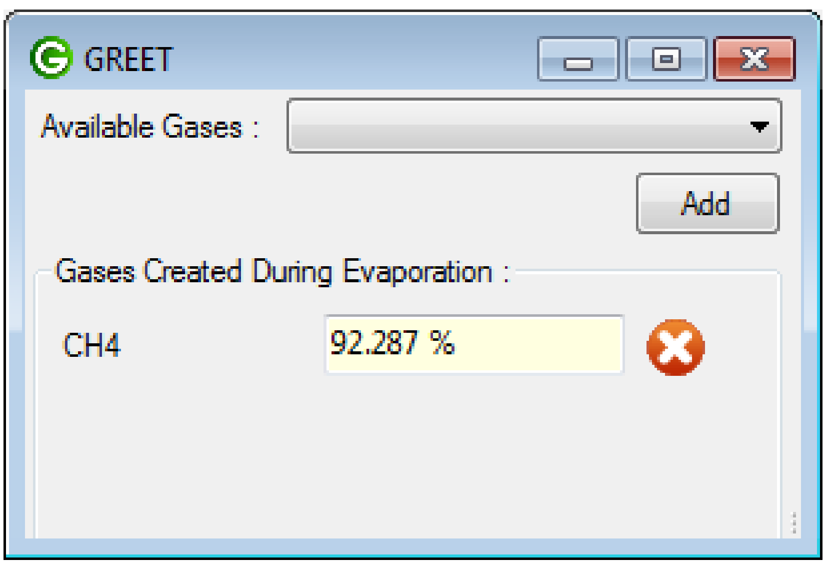


Figure 30: Natural Gas Evaporation Gases Parameters

Figure 30 shows the evaporation parameters for the natural gas resource. To add other gases to the list of emissions related to the evaporation of that resource, you have to select a gas in the drop-down selector and click the **Add** button; a new gas line will be created and you can define the share for that new gas emission. To remove an existing emission, click on the icon.

Group membership: When the primary resource box is checked, the right side of the control shows the memberships (Zone 2 in Figure 29). Memberships allow you to categorize the resource into different groups. Those groups will be used to aggregate results together in the Well-to-Pump main pane as shown in Figure 31 or in the Well-to-Wheels main pane.

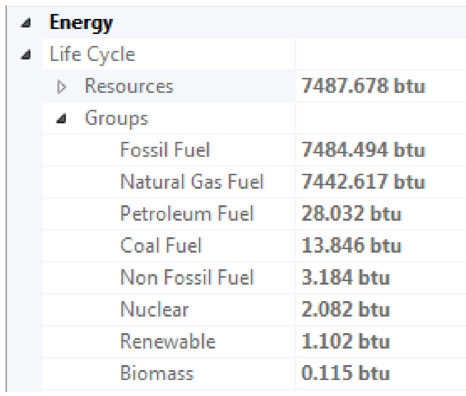


Figure 31: Energy Groups in the Property Display of the Well-to-Pump Main Pane

To assign a resource to a group, check one of the existing groups in the tree representation. To see what are the actual members of a group, right click on it and select **Current Members**. The members of that group will be shown as seen in Figure 32.

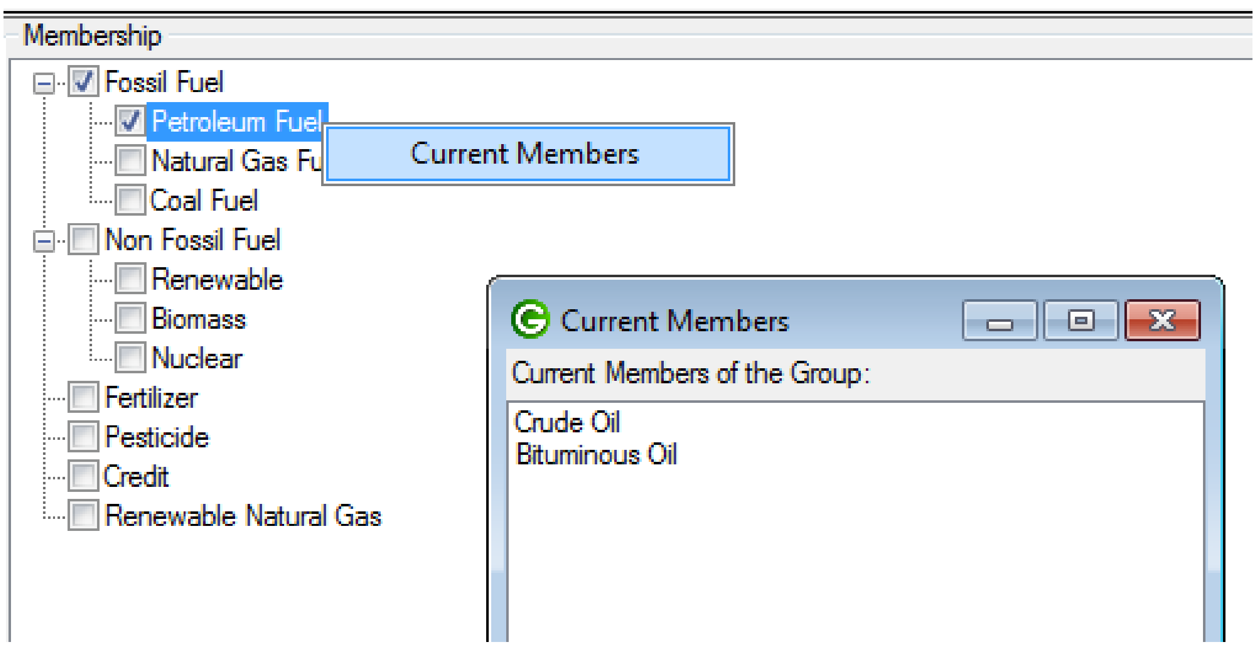


Figure 32: Display Members of a Resource Group

3.8.2 Technologies Editor

Modifying an existing technology: **Technologies** in GREET can be modified by clicking the Data Editor main pane. After opening the **Technologies** menu, click on **Modify Technology**.

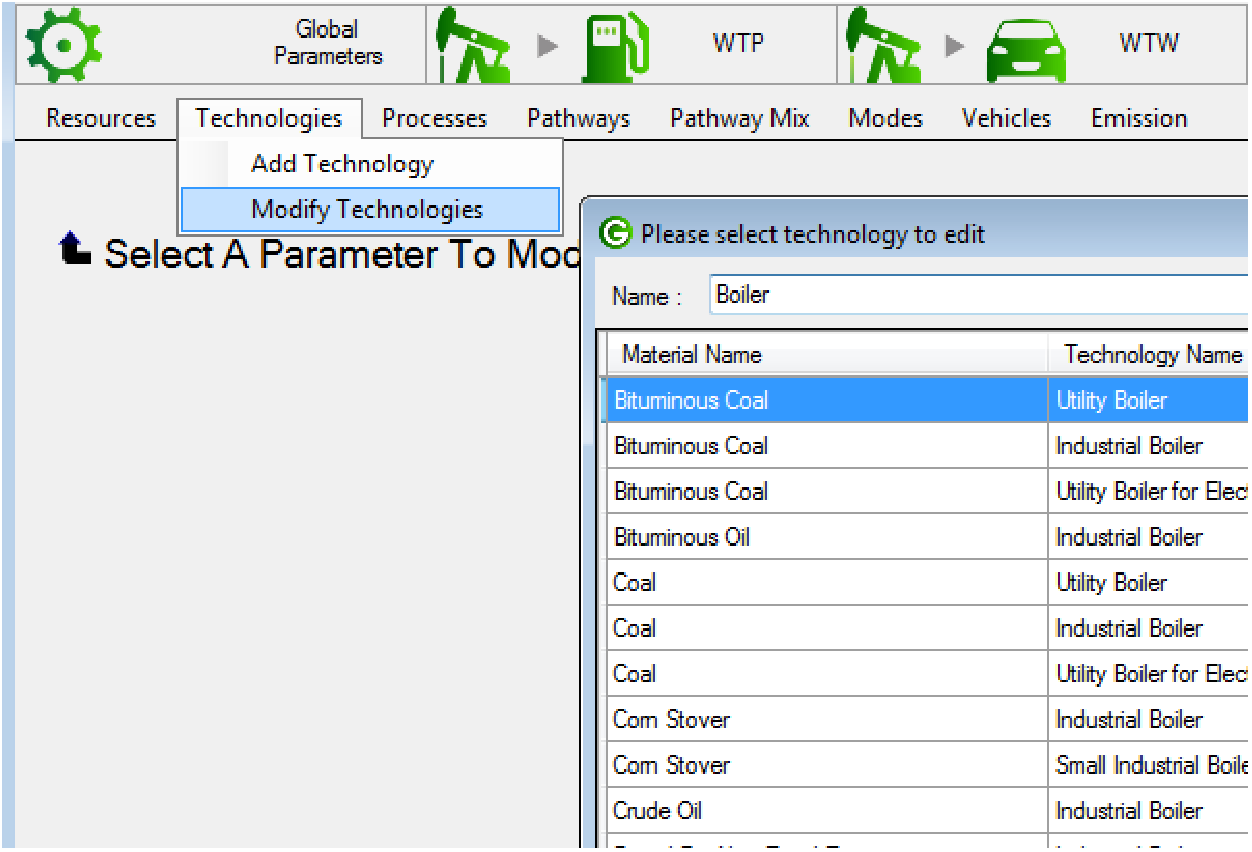


Figure 33: Select the Technology to Edit

The pop-up window will display a list of technologies that can be modified. They are organized by Resource Name and Technology Name. Resource name represents the resource that is being used by the technology.

To help you find the desired technology, this form has a search box that looks through the technology and resource names. See Figure 33. Once the desired technology is found, double-click on it to see the emission factors.

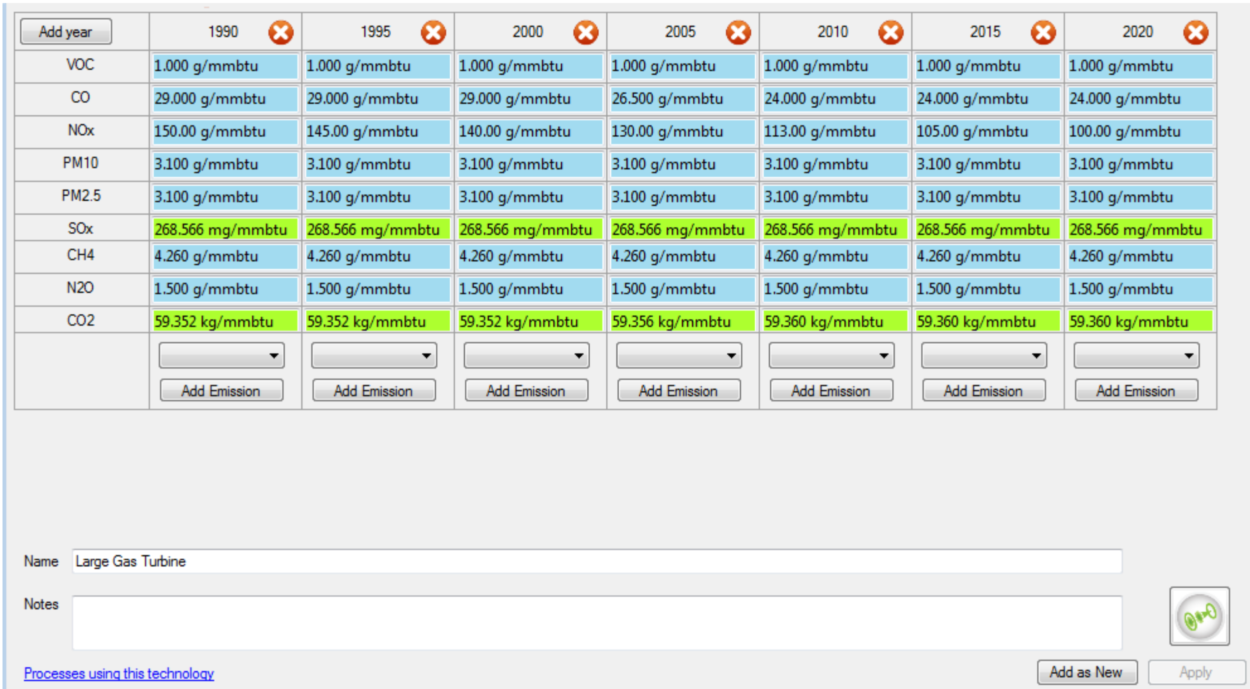


Figure 34: Technology Editor Showing Emission Factors and Properties

The screenshot in Figure 34 shows the technology editor loaded with a Large Gas Turbine for natural gas technology.

Technology parameters: At the bottom of the emission factors, a name and notes can be defined for the technology. A picture can be chosen for the technology. To select a picture, click on the picture selector at the right side of the notes input box. A picture selector will open that allows you to select the appropriate picture from the database or load a new file from your hard drive. Formats such as png, jpg, and bmp are supported.

Technology emission factors: At the top area, the emission factors can be seen for each year. Recording emission factors for different years allows you to keep track of changing emissions as technologies are refined and developed. They can all be modified and they use the same color coding as the values in the input tables. However, we have one more color code here: green background. The green background means that the emission factor displayed is balanced, which means that it has been calculated from the physical properties of the resource and the other emissions factors. See Figure 35. For example, the CO_2 emission factors in the natural gas large turbine are calculated using the carbon content of natural gas (cf. Carbon Ratio in the Resources Editor 3.8.1) subtracted by the carbon content of the VOC, CO, and CH_4 gases and divided by the carbon ratio of the CO_2 gas (cf. Carbon Ratio in the Emissions Editor 3.8.9). To have a value calculated using a balance, the user can right click the value and select **Calculated**. This menu item will then appear as checked, which means this emission factor is now calculated using a balancing equation [1].

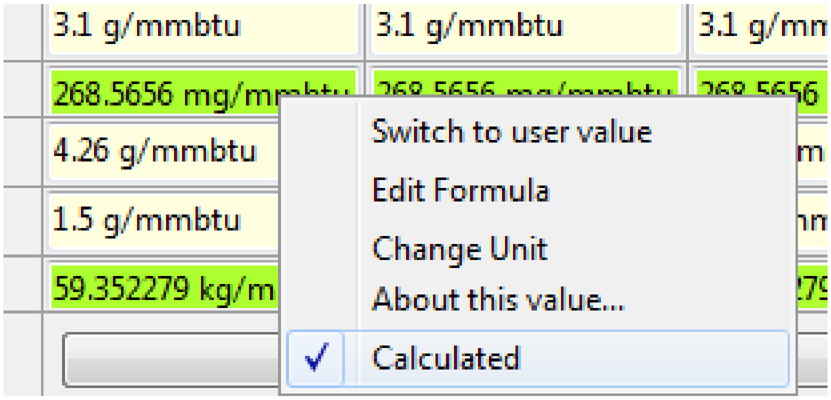


Figure 35: Green Background Indicates Emission Factors Are Balanced

NOTE: Calculating emission factors by Carbon or Sulfur balance is available for CO_2 and SO_x gases.

To define another pollutant for the technology, you can select a gas in the drop-down selector at the bottom of a year column, and click the **Add Emission** button. It will generate a new row that you will have to populate with your emission factors.

Once all the modifications are done, click the **Apply Changes** button to keep your changes in the database.

If you would like to see a list of processes that use the technology, click on the Processes using this technology link at the bottom of the technology editor.

Adding a new technology: Technologies in GREET can be added by clicking the **Data Editor** main pane, opening the Technologies menu, and clicking **Add Technology**. See Figure 36. You will be asked to select the fuel or resource which is used by the technology. For example, if you need to create a new natural gas boiler, find natural gas in that list and double click it to create this new boiler technology.

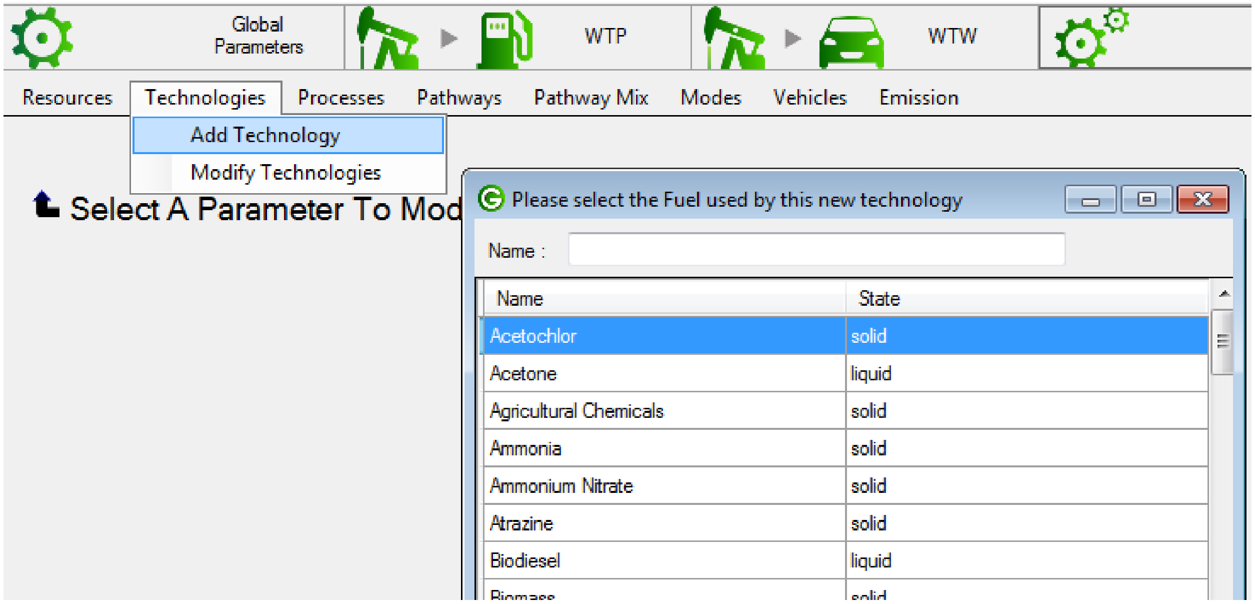


Figure 36: Selecting the Fuel for the New Technology

The new technology will appear as blank. You will have to add emission factors for each gas, and if you wish, time series values can be entered for multiple years. To add emission gases for a certain year, press the **Add Year** button; you will be prompted to enter the year number. See Figure 37. If your emission factors are not year dependent, 0 can be entered there and it will be used for any selected simulation year. Otherwise, the model will always try to take the closest emissions factors available; if the desired simulation year is 2012 and emissions factors are available for 2000 and 2015, the selected values will be the ones from 2015.

Once the year is added, enter your emission factors in the yellow cells; the green cells' emission factors are automatically balanced as explained in the previous paragraph.

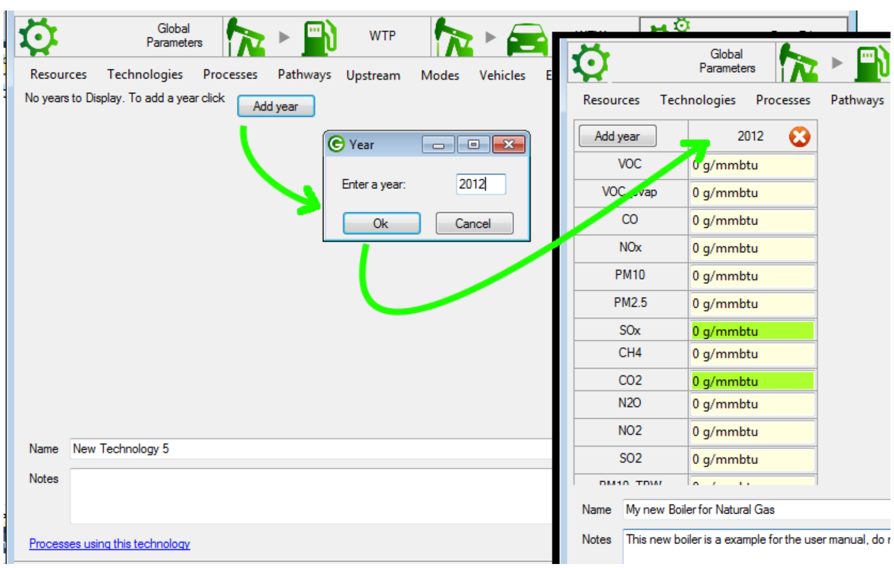


Figure 37: Adding Emissions Factors for Specific Years in the Technology Editor

Finding which processes are using this technology: On the bottom of the control there is a link - Processes using this technology. Clicking on it will open a new window that shows you all the processes in the database that are using this technology. This information can be used to make sure we are looking at the technology we wanted by checking the processes that are using it.

3.8.3 Stationary Process Editor

The stationary process editor allows you to make changes to or create new stationary processes. It is accessible by clicking the **Data Editor** main pane button. When opening the Processes menu, you can choose to Add Stationary Process or Modify Process.

Creating a new process: After clicking **Add Stationary Process** an empty model of a process will be loaded into the editor. See Figure 38.

The global layout used when a stationary process is opened in the editor is:

- Main Output on the top right
- Coproducts on the right, below the main output
- Inputs from groups in the top left
- Other inputs defined by amount on the bottom left
- Other emissions on the right of the other inputs

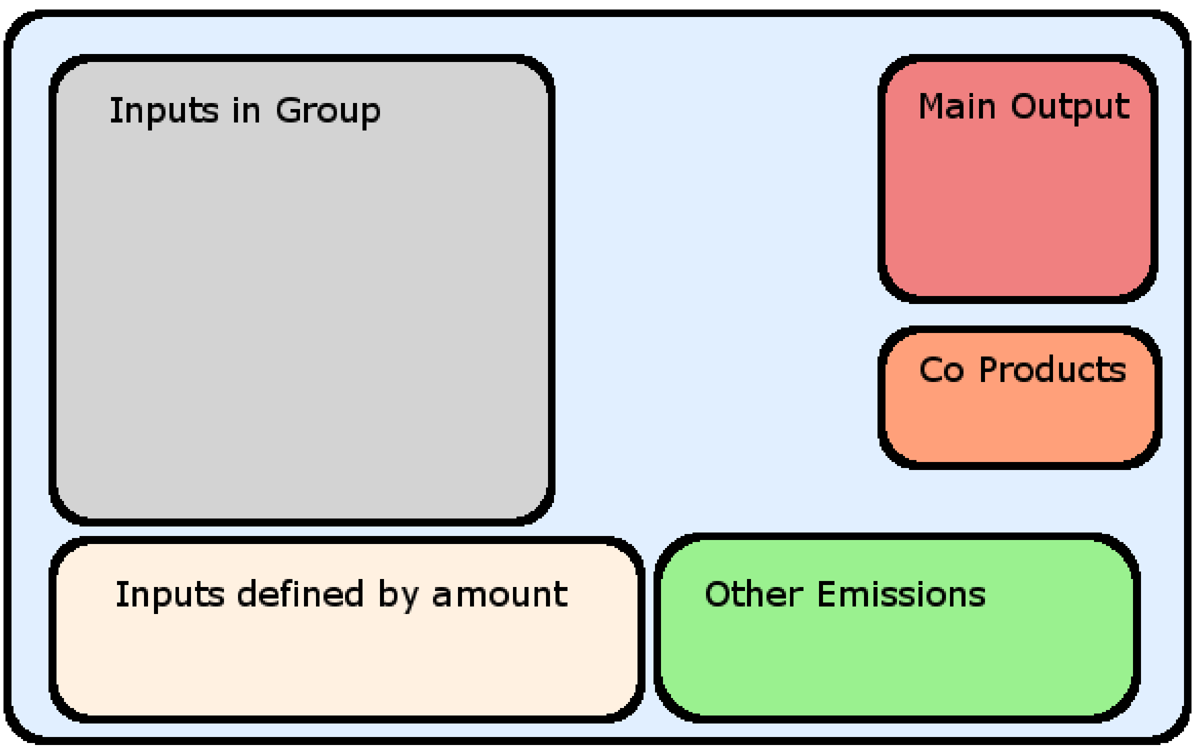


Figure 38: Layout of a Stationary Process in the Editor

Defining the main output: To add a main output to a process which does not have one already, click on the **Resources** button on the bottom left of the editor. The list above will be populated with all the resources defined in the model. Then drag and drop the desired resource into the area reserved for the main output in the stationary process layout.

