User Guide

Center for Transportation Research
Energy Systems Division
Argonne National Laboratory

October 7, 2016

Contents

1 Installation

GREET is available at `http://greet.es.anl.gov/greet/setup2015/greet.application`.

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

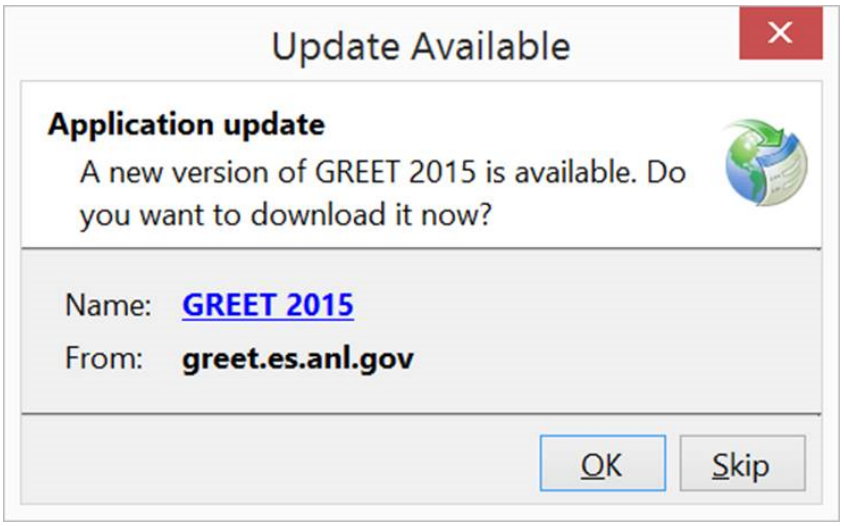


Figure 1: Installation Prompt

1.1 Installation Folder

The GREET executable file is stored under the personal user account folder. The location of GREET's executable file has no relevance unless you need to use GREET's command line feature. For more information on this feature review the "Command Line Tool" section of this manual.

2 Getting GREET Up and Running

2.1 Starting GREET

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

When you first start GREET a registration form will appear. Please complete the requested registration information, check the "I agree with the terms and conditions" check box, then click "Register" to continue. See Figure 2.