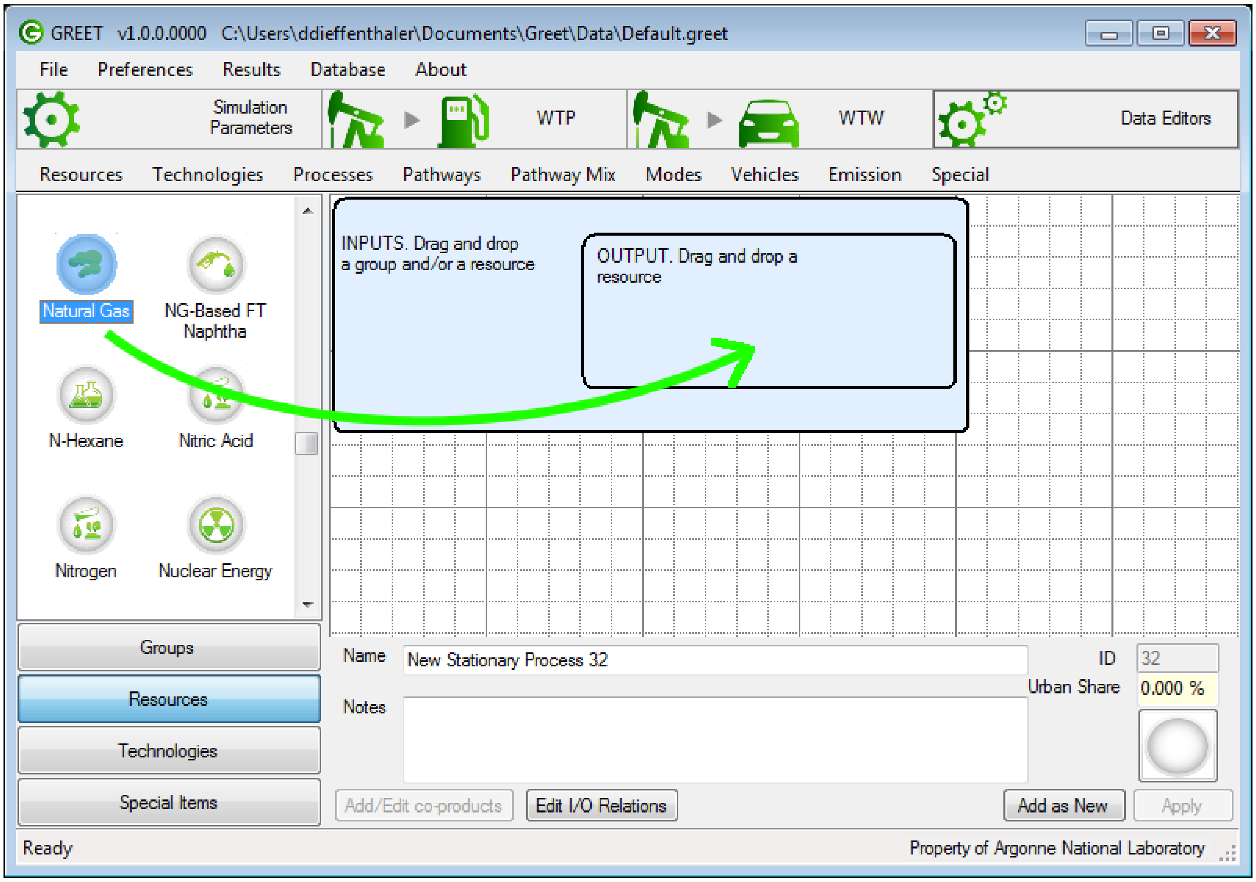


Figure 39: Drag and Drop a Resource for the Main Output

To replace an existing main output, you can delete the main output by right clicking on it and selecting Delete I/O and then following the procedure above to add a new main output, see Figure 39.

Adding inputs: The input needs to be defined per amount of output. This means that the inputs defined represent the necessary resources to produce the specified amount of Main Output and the Co-products if any.

There are two ways to specify the amount of an input. The first and easier way is to define an amount. To do that, click on the **Resources** category button on the bottom left. The list above is getting populated with all of the resources defined in the model. Then drag and drop the resource into the process; new inputs are going to be created automatically. Then, define the amount by entering a value in the Input box.

You can see that **Source** and **Pathway** or **Mix** are automatically selected. We will come back to these parameters later, but they define the origin of the input and associated upstream to produce the resource used by an input. By default, the selected values are the ones that are used the most frequently in GREET.

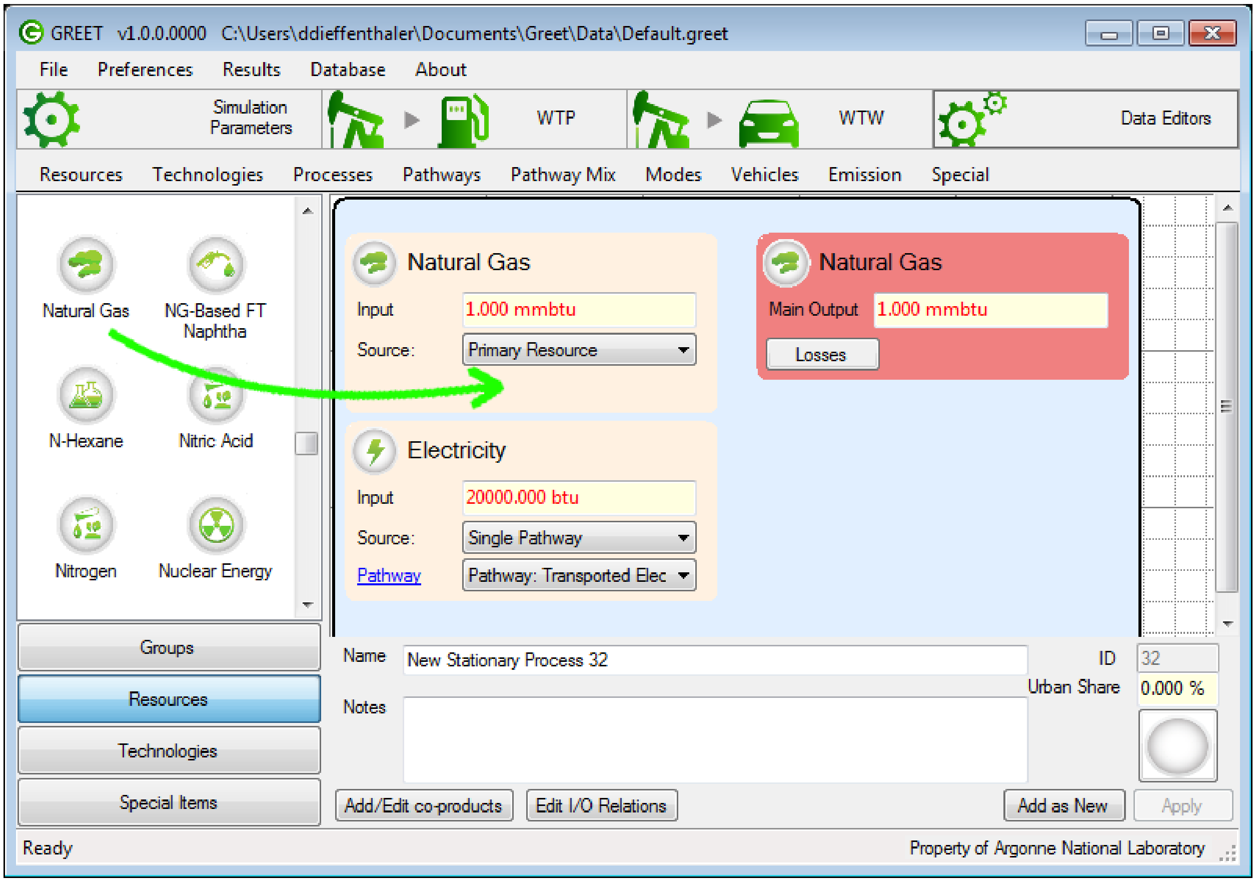


Figure 40: Drag and Drop a Resource as an Input for the Process

The second way is to create a group and place one or more inputs into that group by dragging and dropping them from the resources list and assigning them a share, see Figure 40.

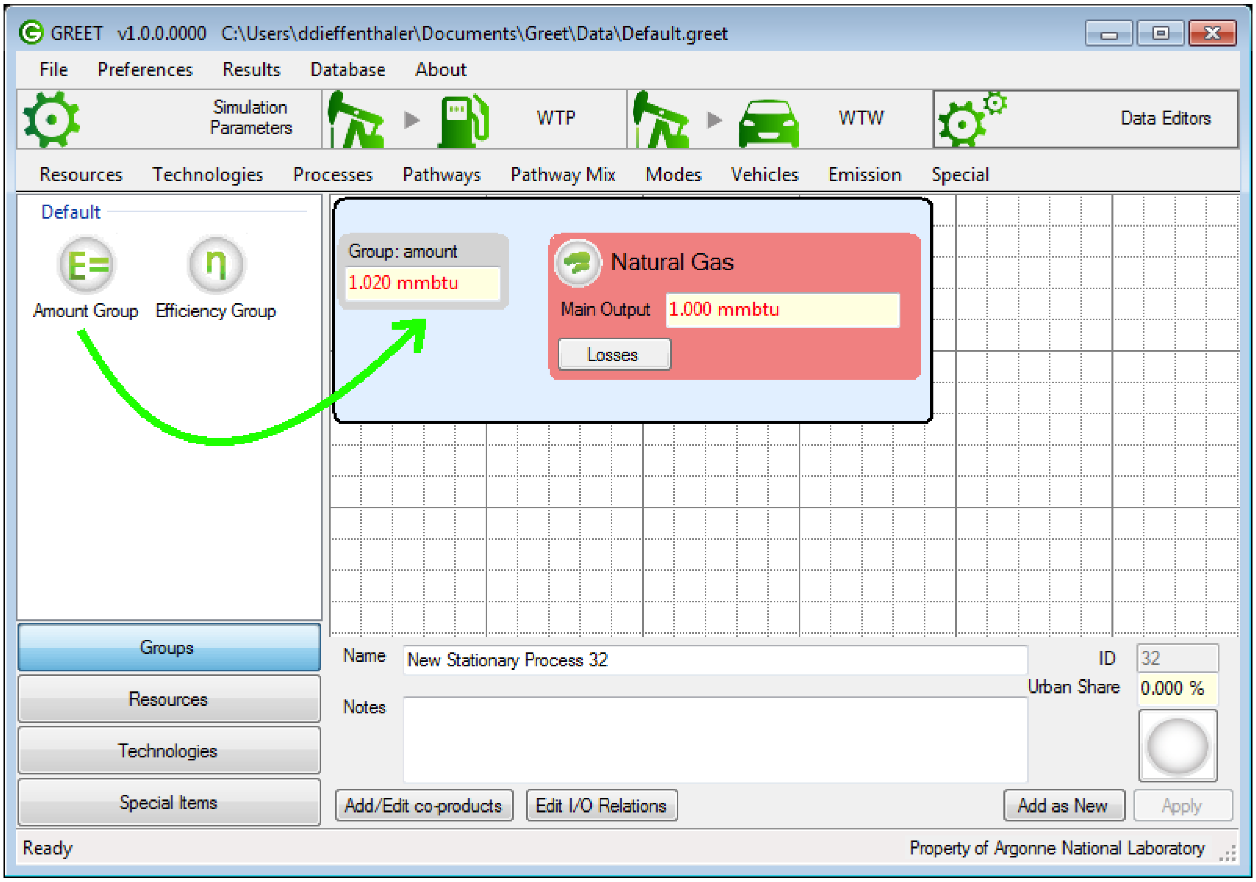


Figure 41: We First Drag and Drop an Amount Group

Figures 41 and 42 show the two steps that need to be done to add inputs to a group. In the case presented in Figure 42, we are using an amount group; define that group as using 1,020,000 Btus and share that amount among the two inputs.

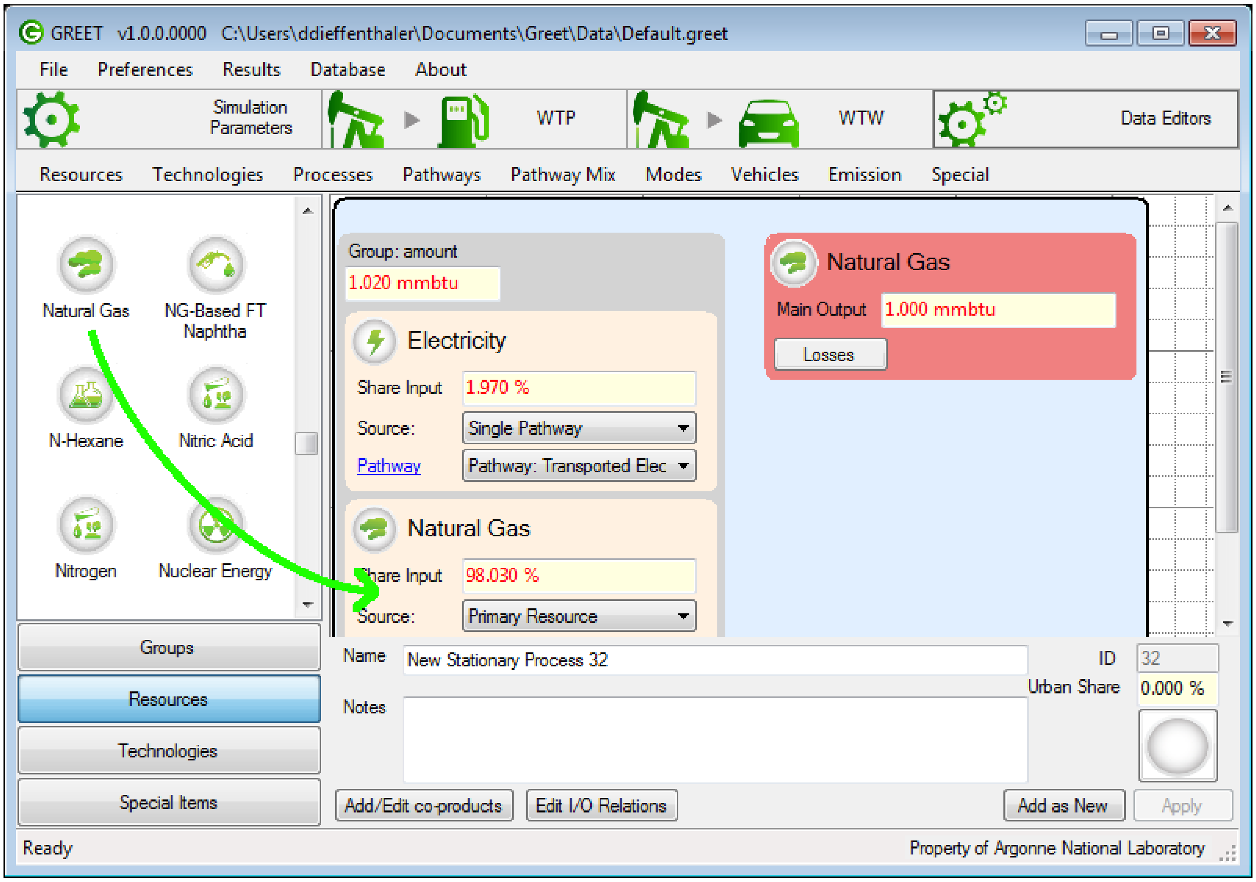


Figure 42: Inputs Shares Are Added to a Group

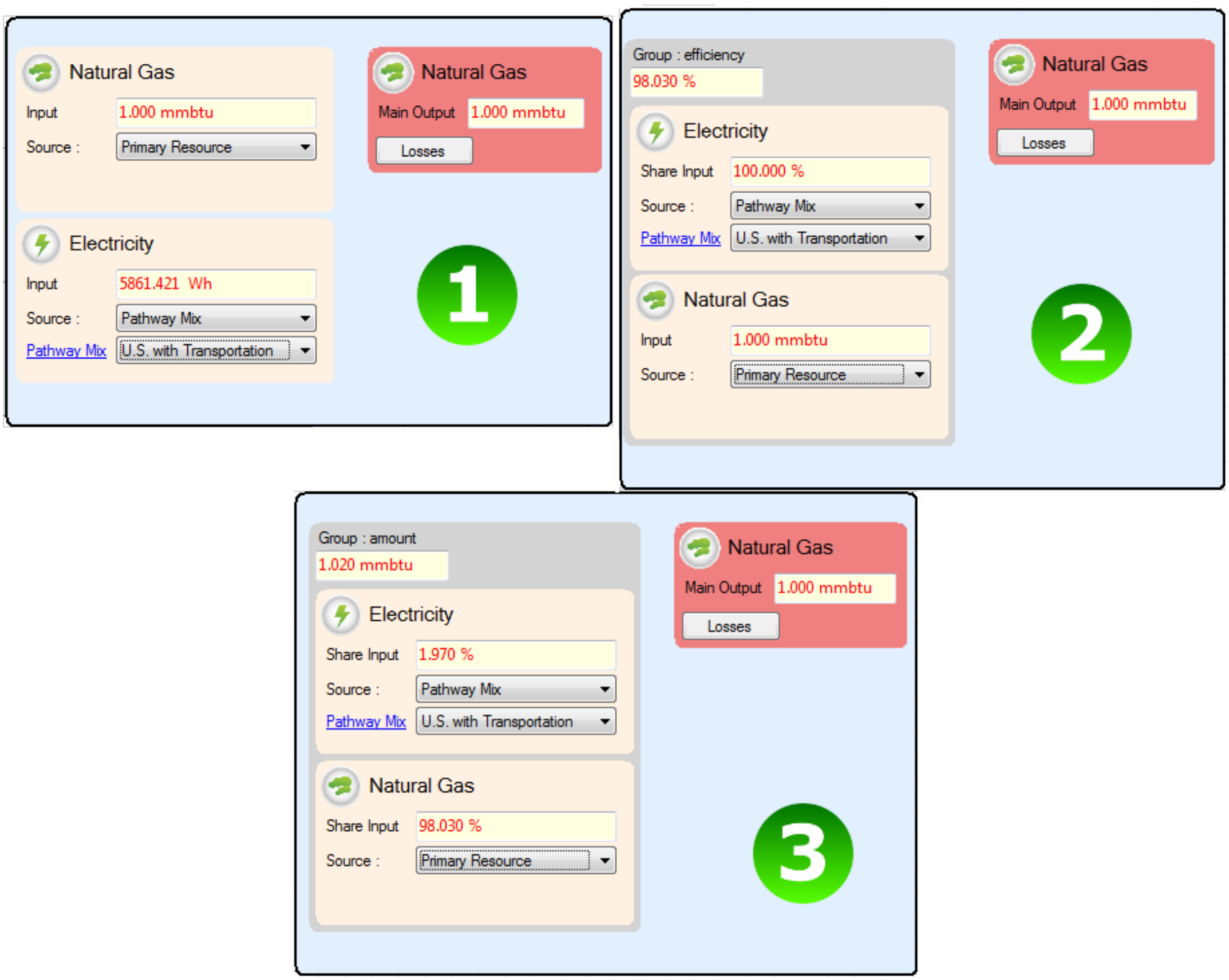


Figure 43: Three Equivalent Stationary Processes

Three processes shown in Figure 43 are **identical**. All three will be converted to the same canonical form before calculating the results. For more details see [1].

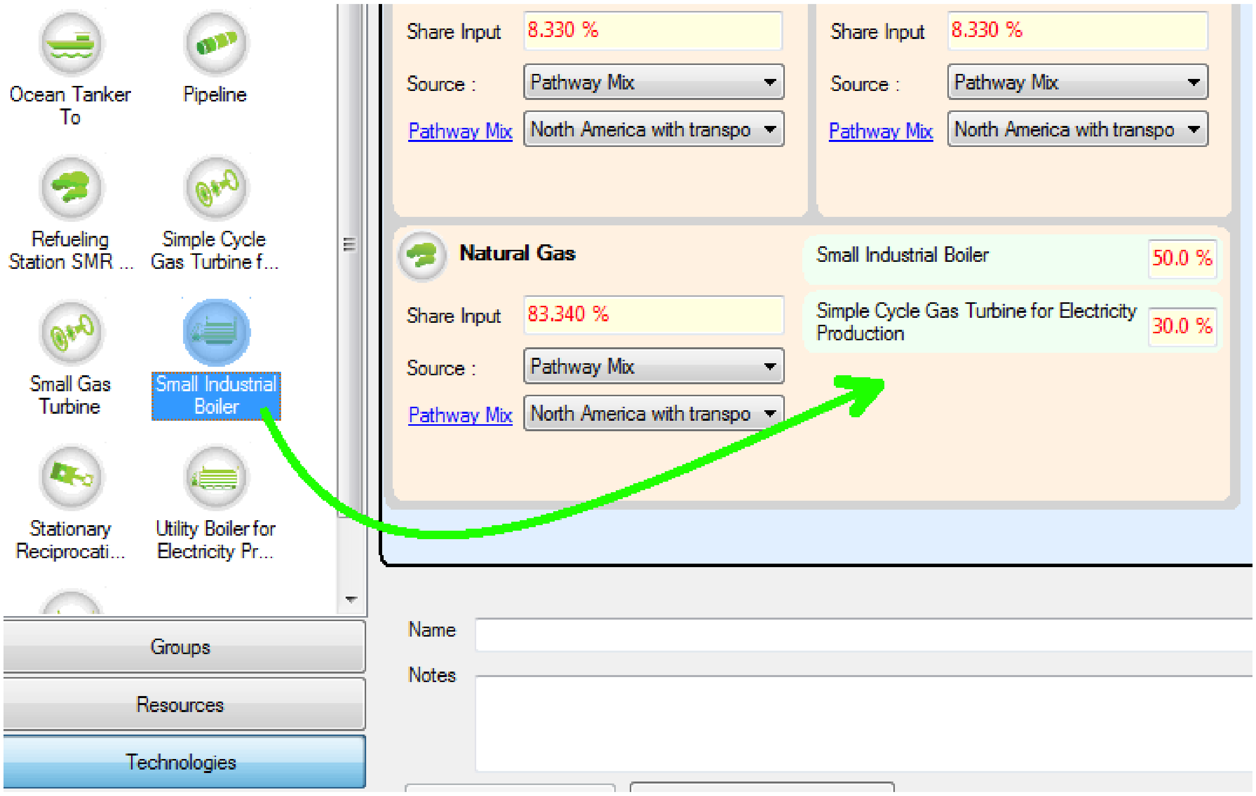


Figure 44: Dropping a Technology Over an Input and Assigning Shares to It

Adding technologies: Technologies are added to a stationary process by dragging and dropping an existing technology over the inputs and assigning a share to it. To do so, click on the **Technologies** button in the bottom left corner, and use one of the available technologies for one of the inputs present in the process. Figure 44 shows dragging and dropping a utility boiler and a simple cycle gas turbine onto a natural gas input.

A technology can be deleted by right clicking it and selecting **Delete Technology**, see Figure 45.

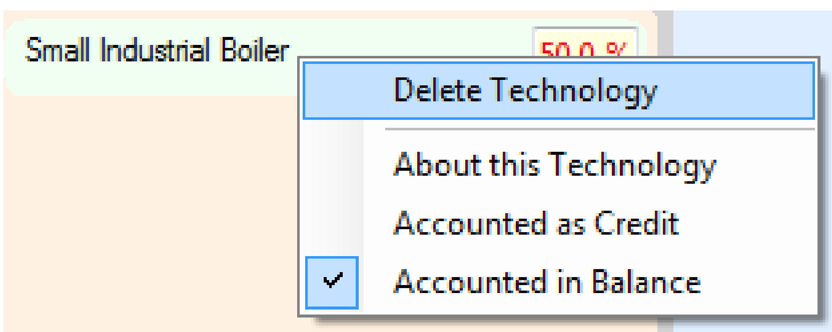


Figure 45: Technology Right Click Menu

About this Technology option allows you to enter some notes for the technology.

If the **Accounted as Credit** item is checked, all the emission values from this technology will be accounted as negative for the process.

If the **Accounted in Balance** item is checked, the emissions from this technology will be used in the emissions of the process. If not checked, the technology is not used in the calculations.

Coproducts: To specify a coproduct, first add an input, then right click on this input and select **Set as CoProduct**. See Figure 46.

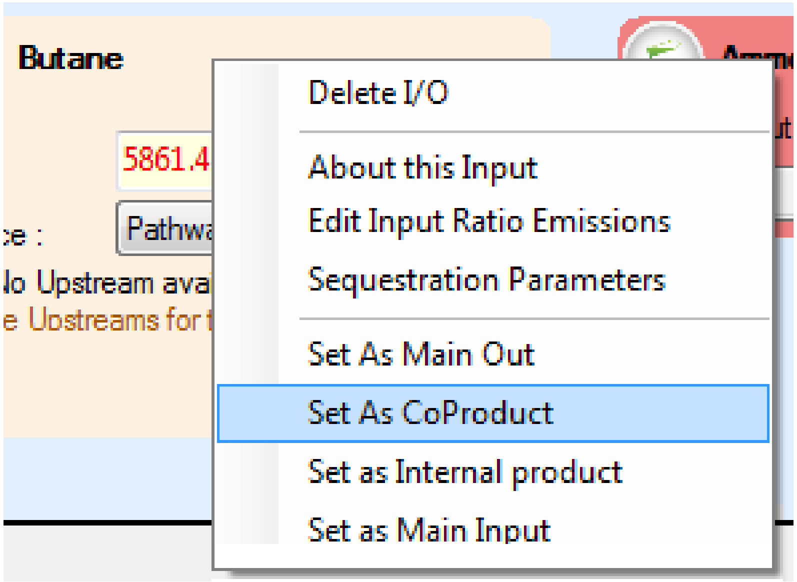


Figure 46: Input Right Click Menu

The resource will then appear as a coproduct in the editor, and the **Add/Edit coproducts** button will be enabled. The button stays disabled until any coproducts are defined in the editor. See Figure 47.

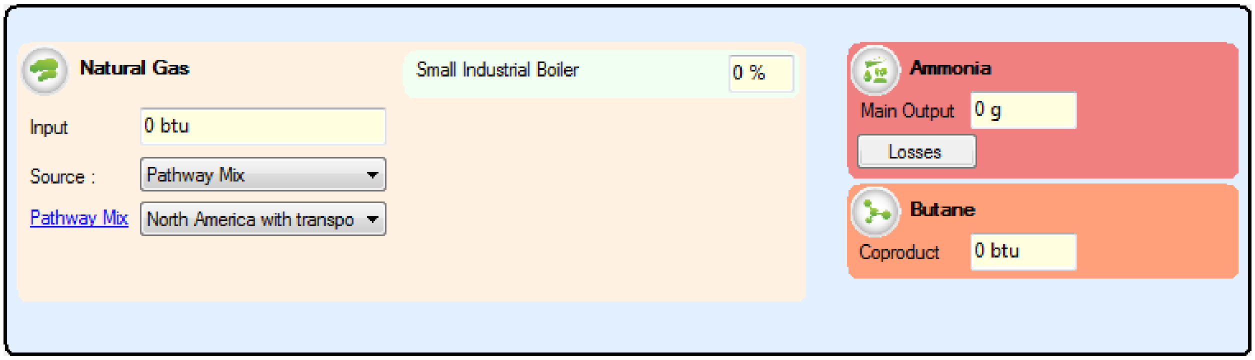


Figure 47: The Input Has Now Been Converted to a Coproduct

Parameters for coproducts are accessed by clicking the button Add/Edit coproducts at the bottom of the editor; the pop-up window appears as shown in Figure 48.

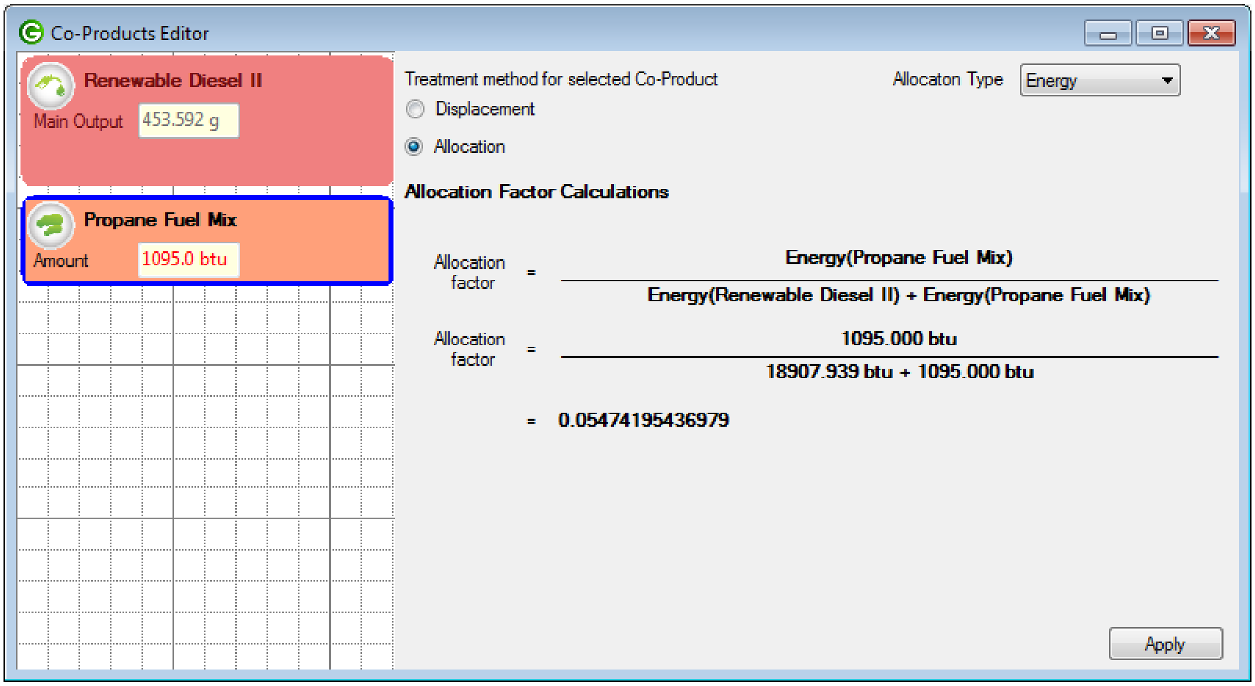


Figure 48: Coproducts Editor

The treatment method can be Displacement or Allocation. Both of these are calculated at the process level, and we do not yet offer pathway level treatment methods.

If the allocation method is chosen as shown on Figure 48 the allocation type can be chosen. The allowed allocation method available will vary according to the physical properties available for all the output materials. In the case presented above, the main output is designed to be a quantity in grams, the coproduct quantity is expressed in Btus. To perform an energy-based allocation, the heating values of the Renewable Diesel II are necessary to calculate the allocation factor.

If the displacement method is selected, the displaced materials and upstreams have to be defined. In Figure 49, Propane is displacing the Pathway Mix Petroleum Gas from North America selected as an upstream for Propane. Multiple displaced resources and upstreams can be added and weight averaged using shares for each of them. A displaced resource can be added by clicking the **Add displacement resource** button and an existing one can be removed by clicking ⊗.

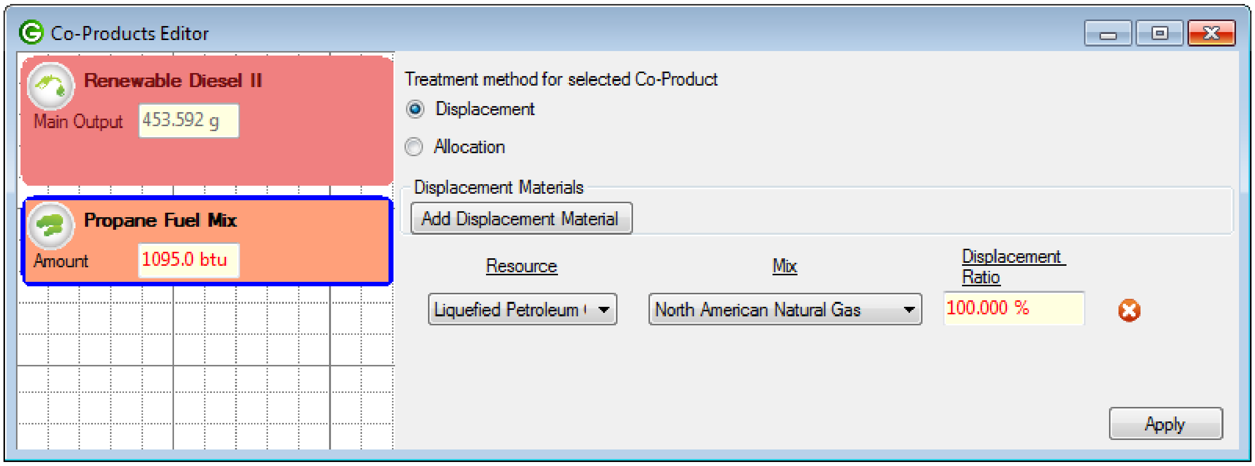


Figure 49: Coproducts Editor with Propane Displacing LPG

Emissions ratios for inputs: This option allows you to specify emissions that are not technology related. For example, when nitrogen is used as a fertilizer, it vaporizes into some gases.

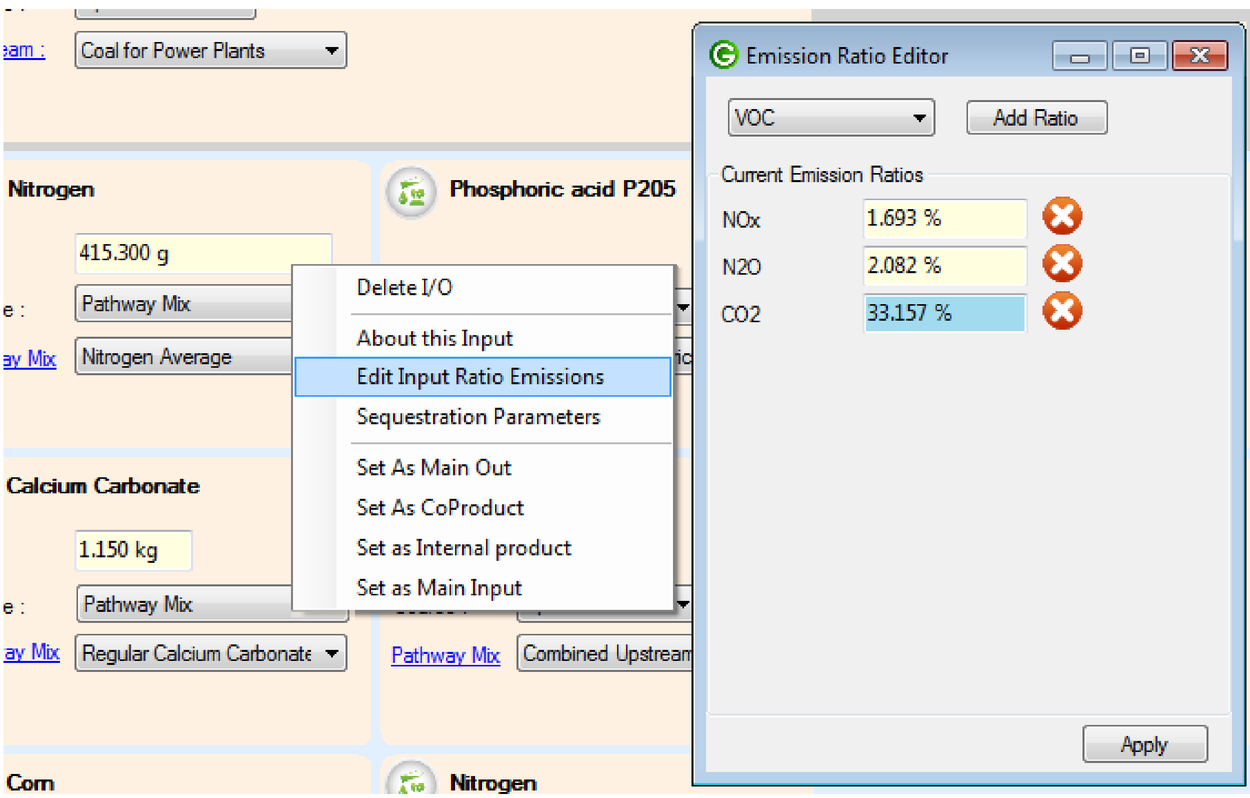


Figure 50: Input Emission Ratios

In Figure 50, it is defined that 1.6% of mass for the Nitrogen input creates NO_x gas, 2.1% creates N_2O gas, and 33.2% creates CO_2 gas. The emissions created from these Emission Ratios will be reported in the Other Emissions section of the Well-to-Pump results (see Section 3.6.3).

The percentages in the Ratio Emissions editor are always defined on a mass basis. Again, the amount for this input does not have to be specified as mass. Physical properties of the material will be used to do the conversion from other dimension (volume or energy) to mass. If the conversion cannot be done, the ratio emissions will be 0 for that input.

I/O relations: This feature allows to specify relations between inputs and outputs. The relations are specified as weights. For example, the user can specify how much "chemically" of inputs are contained in the main output.

As a simple example, an Ethanol plant uses many resources as inputs. However, Ethanol is a result of Corn fermentation. Other resources are used for heat generation, or power the different systems in the plant. In such cases, we are going to define a 1:1 ratio between the Ethanol output and Corn.

This parameter is also very important for Biogenic credits. Whenever ethanol is combusted (in a vehicle or any other technology), we will be allowed to trace the upstream resource used to produce Ethanol. When biomass is used as feedstock all the carbon produced during the combustion of ethanol will be accounted as CO_2 biogenic. In the model it is accounted is a credit for any process and should be seen as a negative value.

By default a 1:1 ratio is assigned for the inputs for which the source is Previous or Primary Resource.

Carbon sequestration: CO_2 sequestration is available for all inputs of a process. In order to define sequestration parameters, right click an input and select **Sequestration Parameters** in the context menu.

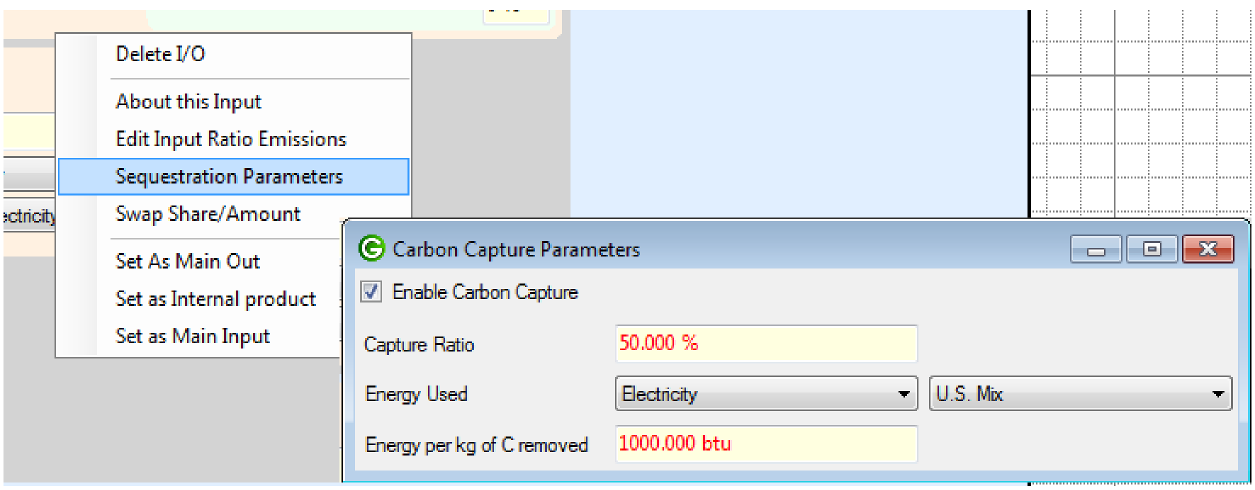


Figure 51: Carbon Sequestration Parameters Accessed from the Stationary Process Editor

The carbon sequestration parameter will be shown as in Figure 51. There are three parameters to be defined:

- Capture ratio: The mass of captured carbon over the total carbon content of the resource used as an input
- Energy used: The type of energy used by the carbon capture filter
- Energy per kg of C removed: The amount of energy used per kilogram of carbon removed by the filter

3.8.4 Transportation Process Editor

The transportation process editor is used to create or edit transportation processes in GREET. It can be accessed either by editing an existing transportation process or by creating a new one.

Opening an existing transportation process: To edit an existing transportation process, click on the **Data Editor** button of the main pane selector. Then open the **Processes** menu and click **Modify Process**. A window will appear with a list of all the processes and selecting a transportation process will open the transportation process editor.

The procedure to edit a transportation process is then the same as creating a new one.

Creating a new transportation process: To create a new transportation process, open the Data Editor main pane. Then open the **Processes** menu and select **Add Transportation Process**.

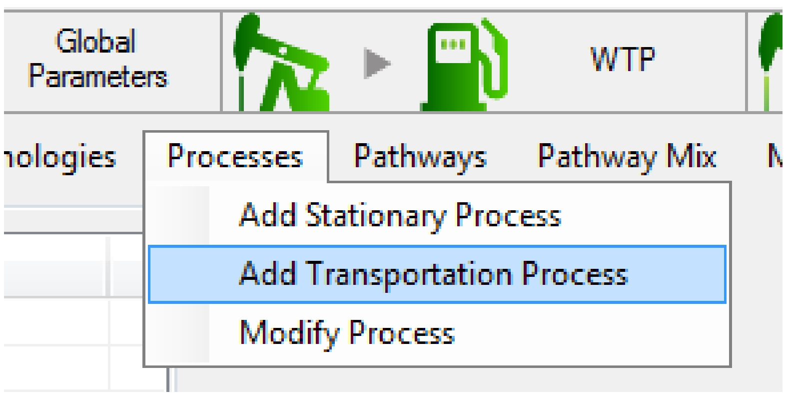


Figure 52: Adding a new transportation process to the database

Then, building a transportation process is based on dragging and dropping the transportation steps you need and linking them together. See Figures 52 and 53. Currently six transportation modes are supported:

- Ocean Tanker
- Barge
- Truck
- Pipeline
- Train
- Connector

The main difference between them is how the energy intensity is calculated [1]. Besides that, they all need to define a distance and a share.

NOTE: To calculate the energy intensity for the Ocean Tanker, Barge, and Truck modes the payload for the material transported needs to be specified. The payloads can be defined in the mode editor (see Section 3.8.7).

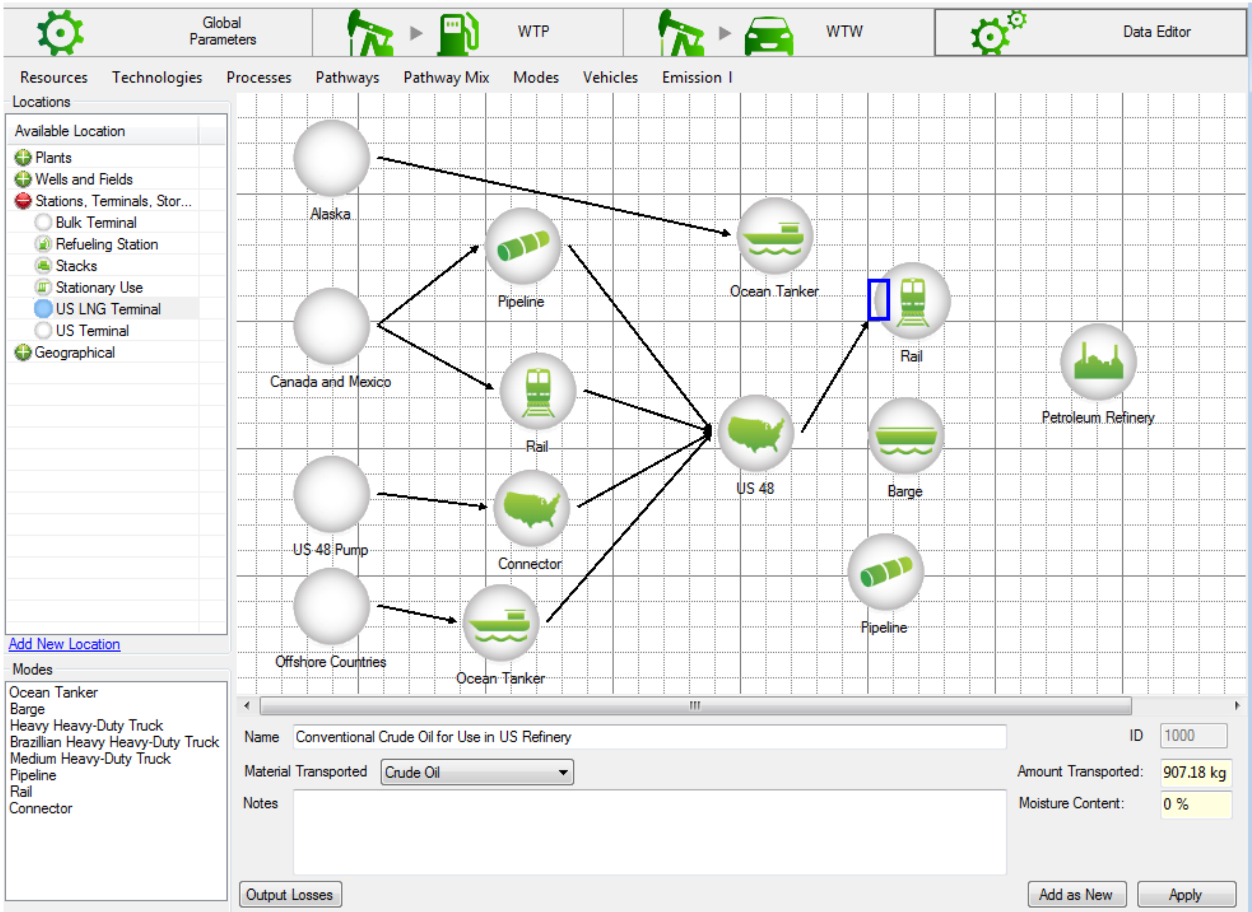


Figure 53: Linking Transportation Steps and Locations in the Transportation Editor

Select the material transported: When building a new transportation process, the first thing to do is to select the material transported. See Figure 54. The reason the material transported needs to be specified is that the energy intensity for most of the modes (except for Rail) depend on it.