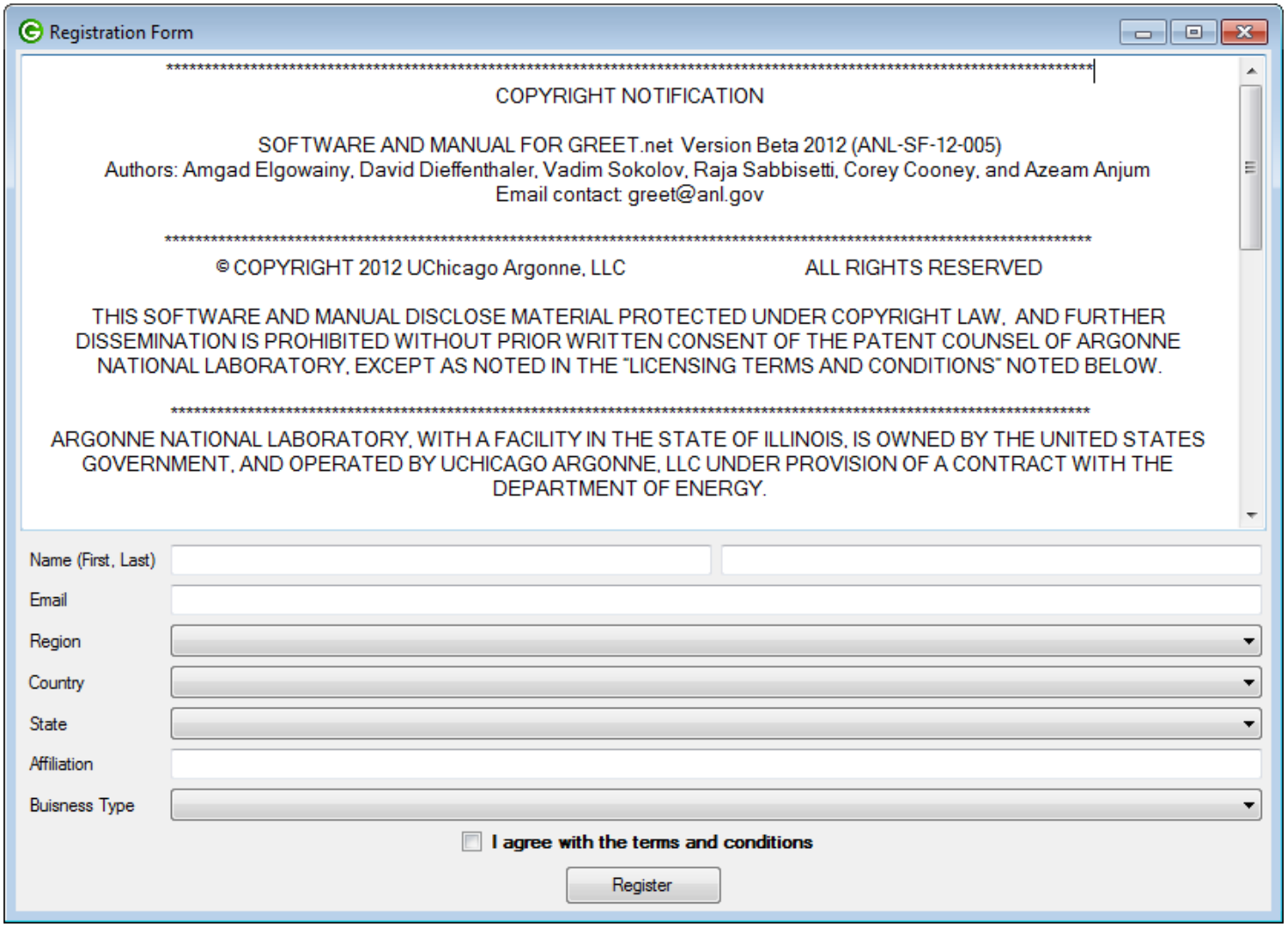


Figure 2: GREET Registration Form

GREET is installed with a default data file. This data file is named Default.greet and will be placed in your My Documents\GREET\Data folder.

If the data file is old and a new version exists, a message box will offer you the opportunity to download the latest available data.

3 GREET Data (Modifications and Simulations)

All GREET data is contained, by default, in a data file ending with the extension .greet.

3.1 Opening a Project

By default, when GREET is started, the default data file located in your My Documents\GREET\Data\Default.greet is loaded.

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

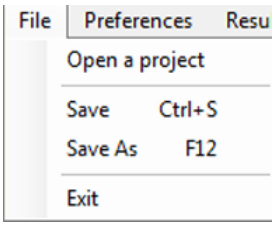


Figure 3: GREET File Menu

3.2 Saving a Project

To save changes to a file, click **Save**. **Save As** will save the data in a newly created data file.

3.3 The .greet Files

GREET uses .greet project files to store data. By double clicking a .greet project files GREET will automatically open the project file in GREET. 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 XML format in the file. GREET files can be opened by any text editor. However, the Default.greet text file may be compressed, depending on the options property settings in (see Section 4.4.4). A compressed text file will not be readable by humans but is readable by greet. If your intention is to manually open, read, and/or alter this project text file you must ensure the GREET options property settings are set to working with project text files that are non-compressed.

NOTE: Be aware that making any modifications to any .greet data file, outside of the GREET interface, carries the possibility of breaking data relationships resulting in a corrupted project file.

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:

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

3.5 Simulation Parameters Pane

The **Simulation Parameters** main pane contains parameters that are reused in many places though the 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).