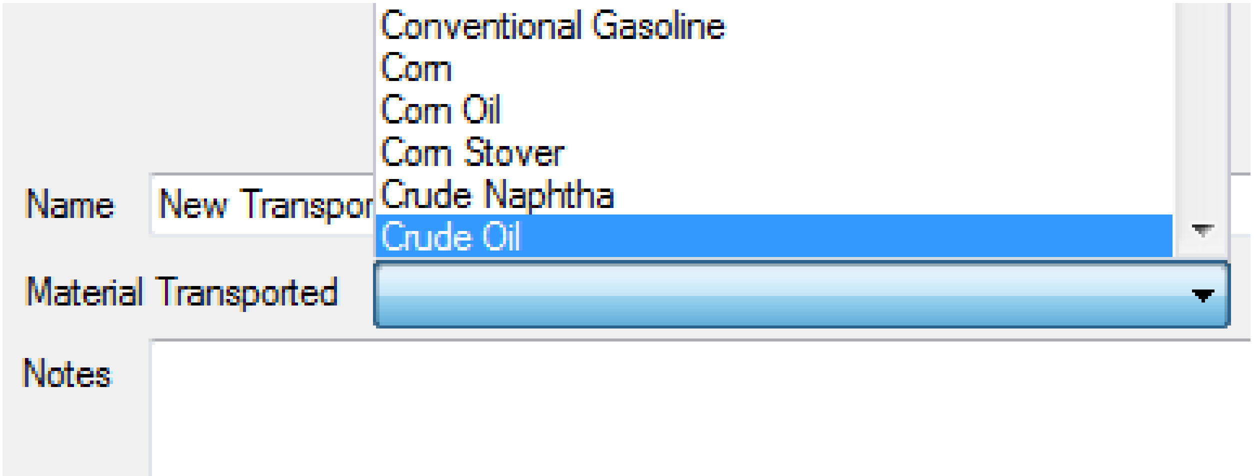


Figure 54: Selecting the Material Transported

Energy intensity assumptions: For Ocean Tanker, Barge, and Truck the energy intensity is calculated based on the payload attribute of the mode for a specific material transported.

For Pipeline, the energy intensity depends on the state of the material; liquids and solids will share the same energy intensity because usually solids are carried dissolved in some fluid. Gaseous materials are usually transported in their gaseous state, so there is a different energy intensity associated with transporting gaseous material.

The energy intensity of Rail mode is constant and does not depend on material transported.

The Connectors are used in rare situations when there is no energy consumption associated with transportation. The Connectors have an energy intensity of zero.

Building the transportation structure: The transportation structure is built by dragging and dropping transportation steps and locations from the left side of the control to the building area.

There is a tree list of locations and a list of modes on the left side of the control. The locations are organized by categories that need to be expanded to see the locations. The modes are a flat list of the actual modes in the database.

Once the material transported is set, you can start dragging and dropping locations and modes, see Figures 55 and 56.

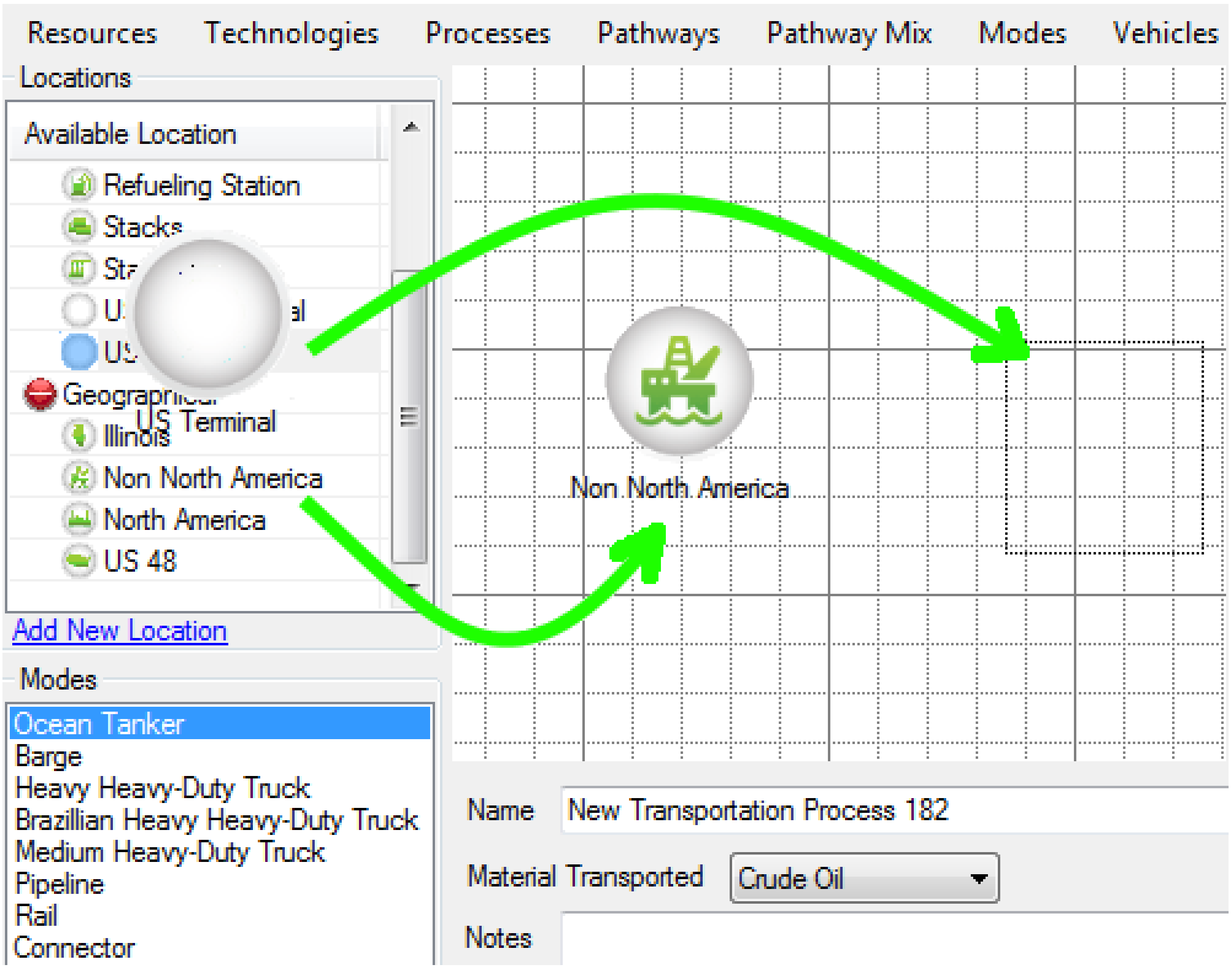


Figure 55: Dropping Locations for the Transportation Process

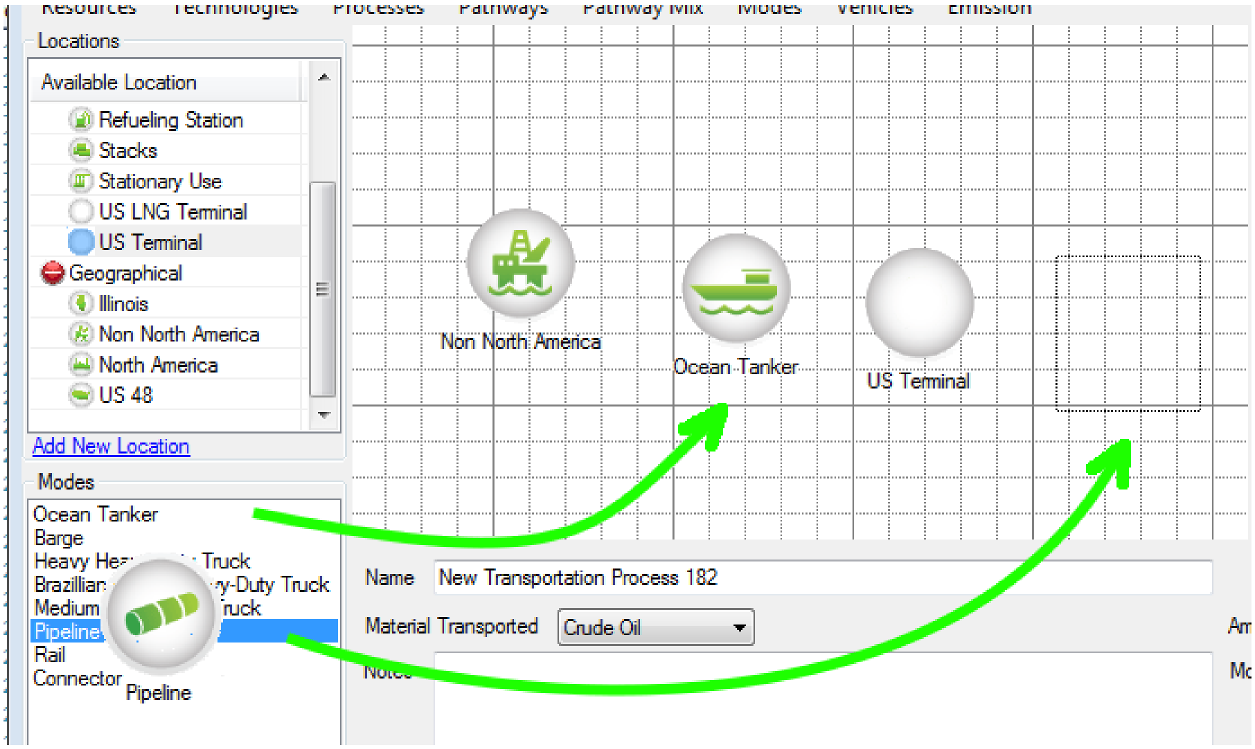


Figure 56: Dropping Modes in the Transportation Process

Multiple transportation modes can be branched into or from a location node, but two transportation modes cannot be linked side by side. The reason it is enforced is to have a clear graphical representation of the process; the locations are, however, not used in any way in the calculations.

Once all of the desired locations and transportation steps are dropped in the building area, they can be linked together as shown in Figure 53. To link them, pass the cursor of the mouse over the pictogram until a blue square appears. Once it appears, left click and hold; move your cursor to the desired location and release the left mouse button.

To delete any of the elements (links, transportation steps, or locations), right click the element and select **Delete** from the menu.

Setting step parameters: There are several parameters that need to be specified for each of the transportation steps.

To access the transportation step parameters, right click on the pictograph and select **Edit Step Parameters**, see Figure 57. A form will appear with the different parameters for that transportation step. The set of step parameters depends on the mode of transportation. It usually includes the distance traveled, the share that represents what fraction of the transported resource is being transported by the step, and the urban share, used to calculate what fraction of the emissions are going to urban areas.

Each mode might have multiple fuel shares defined. A fuel share is a weighted list of fuels used to propel the engine of a given transportation mode. By clicking the right arrow next to the fuel share, the form can be expanded to show the actual shares.

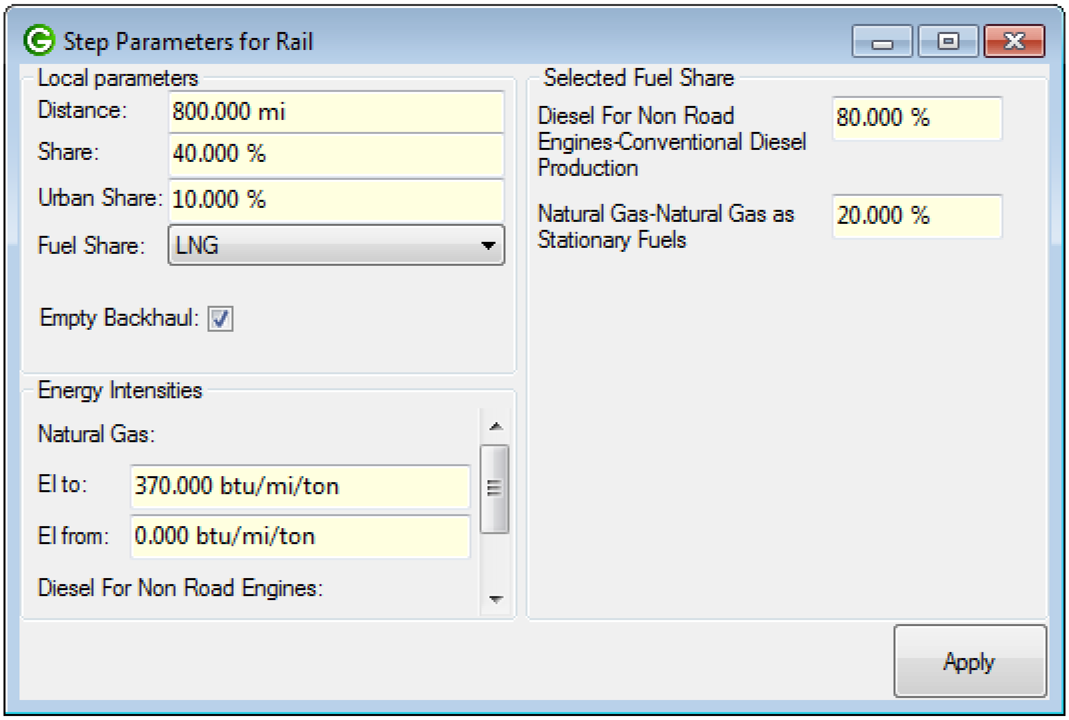


Figure 57: Mode Parameters and Mode Fuel Shares

Multiple sets of fuel shares are predefined and can be selected using the drop-down box. In the example in Figure 57, the fuel share named LNG is selected, which sets the fuels used by the train engine to 80% diesel and 20% natural gas. Fuel shares can be modified in the mode editor.

When the Empty Backhaul check box is checked, the energy and emissions associated with the empty backhaul trip will be attributed to the transportation process.

Below, the energy intensities (EI) are listed for each fuel used. The *EI to* is used during the calculations for the trip from the origin to the destination, and the *EI from* is used for the trip from the destination back to the origin if empty backhaul is checked.

NOTE: The Rail and Pipeline modes have EI from equal to 0. It is assumed there is no empty backhaul for those modes. Thus, the Empty Backhaul flag is irrelevant and it does not effect the calculations.

Step Losses. Losses can be defined for each transportation step. To add or edit losses for a transportation step, right click over the step icon and in the opening menu, select **Edit Step Losses**.

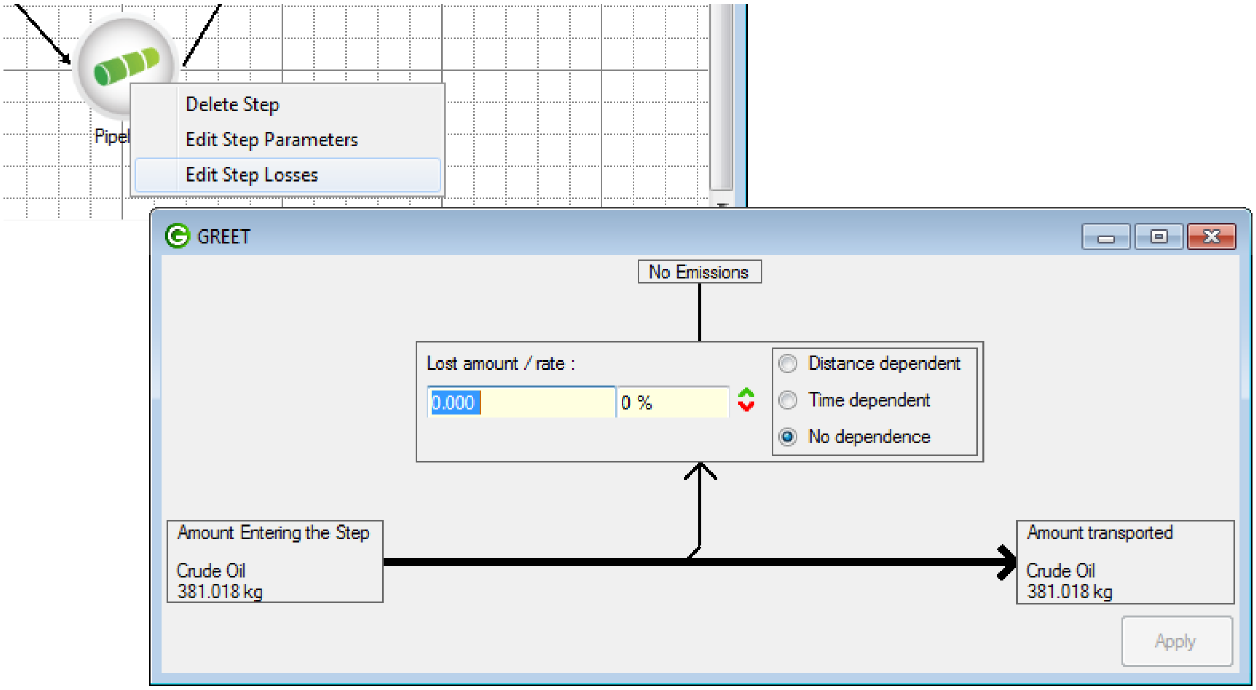


Figure 58: Transportation Step Losses

A new form will pop up, see Figure 58. The loss is assumed to occur at the beginning of the transportation step [1]; it can be defined as a constant rate, or it can depend on distance or travel time.

- Constant loss rate: If the **No dependence** option is chosen, the losses will be calculated by multiplying the loss rate by the amount of resource transported.
- Time dependent loss rate: The time necessary for travel will be calculated using the distance and the average speed of this mode. Then the rate expressed in %/sec is multiplied by the amount of resource transported and the calculated time of travel.
- Distance dependent loss rate: The loss rate will be defined in %/m and the distance of travel will be used as a parameter to calculate the lost amount.

The loss control shows the amount transported, the boundaries of the transportation process system definition, the amount lost with associated emissions, and finally, the amount transported and available at the end of the transportation process.

Input and output losses: The same way that step losses can be defined, Input and Output losses can be defined for the entire process. In that case, the notion of distance or time does not exist anymore and the user needs to specify the loss rate.

NOTE: Losses cannot be defined for transportation process fuels, i.e., fuels used to propel the mode engine.

Amount transported: The amount transported for a transportation process does not have any impact on the results. All our results are automatically converted on display and the energy intensities are in {energy} per {mass} per {distance}. If the process is defined to transport one kilogram, all the results in the memory of the program will be calculated for a kilogram; however, on the screen, this will be converted using your preferences for display, by default per 1 million Btus. So whatever the amount transported is, we will assume a linear relationship between what is actually calculated in the memory and your preference for the functional unit in the "Well-to-Pump" editor.

Moisture content: The moisture content is used to define what percentage of material versus water is actually transported. Because what interests us is always the energy used for transportation of dry matter, and water is a part of the payload, having a moisture content of 20% will affect the results by a ratio of 1/0.8. The energy used to transport the dry matter will increase by this factor, and all the results seen in the Well-to-Pump main pane will be the results equivalent for dry matter.

3.8.5 Pathway Editor

Adding a new pathway: To create a new pathway, open the **Data Editor** main pane, then open the **Pathway** menu and select **Add Pathway**. When the pathway editor opens, you will see the editor as shown in Figure 59.

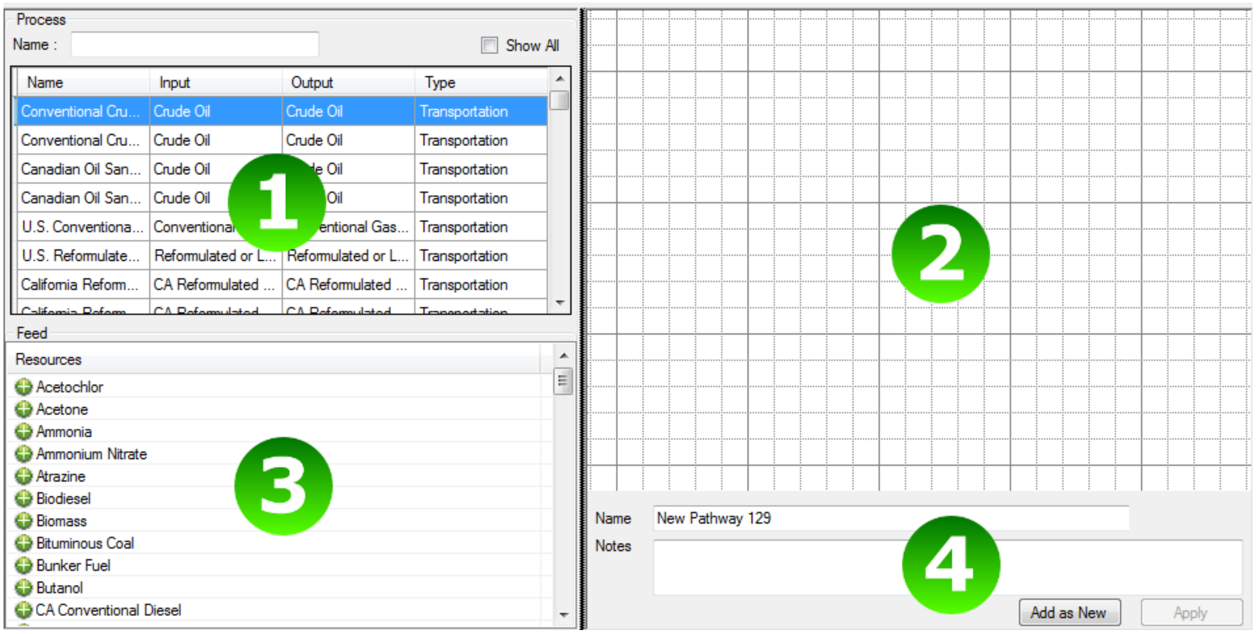


Figure 59: Creating a New Pathway

The pathway editor is designed in 4 main zones. See Figure 59. Zone 1 lists all the available processes from the database. Zone 2 is the building area. Zone 3 lists all the available Pathway Mix. Zone 4 shows the attributes of the pathways.

To build a pathway, you can find processes by name, then drag and drop them into the building area. The list of processes will automatically update and show you only processes which can be linked to an end of the pathway. If a pathway is not built correctly, links will appear in red and a message will be displayed at the top of the building area. Figure 60 shows how to drag and drop the first process.