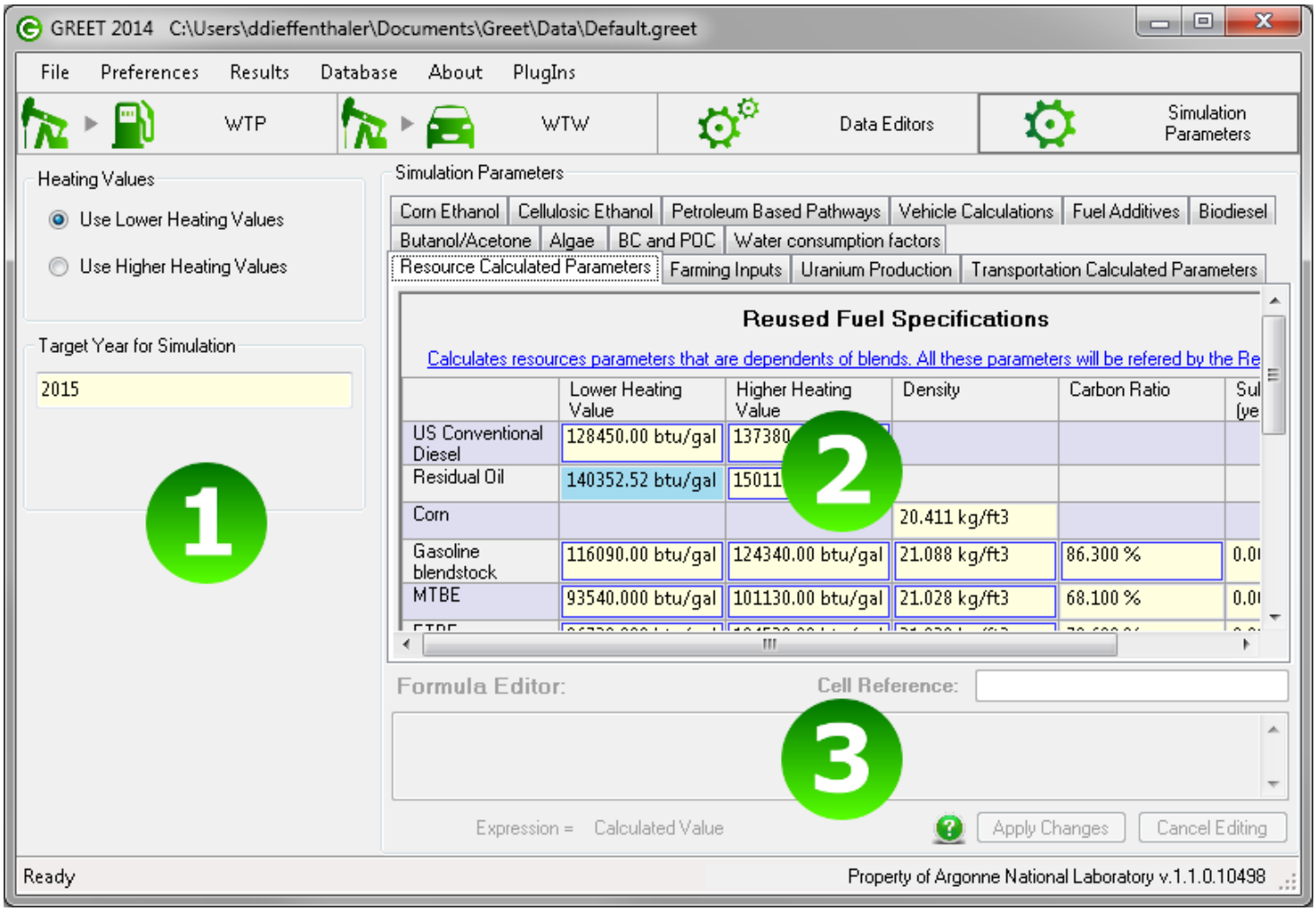


Figure 6: Global Parameters Main Pane

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

Zone 1 shows simulation parameter. 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 target year is used to select an appropriate parameter value when time series are available.

Zone 2 shows the tabs that contain input parameters organized into tables.

Zone 3 shows the area to change a formula used in a cell of a parameter.

A 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 3.5.3).

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 default 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 About the Value

Information about the value can be seen and modified by right clicking on the value and selecting "About this value". A pop-up window will appear which contains various information about the value.

- **Modified By** States the last user(username of the windows login) who modified the value.
- **Modified On** States the last time when the value was modified.
- **Value Identifier** The unique ID of the parameter, which can be changed to define your own name. Identifier helps in creating and visualizing the formula.
- **Notes** Helps the users documenting source of the data. This field can be changed at any time by the user.

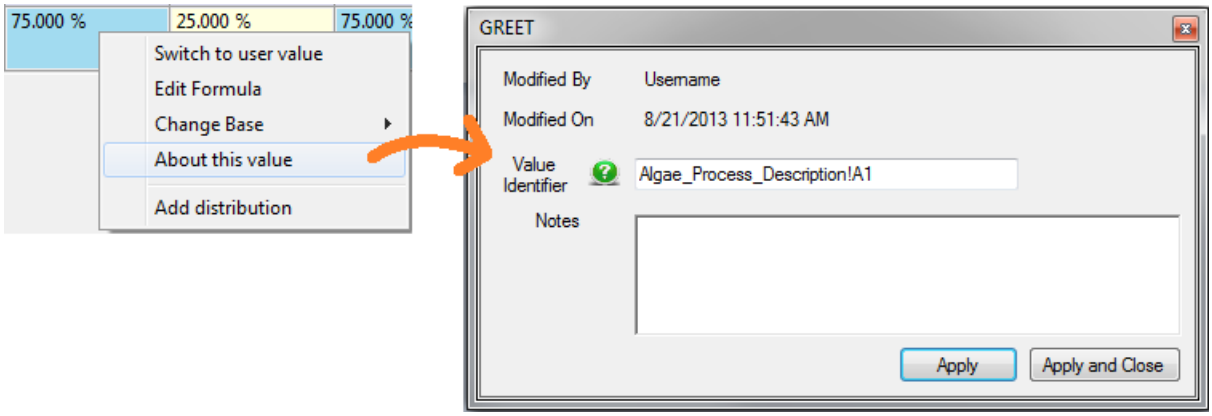


Figure 8: About this Value

3.5.3 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 9). 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]".

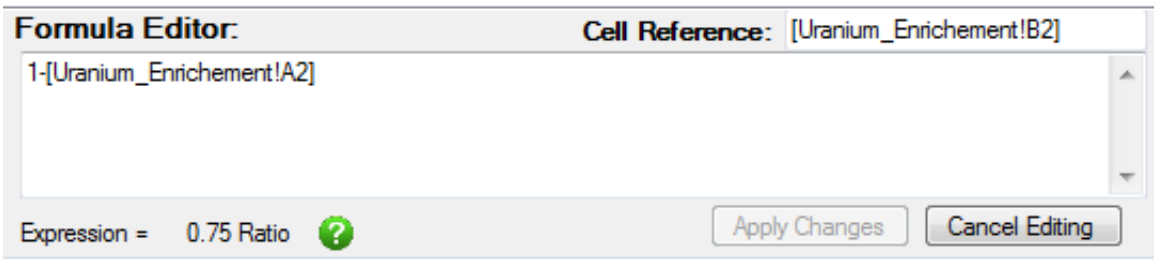


Figure 9: Formula Editor

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.

Another way to create a formula is to use the autocompletion feature. Autocompletion feature can be activated by typing "[" + first two letters of the value identifier(3.5.2). A list of values are shown from which the required value can be selected.

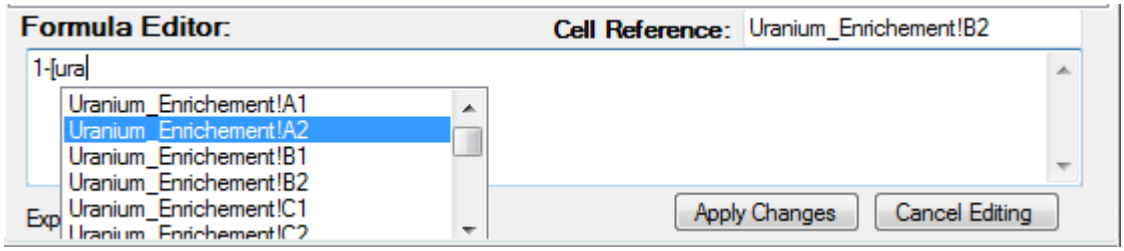


Figure 10: Formula Editor Autocompletion

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

3.5.4 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 11.

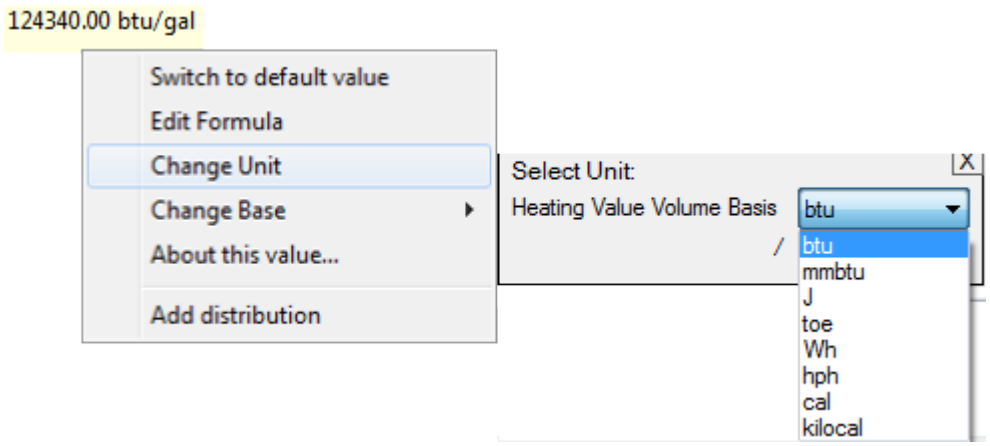


Figure 11: Changing the Unit of a Value in GREET

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.