First, a process has to be selected from the list of available processes. Here we are looking for gaseous hydrogen produced from natural gas.

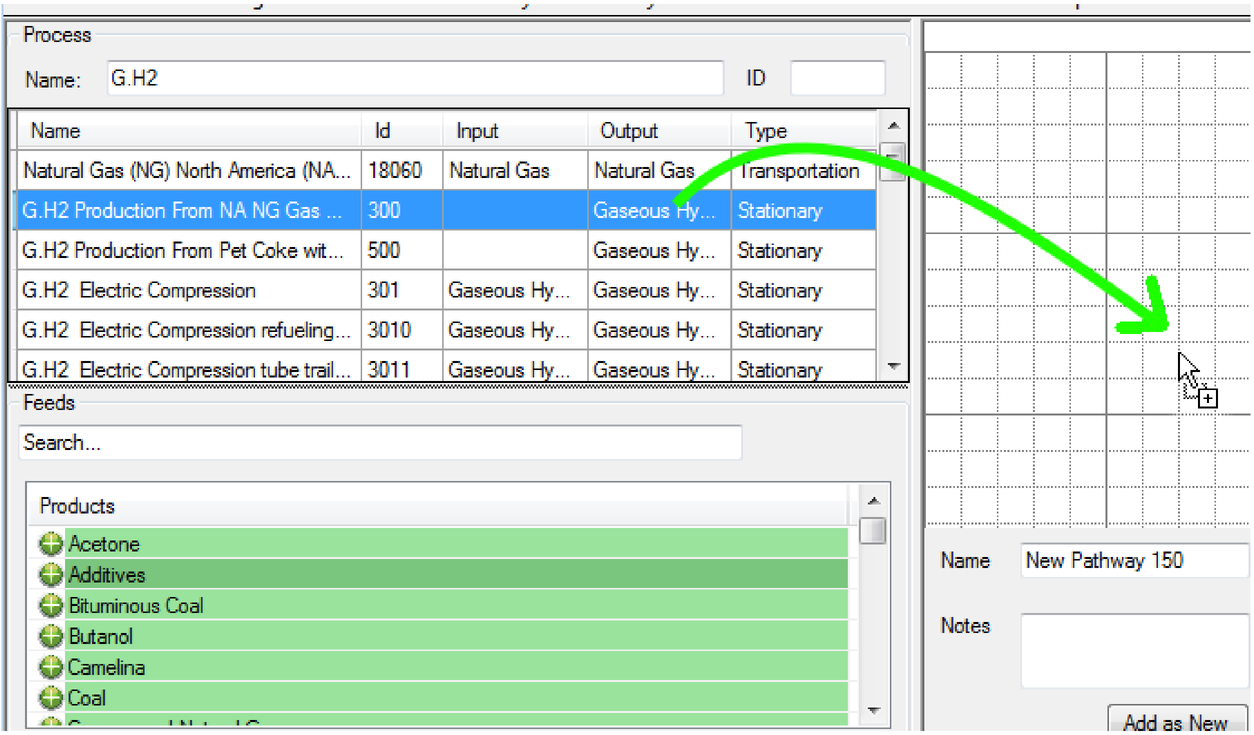


Figure 60: Dropping the First Process

As a result, the process is now shown in the building area, but the pathway editor indicates that we have an issue in this pathway. One of the inputs of the first process is not defined correctly, because we are missing an upstream. The input has as a source Previous which means that there will be a process before it, such as a transportation process delivering natural gas. If the first process of a pathway had an input from Previous, then the pathway must have a feed to indicate what the upstream is before this first process. By double clicking on the + sign at the top right of the process, we can expand it and see which input is not assigned correctly. In this case, it is the natural gas input. See Figure 61.

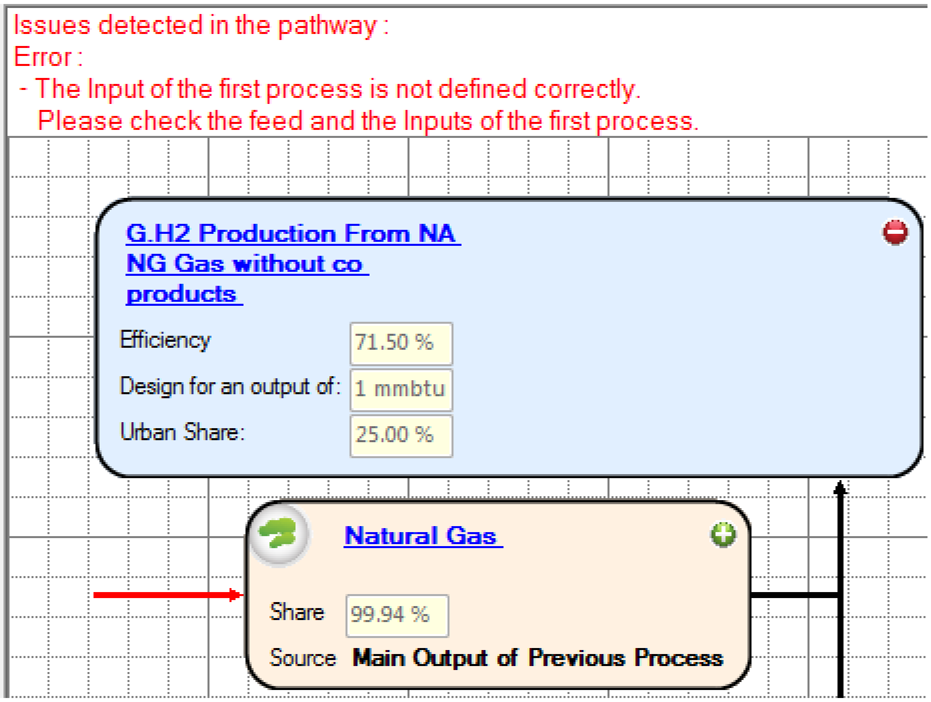


Figure 61: Erroneous Pathway

To fix this issue, a natural gas upstream needs to be selected in Zone 3 of the editor where resources and Pathway Mixes are listed. By expanding the natural gas resource, we can see that multiple Pathway Mixes are available. The Pathway Mixes named North America with Transportation, which consists of a blend of shale gas and regular natural gas transported by pipeline, will be used. To do so, we drag and drop the Pathway Mix to the building area. A dark green rectangle will indicate to you where to drop this element.

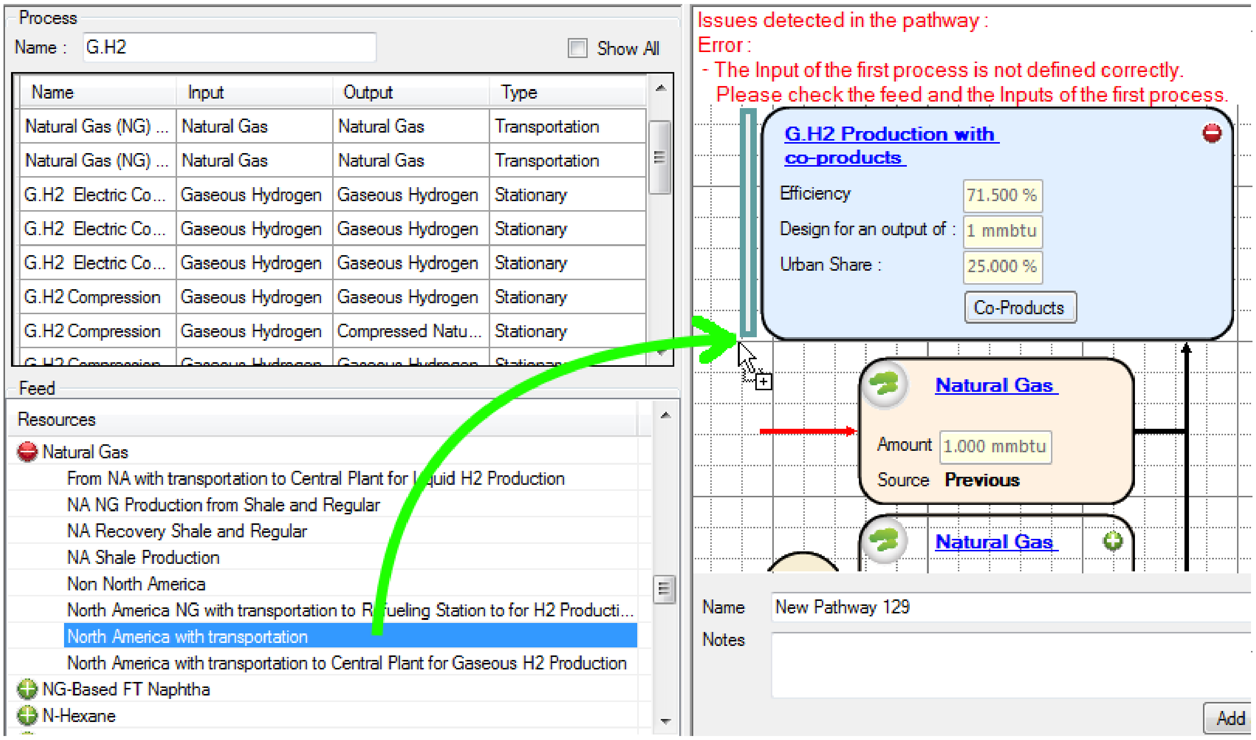


Figure 62: Dragging and Dropping a Pathway Mix to Create a Valid Pathway

Now that the pathway is valid, we do not have any more warnings, and we can save this pathway and include it in our model. See Figures 62 and 63.

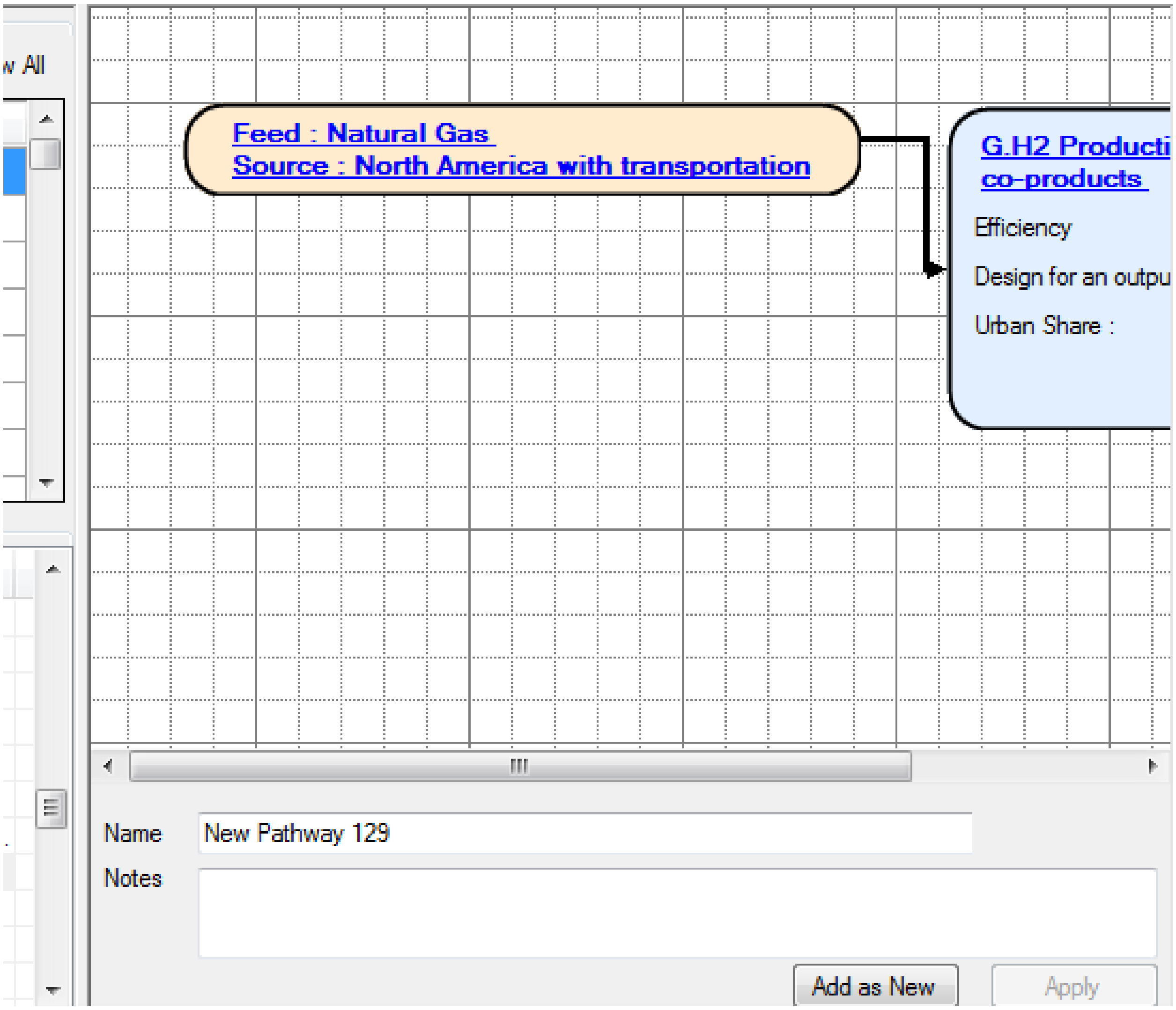


Figure 63: Pathway Is Done and Not Showing Any Warning Signals

NOTE: It would have been possible to fix this by dragging and dropping the natural gas Extraction process that is using a primary resource from a well and outputs natural gas. That way, the upstream would be less significant, and this pathway would represent hydrogen produced from natural gas directly at the extraction well.

Modifying an existing pathway: To modify an existing pathway, open the **Data Editor** main pane; then open the **Pathway** menu, and select **Modify Pathway**. A new pop-up window will appear, and you will have to select which pathway you want to modify. See Figure 64.

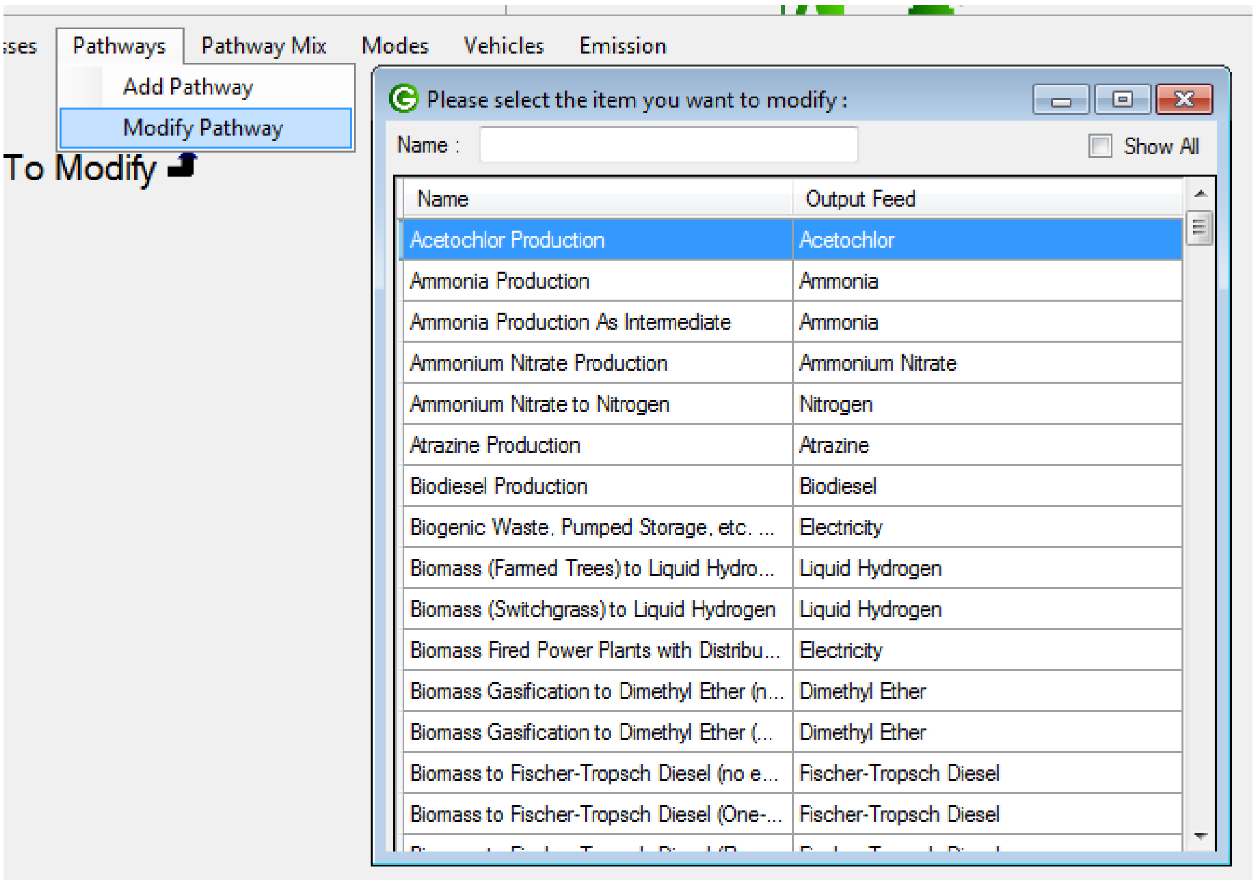


Figure 64: Select an Existing Pathway for Modifications

Follow the same procedures as explained in the previous paragraph for adding new processes and feeds.

To delete some processes, right click on them and click **Delete**. See Figure 65. Keep in mind that deleting a process might break the integrity of the pathway and warning messages might be shown.

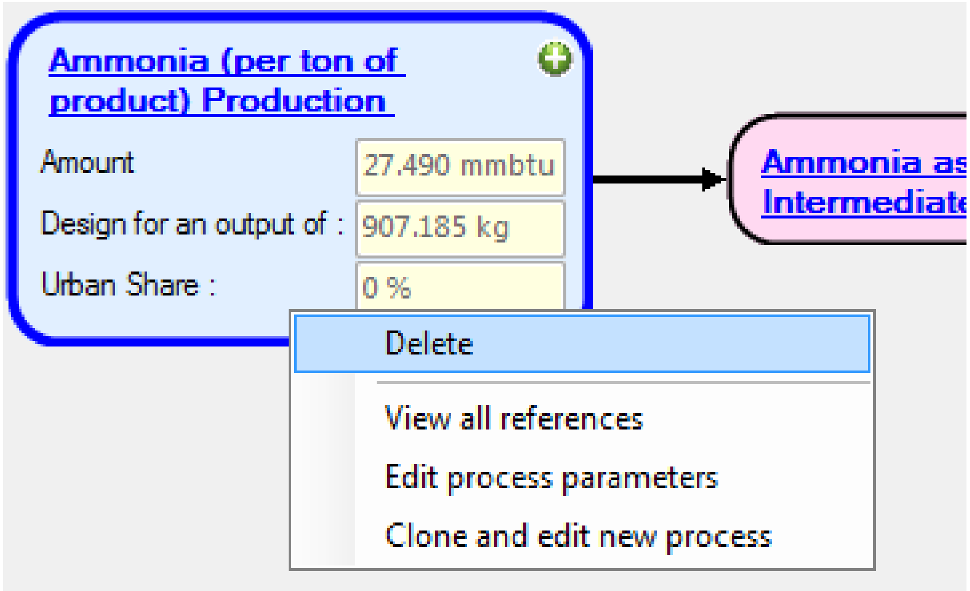


Figure 65: Menu Appearing While Right Clicking on a Process in the Pathway Editor

The **View all references** link will open a form showing all the pathways which are using the selected process.

Edit process parameters will open the selected process into a process editor and allow you to make modifications to it. Once the modifications are done, the pathway will be updated with the edited version of the process. This can be used to solve input-output issues within a pathway.

Clone and edit new process will create a copy or a clone of the selected process and use it in place of the selected process. This can be used to create many different pathways where a process only changes slightly between each version.

Possible errors while building a pathway: *There are no processes in the pathway*: you are trying to save a pathway which does not contain any processes. You can either add processes or not save this pathway.

The Input of the first process is not defined correctly. Please check the feed and the Inputs of the first process: The first process might have an input from previous, but there is no previous process or previous upstream. Therefore, one of the inputs is not linked to anything, and the process is not considered as valid. To fix that, add a feed to the pathway. Using an existing Pathway Mix corresponds to the first process input resource marked as from Previous for this pathway.

The pathway has not been given a name: GREET requires that all pathways have a name; if not, it would be hard to find them in the database. To fix that, just add a name in the properties area of the pathway.

The process PROCESSNAME input from previous RESOURCENAME does not match the feed resource RESOURCENAME2: This issue means a Pathway Mix is being used as a feed for the pathway which is a different resource than what the first process is expecting. For example, if we are using a Pathway Mix crude oil production, while the first process inputs from previous needs natural gas, you will see this error. To fix that, change the first process or replace the feed with another Pathway Mix.

PROCESSNAME1 output does not match PROCESSNAME2 input: Processes in the pathway should not be linked together. The output of a process resource does not match the input from the next process in the pathway. In this case, an intermediate process should be added to convert the output of process 1 to what process 2 needs. Or you need to review which processes have been dropped.

3.8.6 Pathway Mix

The Pathway Mix allows you to define how multiple pathways are used together to form the source for a product. See Figure 66. There are, for example, multiple ways of obtaining crude oil, and the Crude Recovery for U.S. Refineries defines which pathways are used on average in the United States.

To access the upstream editor, click on the **Data Editor** button of the main panes selector; then open the **Pathway Mix** menu, and select **Add Resource Pathway Mix** or **Modify Resource Pathway Mix**.

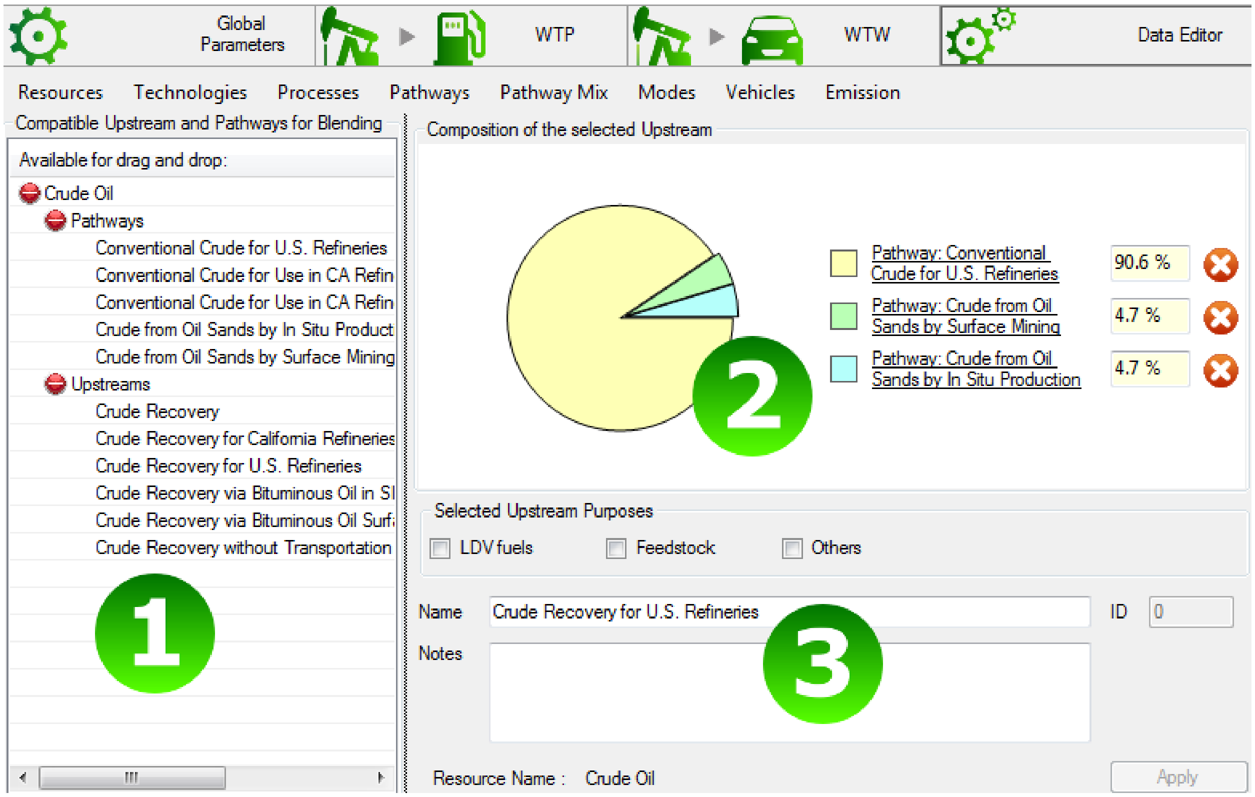


Figure 66: Pathway Mix Showing the Crude Recovery for U.S. Refineries

3.8.7 Modes Editor

The Modes Editor allows you to change a mode's parameters as well as to create new modes.