3.5.5 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 12.

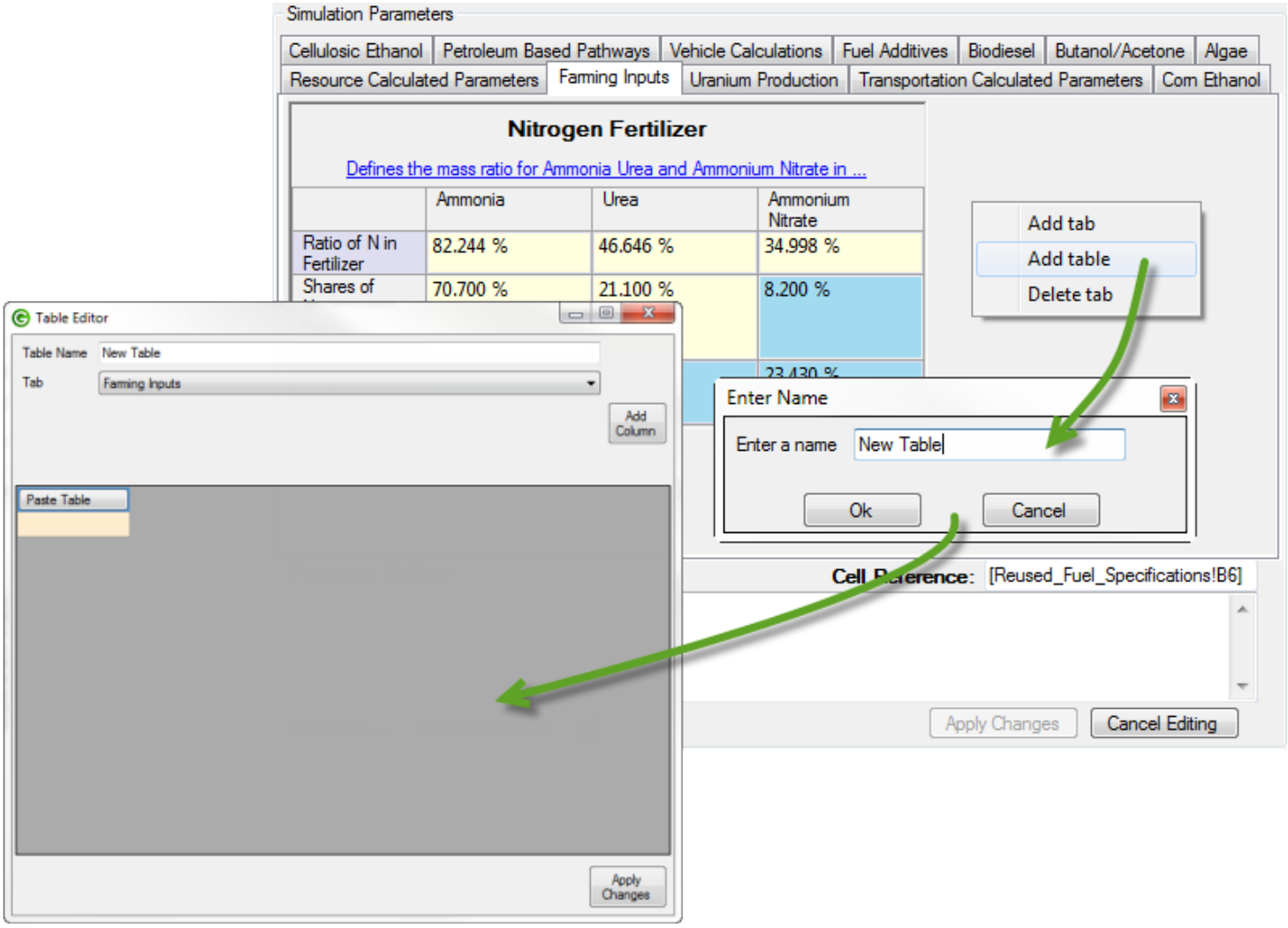


Figure 12: Adding Global Parameters Table

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 12). 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.

3.5.6 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 12).

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, or can be manually started by hitting the F9 key.