To edit/create transportation modes, open the **Data Editor** main pane, then open the **Modes** Menu, and select **Edit Modes**.

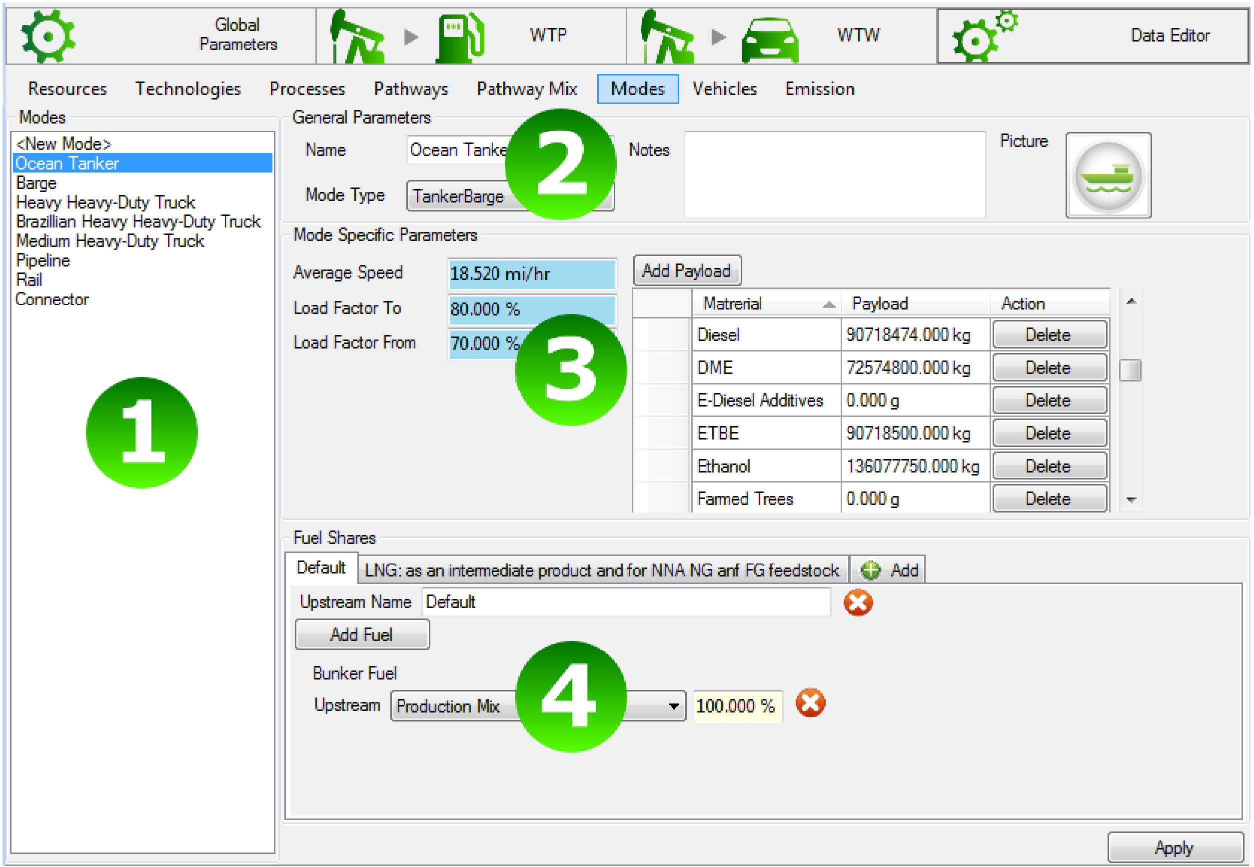


Figure 67: Mode Editor Showing the Parameters for the Ocean Tanker

The Modes Editor is composed of 4 different zones. See Figure 67. Zone 1 is marked where you can select the mode to edit. By selecting different modes in this zone, the area to the right will be populated. Zone 2 contains the basic properties of the mode; its name, notes, picture, and type. GREET supports 6 mode types:

- Ocean Tanker
- Barge
- Truck
- Pipeline
- Train
- Connector

Zone 3 contains the specific parameters of a mode; these might change depending on which mode type is being used for the currently edited mode. Zone 4 contains the fuel shares for this mode.

Specific parameters for Ocean Tanker and Barge: Ocean Tankers and Barges are close to the same transportation modes and their specific parameters are not different from each other. We have the following parameters:

- Average Speed: This represents the average speed from the destination to the originating location.
- Load factor to: The load factor to represents the throttle or the percentage of power used over power installed from the originating location to the destination.

- Load factor from: The load factor from represents the throttle or the percentage of power used over power installed from the destination location back to the originating location.
- Payloads: Payload for each resource transported by the mode – they need to be defined to calculate the energy intensities for the modes.

Specific parameters for Trucks: For trucks, the specific parameters are limited to:

- Fuel Economy To: Fuel economy from origin to destination (assumed to be loaded or Full)
- Fuel Economy From: Fuel economy from destination back to origin (assumed to be unloaded or Empty)
- Payloads: Payload for each resource tranported by the mode – they need to be defined to calculate the energy intensities for the modes.

Adding a payload for Ocean Tanker, Barge, or Truck: To add a payload to one of these modes, click the **Add Payload** button in the mode specific parameters. See Figure 68. A pop-up window will open asking you to select a resource transported.

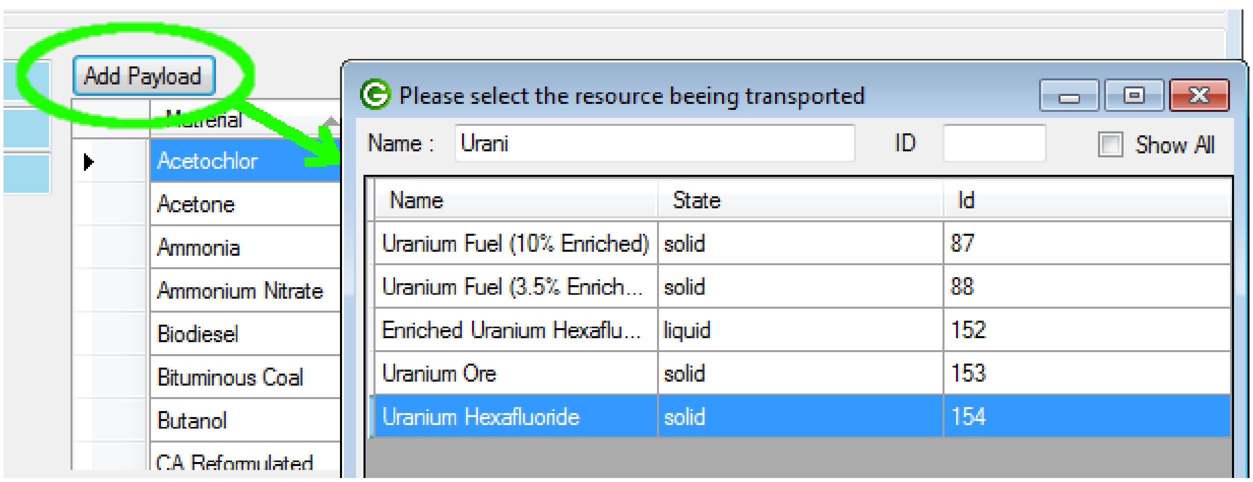


Figure 68: Selecting a Resource Being Transported after Clicking on the Add Payload Button

When the resource has been selected, it will be added to the alphabetically ordered list of payloads; find it in the list and enter a payload for it. In this example, we entered 5 ST. See Figure 69.

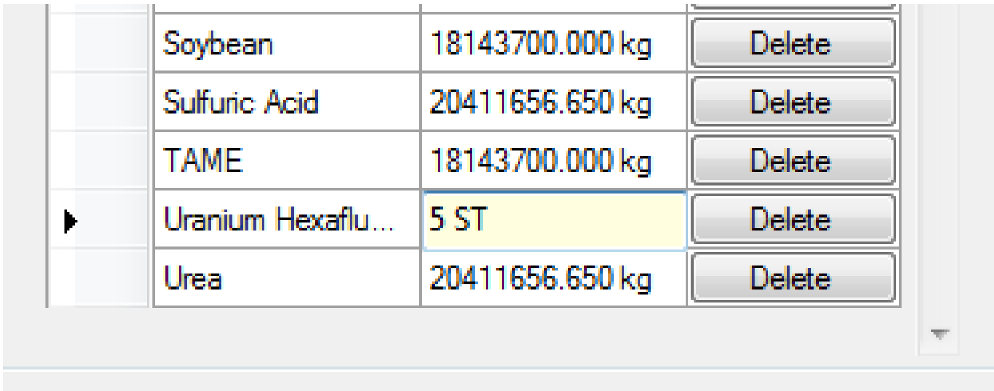


Figure 69: Entering Five Short Ton as a Payload for Uranium Hexafluoride

Specific parameters for pipeline: For pipelines, the energy intensities are not calculated; instead, they are defined as a value for liquids or solids and a few are defined for specific materials.

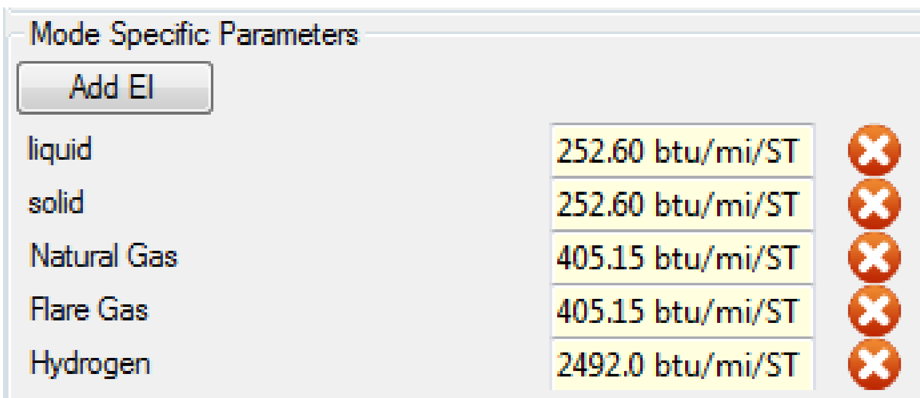


Figure 70: Energy Intensities for a Pipeline

Energy intensities (EI) can be added by clicking the **Add EI** button and then selecting the resource being transported. See Figure 70. To delete an energy intensity, click the red circle with the white x to the right of the energy intensity to be deleted.

Specific parameters for rail: The specific parameters for Rail only includes average speed and a single energy intensity. The same energy intensity is used for any resource being transported and for any fuel used by the train.

Mode fuel shares: The mode's fuel shares are designed to help select different fuel blends to run the different modes. Each transportation mode energy is provided using some fuel associated with some upstream. The mode fuel shares allow you to store different configurations of fuel shares for a mode.

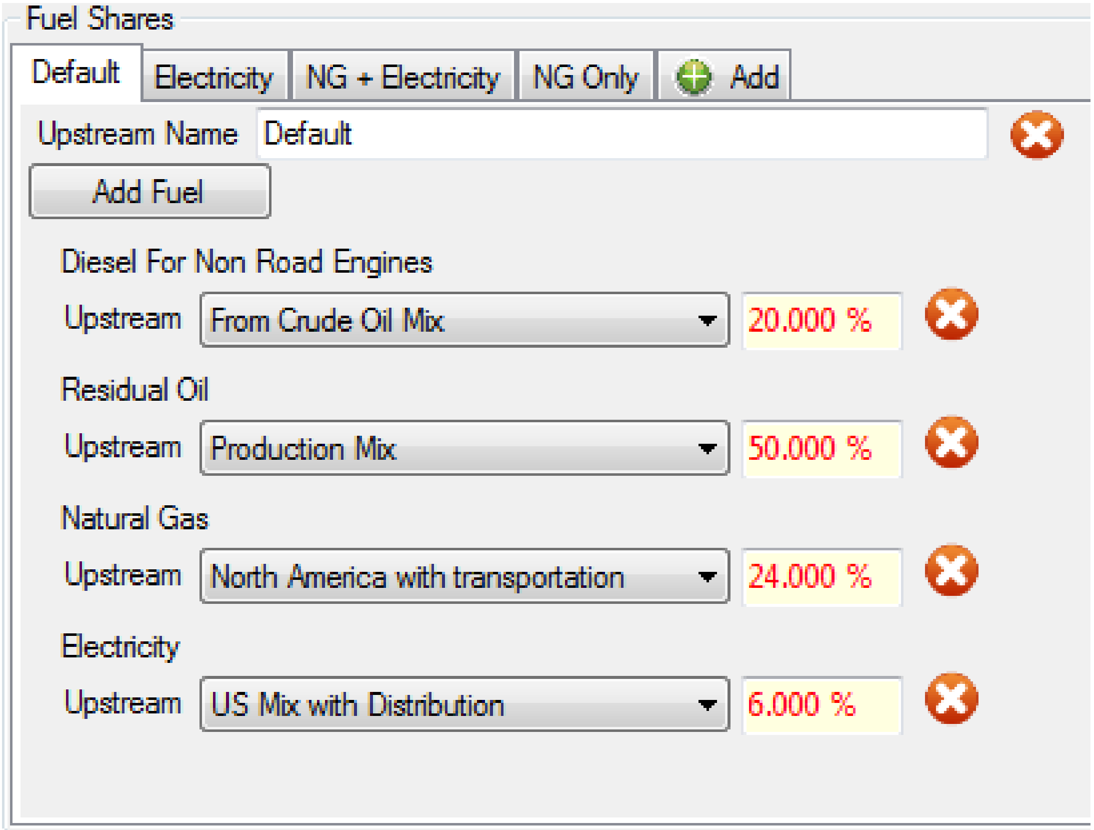


Figure 71: Fuel Shares for a Pipeline Mode

Figure 71 shows, as an example, the different fuel shares for pipelines. This assumes that the energy necessary to operate the pipeline is coming, on average, from some diesel, residual oil, natural gas, and electricity. This configuration is stored as the default fuel share for the pipeline mode. Another configuration would be to use electricity only which is defined on the second tab.

The fuel share configurations are defined here in the Mode editor, but they do not define what is actually used by the calculations for a transportation process. The selection has to be made in the mode parameters in the transportation process editor. That is the place to select the fuel share for the calculations for a specific pipeline of a specific process.

3.8.8 Vehicles Editor

The vehicles editor allows you to change parameters of a vehicle such as fuel used, energy consumption, and emissions. See Figure 72.

To edit/create vehicles, open the **Data Editor** main pane; then open the **Vehicles** menu and select **Modify Vehicles**.

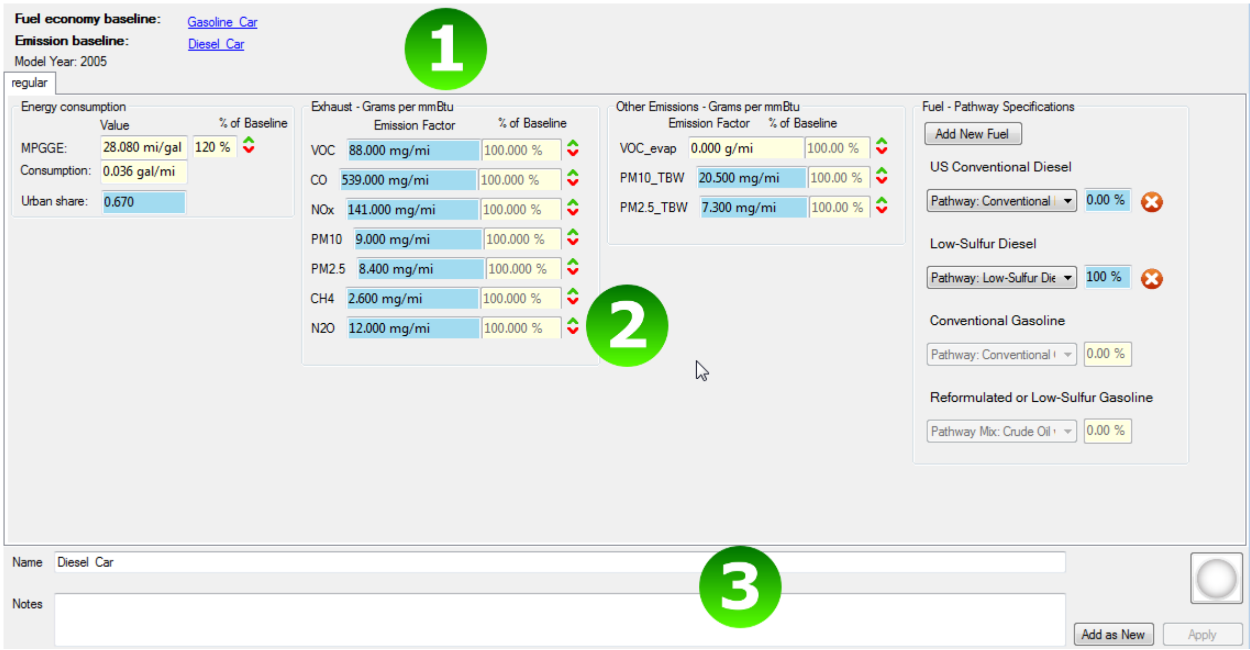


Figure 72: Vehicle Editor Showing the Properties for the Baseline Gasoline Vehicle

The vehicle editor is organized into 3 different zones. Zone 1 shows what vehicles we are actually looking at and what the baselines for emissions and energy consumption are. Zone 2 shows the parameters for the vehicle; this might be different for regular or plug-in hybrid electric vehicles (PHEVs). Zone 3 is where the name of the vehicle can be recorded.

Vehicle modes: Vehicle parameters in GREET are categorized by mode. There are two types of operating modes:

- Regular or Charge Sustaining (CS): In this mode, the vehicle is using energy from the tank only and any secondary means of energy storage is not depleted. In a hybrid vehicle, some of the energy from the battery may be used to assist the petrol engine; however, this is a small amount of energy compared to the large batteries used in PHEVs.
- Charge Depletion (CD): The charge depletion mode is used by PHEVs. This mode assumes that electricity from a battery is used as a primary source of energy.

When multiple modes are used for a vehicle they show as tabs over the properties. See Figure 73.

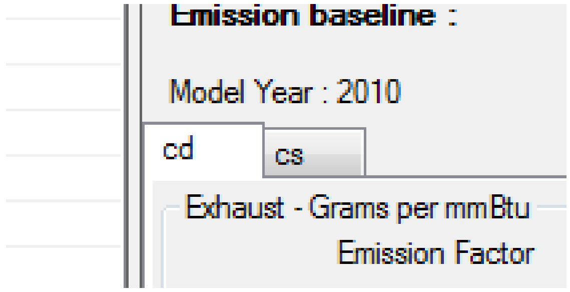


Figure 73: Charge Depletion and Charge Sustaining Modes Showing as Tabs in the Vehicle Editor

Energy use: The energy used by the car while operating in the regular or charge sustaining mode has to be set in the energy consumption group. See Figure 74. The unit used is the miles per gallon of gasoline equivalent (MPGGE) for the fuel economy and gallons per mile of gasoline equivalent for the consumption definition.

If the edited vehicle has a baseline for energy, the energy used can also be adjusted using the “% of Baseline” or the up and down arrows.

To revert back to the original value, right click on the value and select Switch to default value.

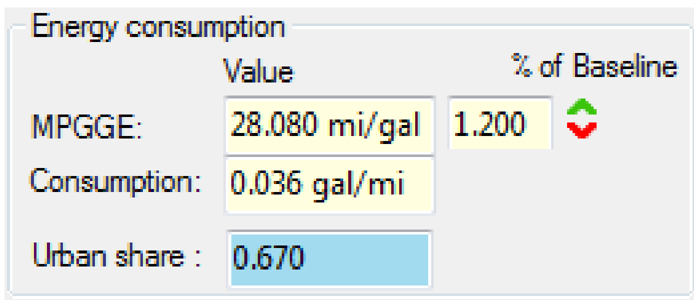


Figure 74: Vehicle Energy Use Parameters

Fuel and upstream specification: The fuel group represents the fuels and Pathway Mixes that are used by this vehicle. In the example shown in Figure 75, the fuels used by the car are methanol, conventional gasoline, and reformulated or low-sulfur gasoline. Only the methanol can be modified; this is because of the fuels used by the baseline vehicle. In this case, the baseline vehicle uses 50% conventional gasoline and 50% reformulated or low-sulfur gasoline. The current vehicle defines that it uses 90% methanol, the other 10% is shared 50/50 (due to the baseline shares) between the baseline fuels used. We end up considering the fuel use to be 90% methanol, 5% conventional gasoline, and 5% reformulated or low-sulfur gasoline.

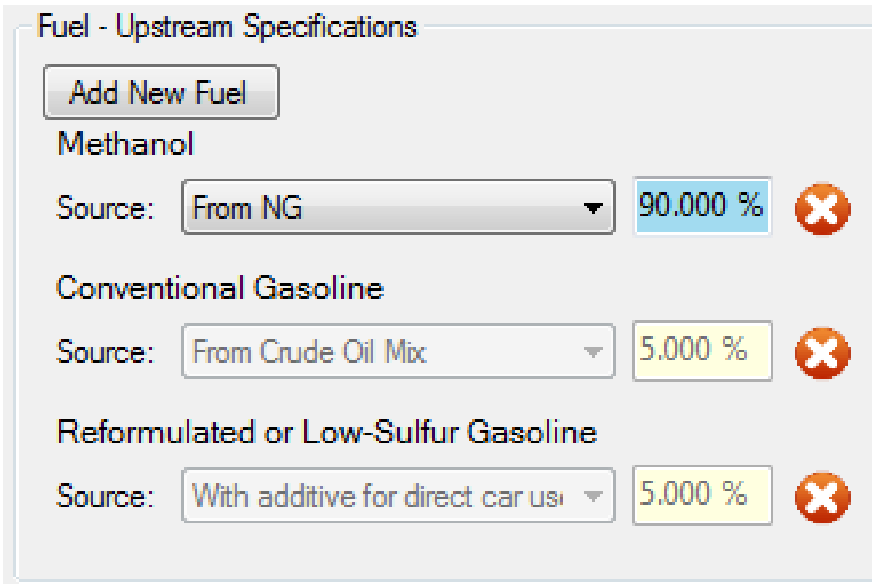


Figure 75: Vehicle Fuel Upstream Parameters

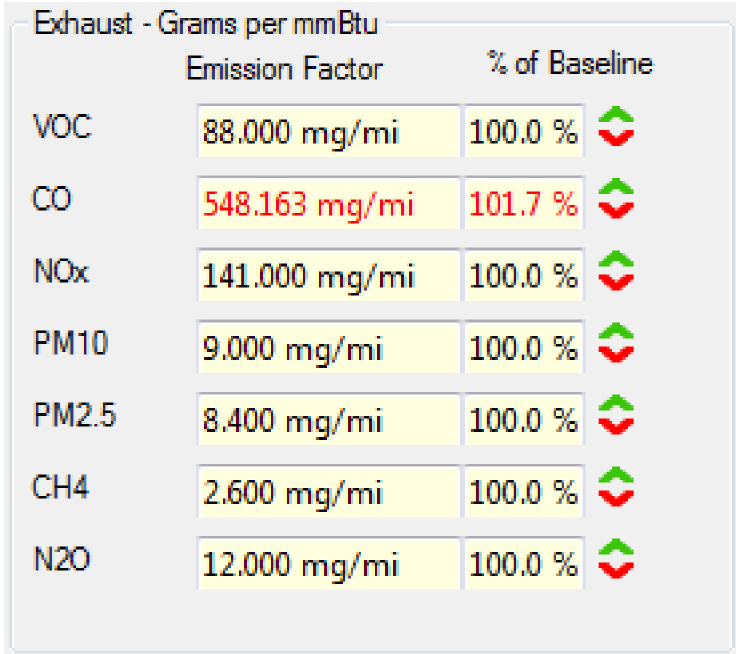


Figure 76: Vehicle Exhaust Emissions

Exhaust emissions: The exhaust emissions define the average emissions per mile. See Figure 76. This can be defined as a percentage of the baseline vehicle for emissions or as a value.

Other emissions: The other emissions are not related to the internal combustion engine operation, but to external factors, such as tire and brake use or gasoline evaporation. See Figure 77.

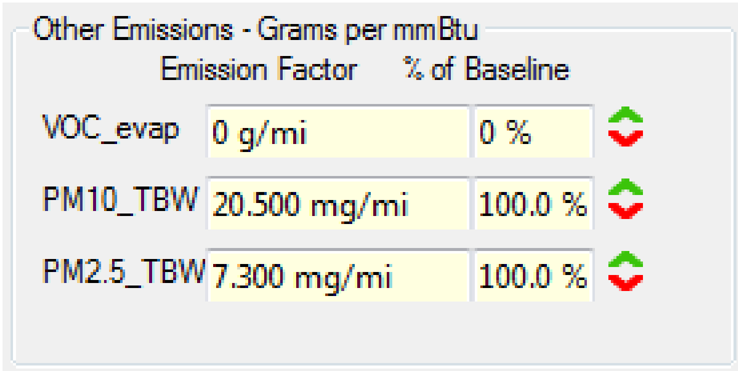


Figure 77: Vehicle Other Emissions

PHEV parameters: The PHEV parameters are only available in the CD mode. See Figure 78.

- Electric Range (rated): The constructor design's electric range
- Electric Range (actual): The actual electric range estimated on regular use (air conditioning, heating, headlights, etc.)
- Fuel Consumption: The internal combustion engine fuel averaged consumption in the CD mode. This value is automatically extracted from the fuel consumption in CD mode table.
- Electricity Consumption: The average electric consumption from the battery per mile. This value is automatically extracted from the electricity consumption in CD mode table.
- Charging Efficiency: Assumed charging efficiency for the vehicle or losses between the electricity upstream and the energy contained in the battery.
- Electricity Upstream: The electricity Pathway Mix for charging this vehicle

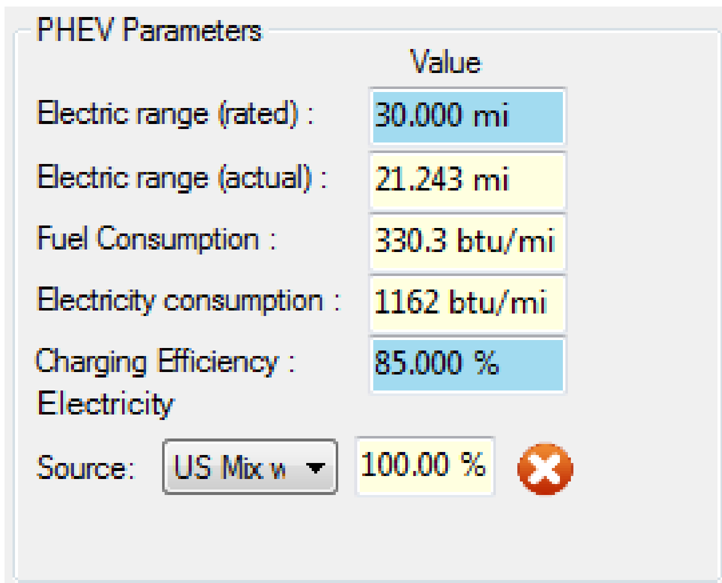


Figure 78: Vehicle PHEV Parameters

Fuel consumption CD mode: The fuel consumption in CD mode represents the energy used by the internal combustion engine to assist the electric propulsion. See Figure 79. The table shows, in relation, the rated electric range and the average fuel consumption. The big gap in the fuel consumption when changing from the 25 miles range to the 30 miles range is due to a change in the hybrid vehicle drivetrain used; power is split for ranges lower than 30 miles and in series for higher ranges.

Fuel Consumption CD Mode

Electric Range:	Consumption:
5.000 mi	1597.574 btu/mi
10.000 mi	1597.574 btu/mi
15.000 mi	1528.428 btu/mi
20.000 mi	1459.282 btu/mi
25.000 mi	1459.282 btu/mi
30.000 mi	330.337 btu/mi
35.000 mi	261.029 btu/mi
40.000 mi	191.721 btu/mi
45.000 mi	191.721 btu/mi
50.000 mi	191.721 btu/mi
55.000 mi	191.721 btu/mi
60.000 mi	191.721 btu/mi

Figure 79: Vehicle Fuel Consumption in CD Mode

Electricity consumption CD mode: The table shows the rated electric range and the averaged electricity consumption while in CD mode. See Figure 80. The big gap in the fuel consumption when changing from 25 miles range to 30 miles range is due to a change in the hybrid vehicle drivetrain used; power is split for ranges lower than 30 miles and in series for higher ranges.

Electricity Consumption CD Mode

Electric Range:	Consumption:
5.000 mi	658.383 btu/mi
10.000 mi	658.383 btu/mi
15.000 mi	652.468 btu/mi
20.000 mi	646.554 btu/mi
25.000 mi	646.554 btu/mi
30.000 mi	1162.519 btu/mi
35.000 mi	1171.344 btu/mi
40.000 mi	1180.169 btu/mi
45.000 mi	1180.169 btu/mi
50.000 mi	1180.169 btu/mi
55.000 mi	1180.169 btu/mi
60.000 mi	1180.169 btu/mi

Figure 80: Vehicle Electricity Consumption in CD Mode

Operating all electric range: The table shows the relation between the rated electric range and actual electric range when accessories energy consumption is taken into account. See Figure 81. The big gap in the fuel consumption when changing from the 25 miles range to the 30 miles range is due to a change of the hybrid vehicle drivetrain used; power is split for ranges lower than 30 miles and in series for higher ranges.

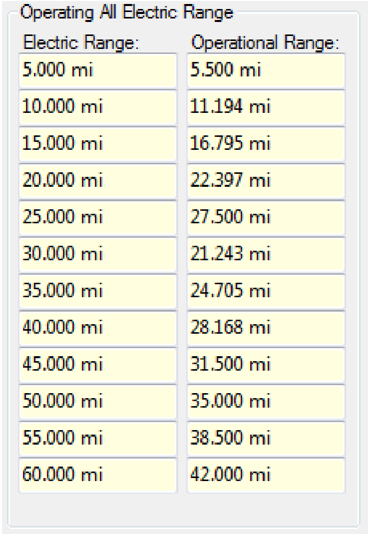


Figure 81: Vehicle All Electric Range

3.8.9 Emissions Editor

The emissions editor is used to modify the properties of gas emissions. See Figure 82. To edit/create emissions, open the **Data Editor** main pane, then open the Emission menu and select Add Emission or Modify Emission.

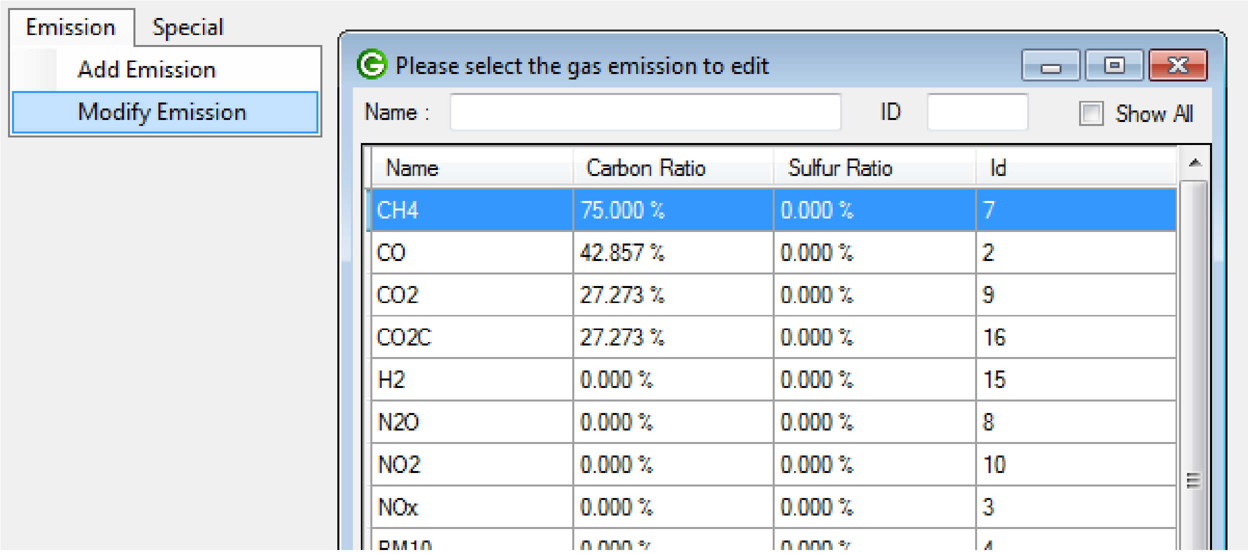


Figure 82: Selecting a Gas to be Opened in the Emissions Editor

Figure 83 shows how the Emissions editor looks when it is opened.

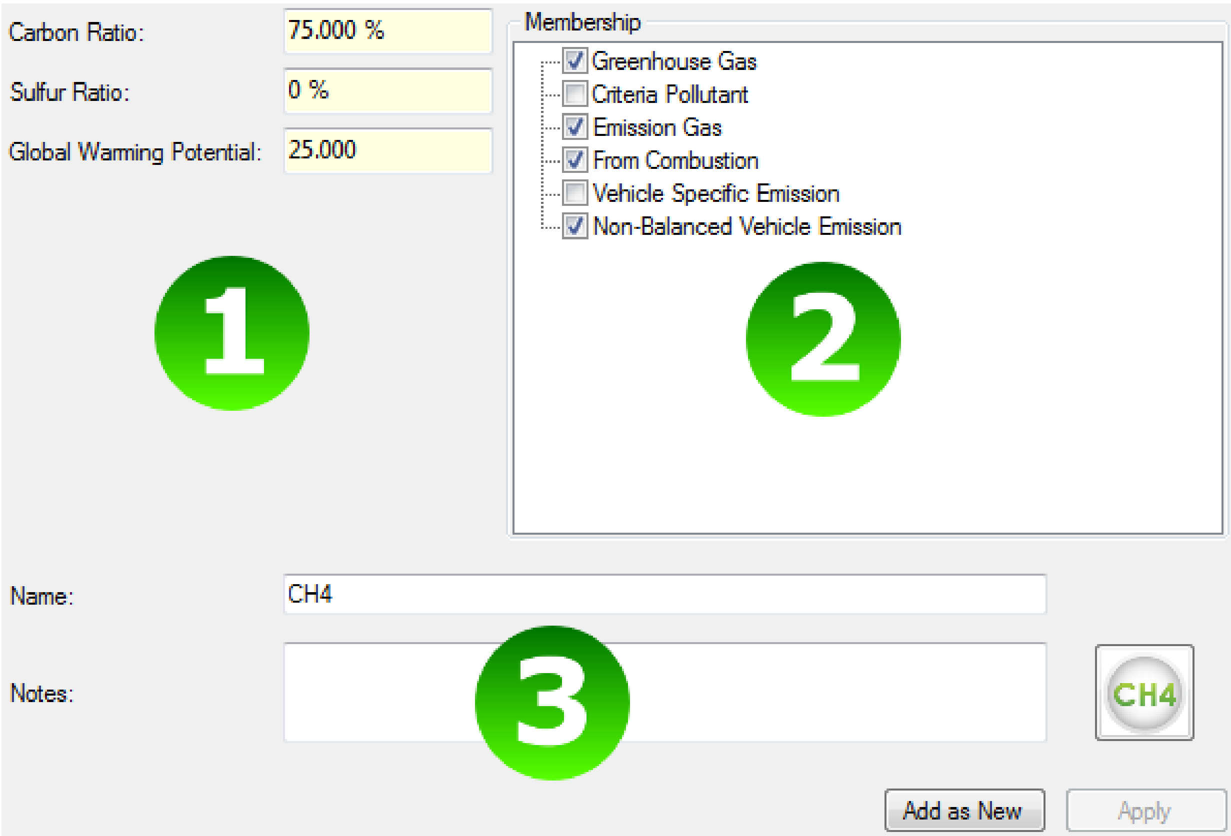


Figure 83: Emissions Editor

The emissions editor or gases editor is made up of 3 zones. Zone 1 defines the properties of the gas. The carbon ratio indicates the mass ratio of carbon atoms in the molecules of the gas. The sulfur ratio indicates the mass ratio of sulfur atoms in the molecules of the gas and the global warming potential represents the factor to be used to compare the global warming potential of this gas to CO_2. Zone 2 lists the groups memberships for this gas. To include this emission gas in a group, just check any of the available groups. Zone 3 contains the name of the actual edited gas and notes associated with it.

4 GREET Features

4.1 Database Automatic Updates

When a .greet file is loaded, GREET checks on our server to see if new data is available. If new data has been made available, you will be asked if you want to download the new data. See Figure 84.

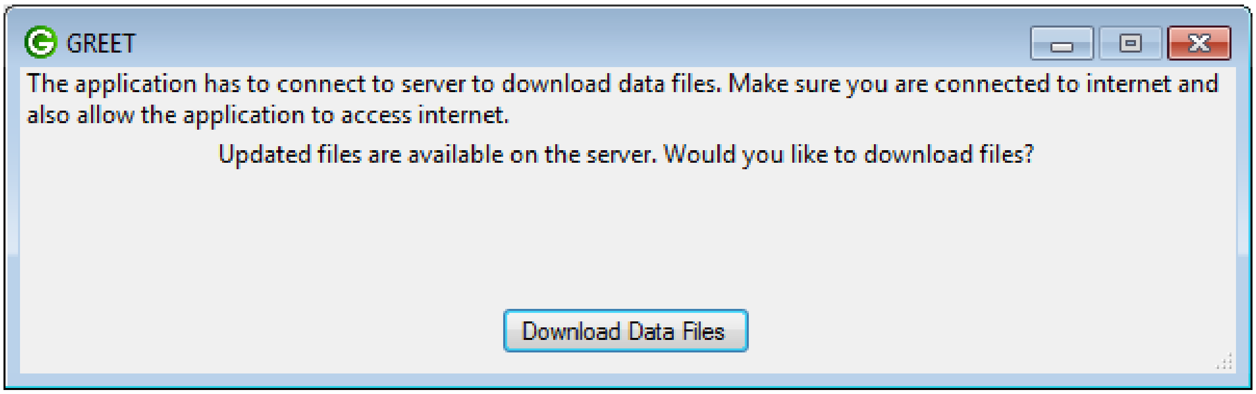


Figure 84: Download Data Files Option

Remember that the downloaded data will overwrite the default file located in the default location: /Documents/Greet/Data/Default.greet. This is why we recommend you do not store your modifications in this file as you might lose them in the case of a database update.

To manually check for new data, please refer to the Data Information and Manual Updates section of this manual.

4.2 Data Information and Manual Updates

The **Database** menu offers you the possibility of knowing which version of the database is actually being used as well as the ability to check and see if new data is available on the server. See Figure 85. If no new data is available, a message will appear telling you so. If new data is available, you'll see a message asking you to click on a Download Data Files button to download the newest data. Please refer to the Database Automatic Updates Section 4.1 of this manual.

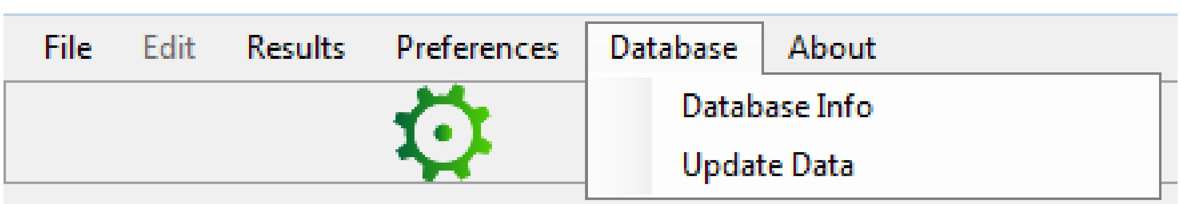


Figure 85: Database Menu

4.3 Monitored Results Overview

In GREET, all of the results can be monitored between each calculation run and boundaries can be set. Then, every time the calculation is run, the software will check to see if these values are within their set boundaries.

To see all the monitored values, open the **Results** menu and select **Monitoring Results**. See Figure 86.

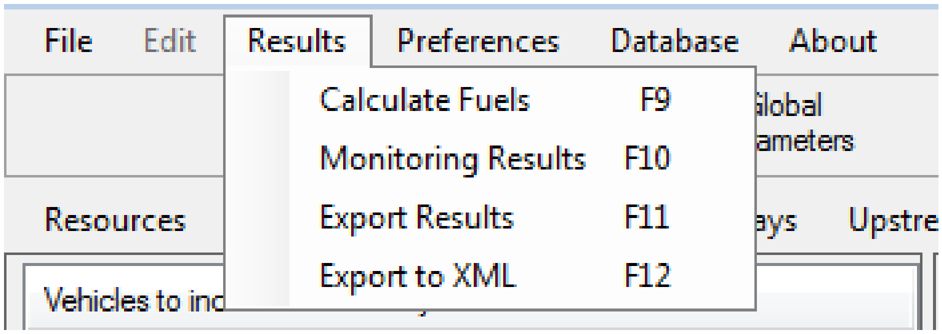


Figure 86: Results Menu

A new window will appear showing you all the monitored results. See Figure 87.

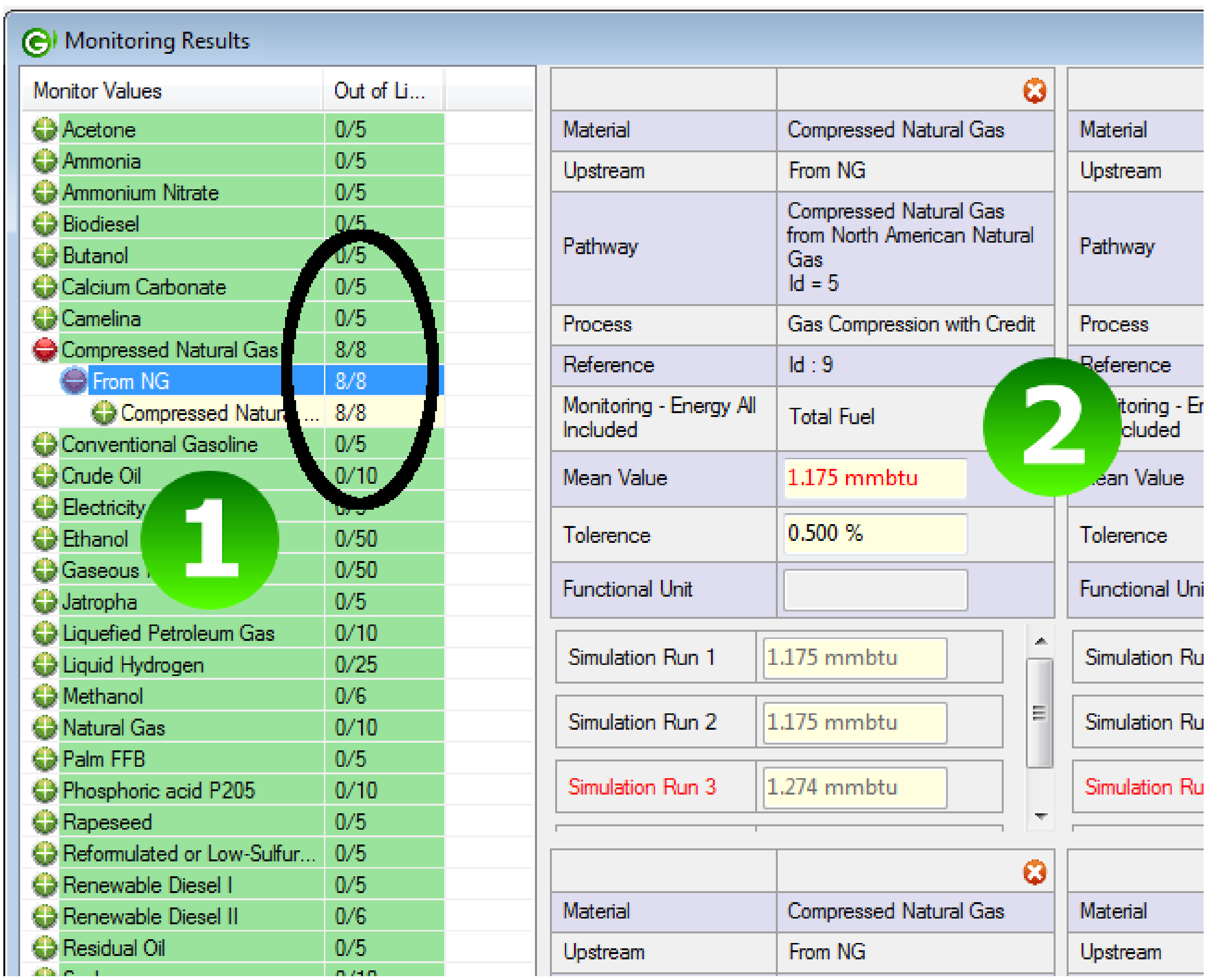


Figure 87: Result Monitored Window

This window is composed of two main zones.

Zone 1 shows a list of all the monitored values organized in a tree structure by Resource, Pathway Mix, and Pathway. The second column shows how many values are out of their boundaries over how many values are being monitored. In the black circled area in Zone 1, you can see that for the Compressed natural gas pathway eight-out-of-eight values are out of their boundaries.

By selecting elements in the tree of monitored values, their actual value and their boundaries can be seen in Zone 2. In the example above, we can see that the total energy used for the process Gas Compression with Credit that is used in the pathway Compressed natural gas from North American natural gas which is a part of the Pathway Mix from NG for the resource Compressed natural gas is out of its boundaries.

The mean value for this monitored result should be 1.175 mmBtu with a tolerance of +- 0.5%. The results of simulation one and simulation two are within the boundaries, but the results of simulation three are outside the boundaries. This is because between simulation two and simulation three the length of a pipeline for this pathway was changed.

To manually add a new monitored value, please refer to the GREET - Well-to-Pump (WTP) - Exploring the Results Section 3.6.3 of this document.

4.4 Preferences

4.4.1 Units

The units displayed can be changed globally using the Global Parameters. See Figure 88. To access the unit Parameters, open the **Preferences** menu, then click on **General Preferences**. The general preferences are made of four tabs. The first tab is **Preferred Units**.

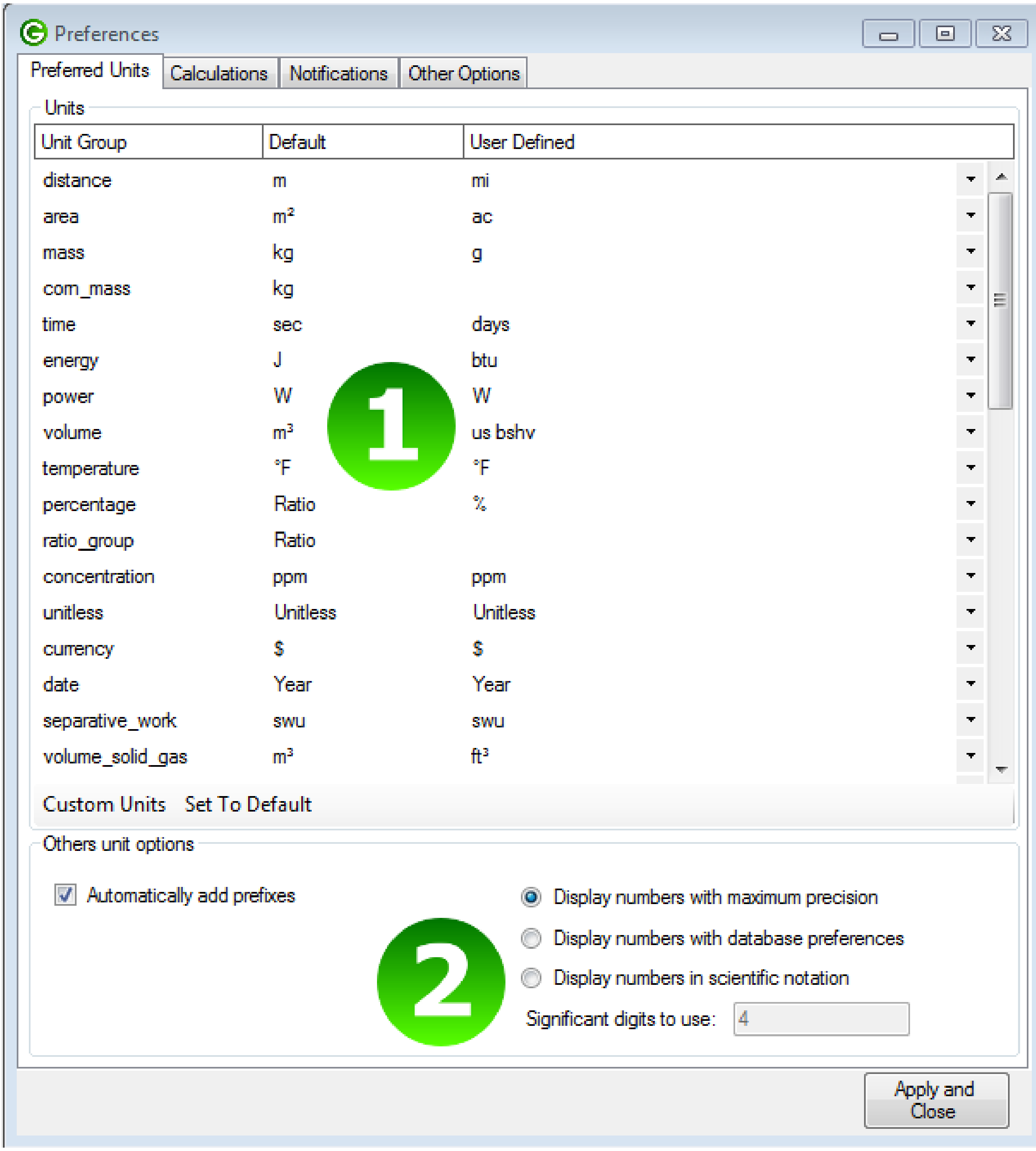


Figure 88: GREET Preferences

The unit preferences options are organized in two zones. Zone 1 represents a list of all the different quantities used in GREET. The list is organized into three columns. The first column indicates the unit type name such as energy or density. The second column indicates the default unit for this type (the default unit is what is used in the calculations) and the last column represents what the user chose to be displayed. As an example, the area is represented internally in meters squared, but the way it is represented in the user interface is acres.

Zone 2 shows some preferences for the display:

- Automatically add prefixes: If checked, the units will be automatically prefixed. A value like 1504266 Joules will be represented as 1.504 MJ.
- Display numbers with maximum precision: If checked, all available digits will be displayed for the results. **NOTE**: All the calculations are performed to 16 digits; although, because the last few digits can contain numerical rounding errors, we chose to cut the results to 14 digits.
- Display numbers with database preference: If checked, digits will be displayed as set in the database. Usually, this is 3 digits after the decimal point for the energy values and 4 digits after the decimal point for emissions.
- Display in scientific notation: If checked, scientific notation will be used everywhere with a user setting for the number of digits to display after the decimal point.

4.4.2 Calculations

In the **Calculations** tab, the rules governing calculations can be modified. See Figure 89. The preferences are organized in two different categories **Automations** and **Running Options**.

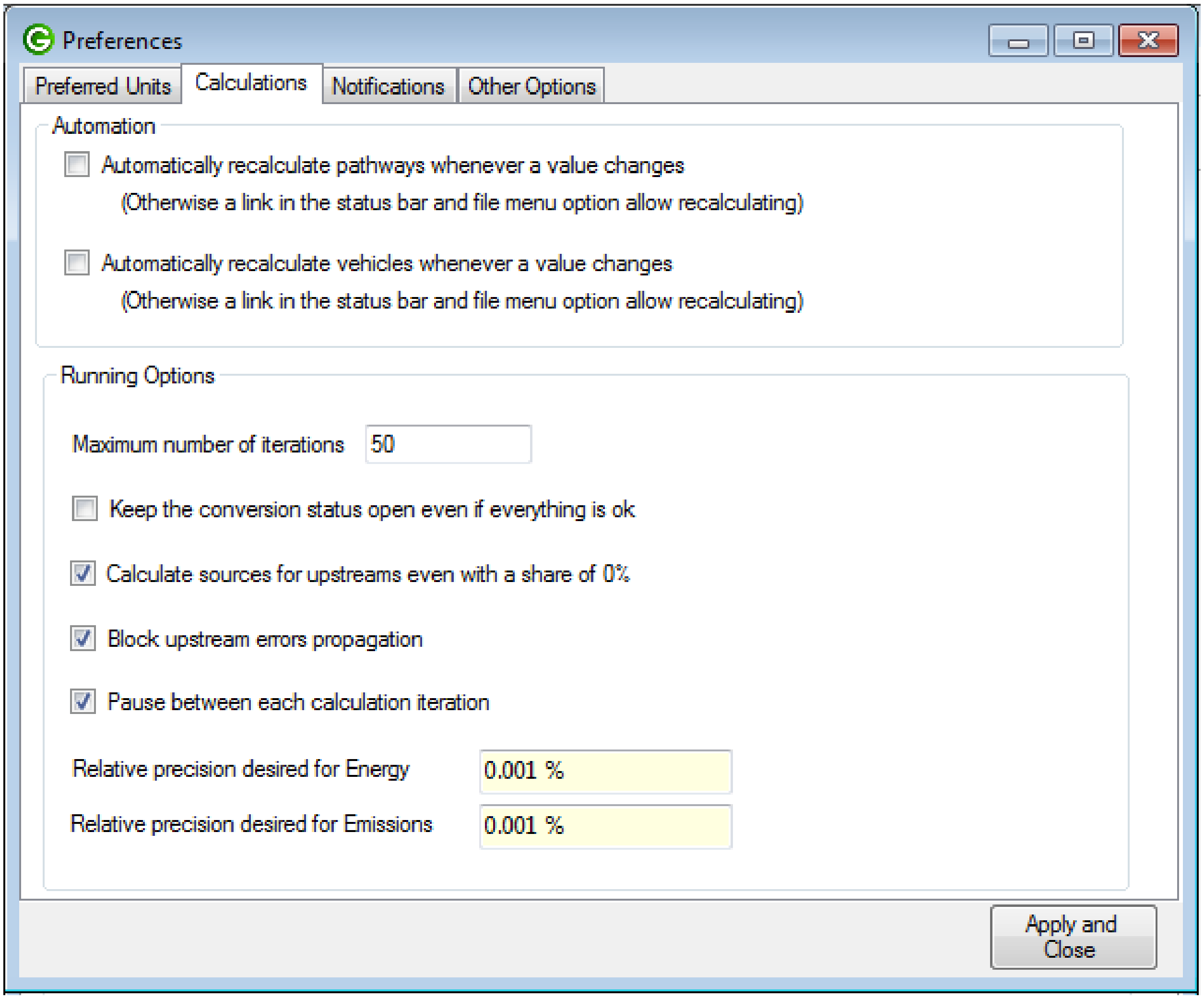


Figure 89: GREET Preferences: Calculations Tab

Automation: The automation section offers an option to automatically re-calculate the results of Well-to-Pump, Pump-to-Wheel, or both, automatically; whenever a parameter is changed. We do not advise you to use this unless you have a powerful computer that is capable of processing the calculations in about a second or two.

Running options: The **Maximum number of iterations** is the maximum limit of iterations done by the software. Usually the convergence is obtained at about 10-15 iterations, depending on the relative precision defined at the bottom of this options group. If the calculations are not converging (because of some issues with the parameters of the simulations), the calculations will be stopped when this number of iterations is reached.

The relative precision for energy and emissions defines when to declare a resource converged. The resource is flagged as converged when the new iteration results are not different by more than relative precision (default value 0.0001%) from the previous one.

When the calculations are running, a form is opened to show which resources are being processed, which ones converged already, and which ones are not calculated. This form usually closes after the calculations are done. By checking Keep the conversion status open even if everything is ok, this form will not be closed.

When a pathway or a Pathway Mix is defined with a 0% share, this pathway or Pathway Mix need not be calculated because it has no impact on the results. However, if it is not calculated, no results will show up in the Well-to-Pump results for it. Therefore, if you don't care about saving calculation time, the option Calculate sources for Pathway Mix with a share of 0% can be enabled to run the calculations on all pathways.

The **Block upstream errors propagation** will prevent the NaN (Not a Number) errors from being propagated. Those kinds of errors usually happen when an accidental division by zero is performed. If this happens and the blocking option is enabled, only the non-valid pathway will be touched, instead of all of them. A pathway with errors will be accounted with an upstream of zero in the system.

When the iterative calculations are running, all processes update their upstream with the upstreams of the previous iteration. The **Pause between each calculation iteration** allows you to follow that mechanism by pausing between each iteration. The results can be seen from the Well-to-Pump results, as usual. However, the vehicles will not be calculated at this point because we must wait for the Well-to-Pump calculations to be completed before running the vehicle calculations.

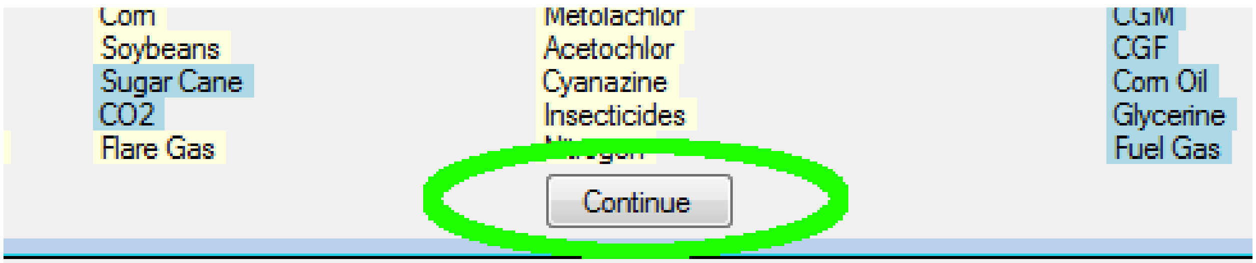


Figure 90: The Continue Button Appears When the Calculations Are Paused between Each Iteration

The relative precision for energy and emissions defines when to stop the iterative calculations. By default, the iterative calculations are stopped when the results from the new iteration are not different by more than 0.001% from the previous one. See Figure 90.

4.4.3 Notifications

GREET shows you messages about special events. For a few of them, you have the ability to choose if you want them to appear or not. Usually, a check box **Do not show this message again** is associated with the message itself. To make those messages appear again, uncheck the appropriate checkboxes in the **Preferences Notifications** pane. See Figure 91.

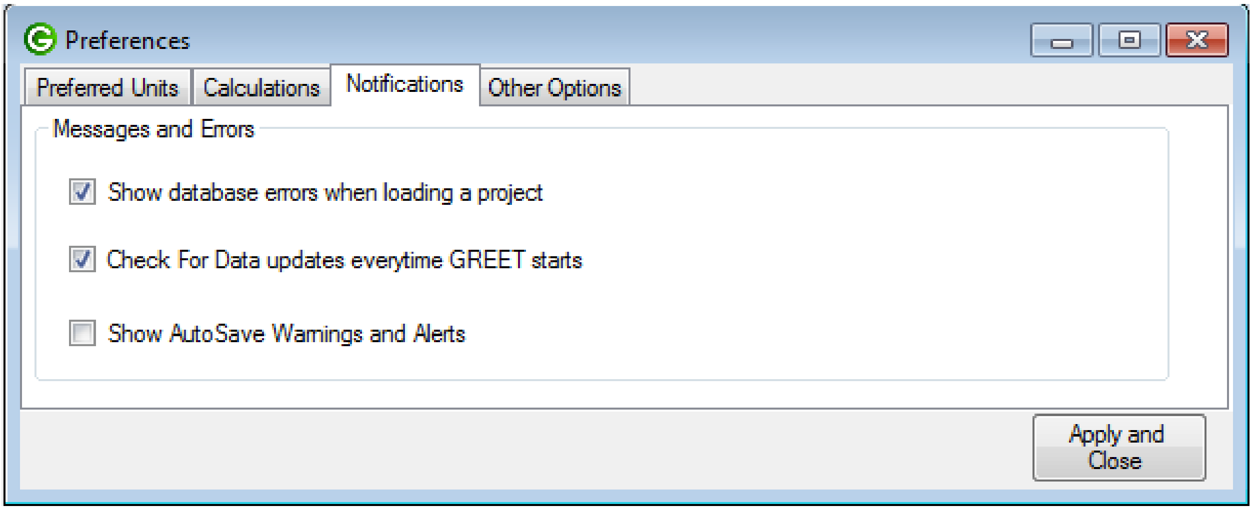


Figure 91: GREET Preferences: Notifications Tab

4.4.4 Other Options

The **Other Options** tab displays two categories of options: **User Interface** features and **Loading/Saving** options. See Figure 92.

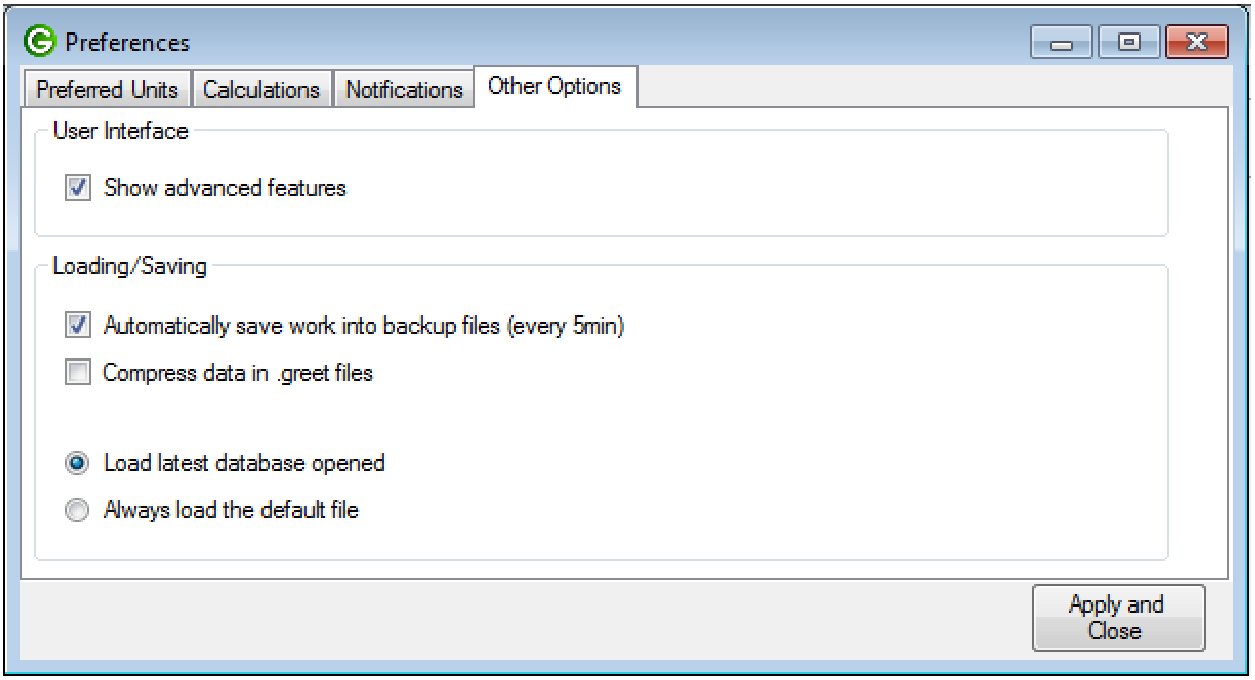


Figure 92: GREET Preferences: Other Options Tab

User interface options: The user interface option contains the check box Show advanced features. If this option is enabled, more information will be shown in the user interface, such as internal IDs for all of the different objects, options for outputting the calculation iterations as a text file, or some database cleaner in the **Data Editor** pane. Those features are more addressed towards developers or advanced users.

Loading/saving options: The auto save option Automatically save work into backup file (every 5 min) will save your actual database into a separated file in the same folder as the original with "autosave" added to the name. If GREET is not closed properly, the work can be opened again from this location by opening Windows Explorer and double clicking on the latest saved backup file.

The **Compress** data in .greet file will compress the data stored in there. This allows you to save smaller files which are easier to send by e-mail and take less space on your hard drive. On the other hand, compressed files cannot be opened with a text editor while uncompressed files can be.

The latest options allow you to choose between loading the latest opened database with GREET or loading the default file from the default location.

5 Troubleshooting

5.1 Starting

- If GREET crashes just after loading or just after you tried to run it, it is possible that you have either:
  - User Preferences issues
  - Database issues

  The first thing to do is to delete your user preferences file. To do so, find the local parameters file of GREET. In Windows 7, this file is located in: C:\Users\YOUR-USERNAME\AppData\Local\ANL. Delete all of the files in this folder and restart GREET.

  If problems persist, try to delete the default database. By doing so, GREET will automatically download the latest database from the web and try to load it. The default database is located in Documents\Greet\ata\Default.greet. Delete this file and restart GREET to download the newest database from the web.

  If problems persist, make sure that you click **OK** when you are prompted to download the software. We include many fixes from one revision to the next one, so you should always use the newest version.

- Cannot read the database

  If errors are thrown after loading the database or into GREET after loading a .greet file, first make sure you have the latest version of the software.

  If you do, the database file is probably corrupted. You can try to fix it manually if the data is not compressed. If this is not possible, delete this file and use the .greet file provided by the server when clicking on the **Database** menu. Then update or remove the default file and start GREET as explained in the paragraph above.

5.2 Reporting an Issue

With GREET, you will most of the time see the Error Report window appear, if you have an error. See Figure 93. It is very helpful if you provide as much detail as possible about what happened just before the crash — especially which buttons you clicked and which options you've changed.

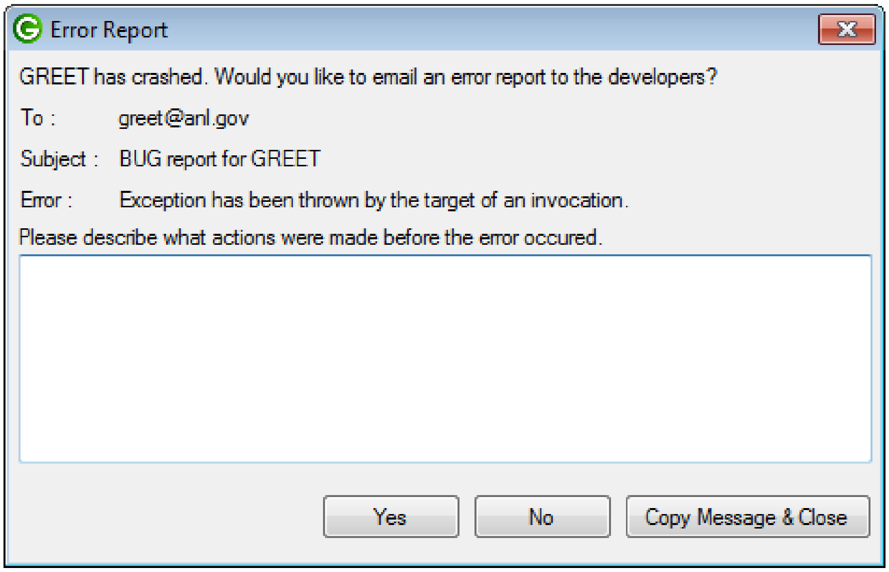


Figure 93: Error Reporting Form

By filling out this form and sending us an e-mail, you will help the entire GREET community.

NOTE: If you are using a web-based application as your default e-mail client, such as Gmail, sending the e-mail might fail. If this is the case, use the **Copy Message and Close** button. Then paste the message into your e-mail client.

For more information on user support visit `http://greet.es.anl.gov/greet/support/`.

References

[1] *GREET Mathematical Model*, Argonne National Laboratory Technical Report 2012.