3.6.1 Exploring the Structure of Pathways and Pathway Mixes

To explore the results in GREET, browse through the products tree to find a pathway or a pathway mix you are interested in.

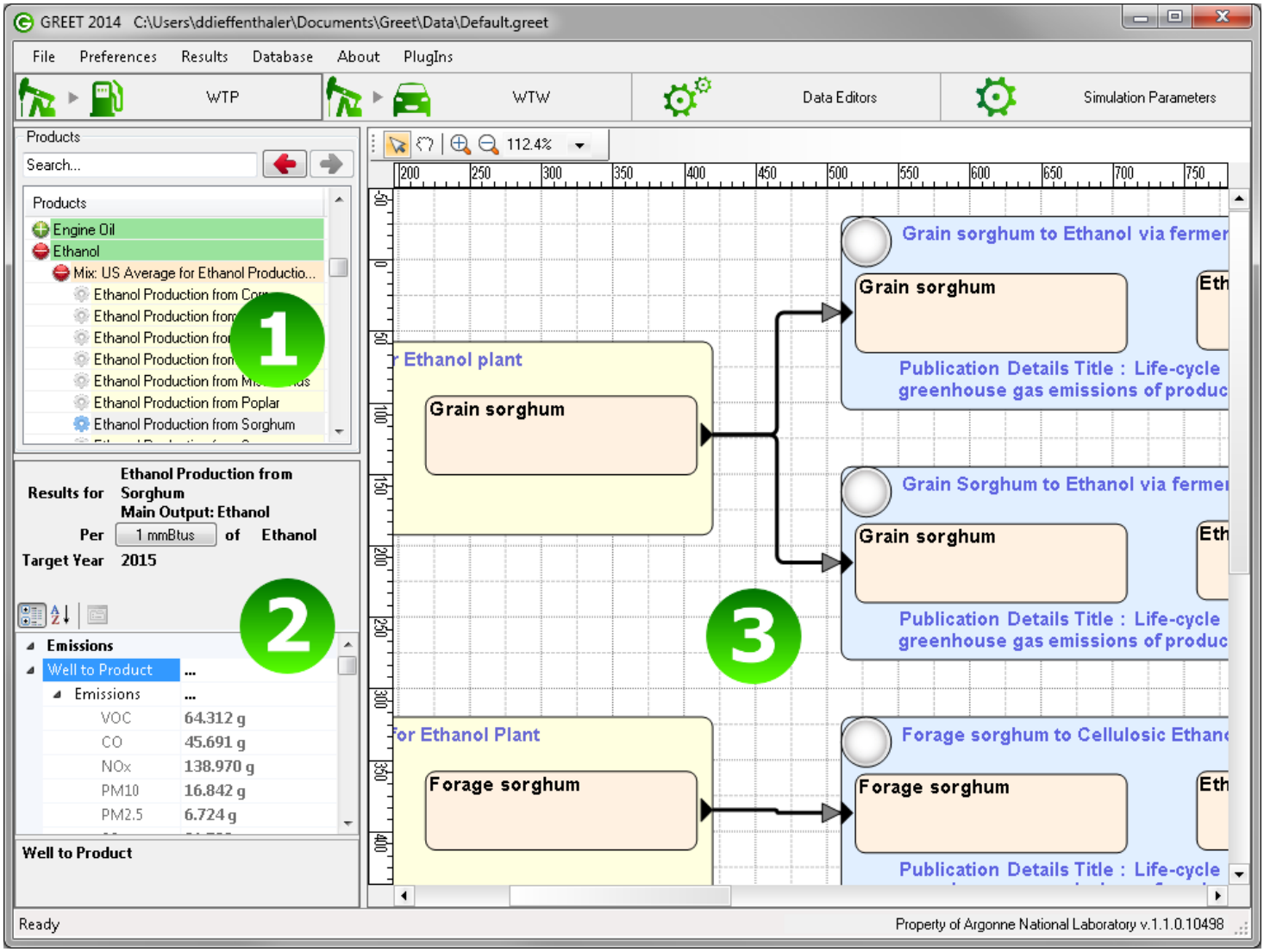


Figure 13: Well-to-Pump Pane Showing the Representation of a Pathway

The WTP pane has 3 zones, as shown on Figure 14. Zone 1 represents the products 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, "Ethanol Production From Sorghum" is selected and represented in Zone 3. Zone 3 shows either a representation of the pathway or a pathway mix. Figure 13 shows a pathway represented in the explorer and Figure 14 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 14.

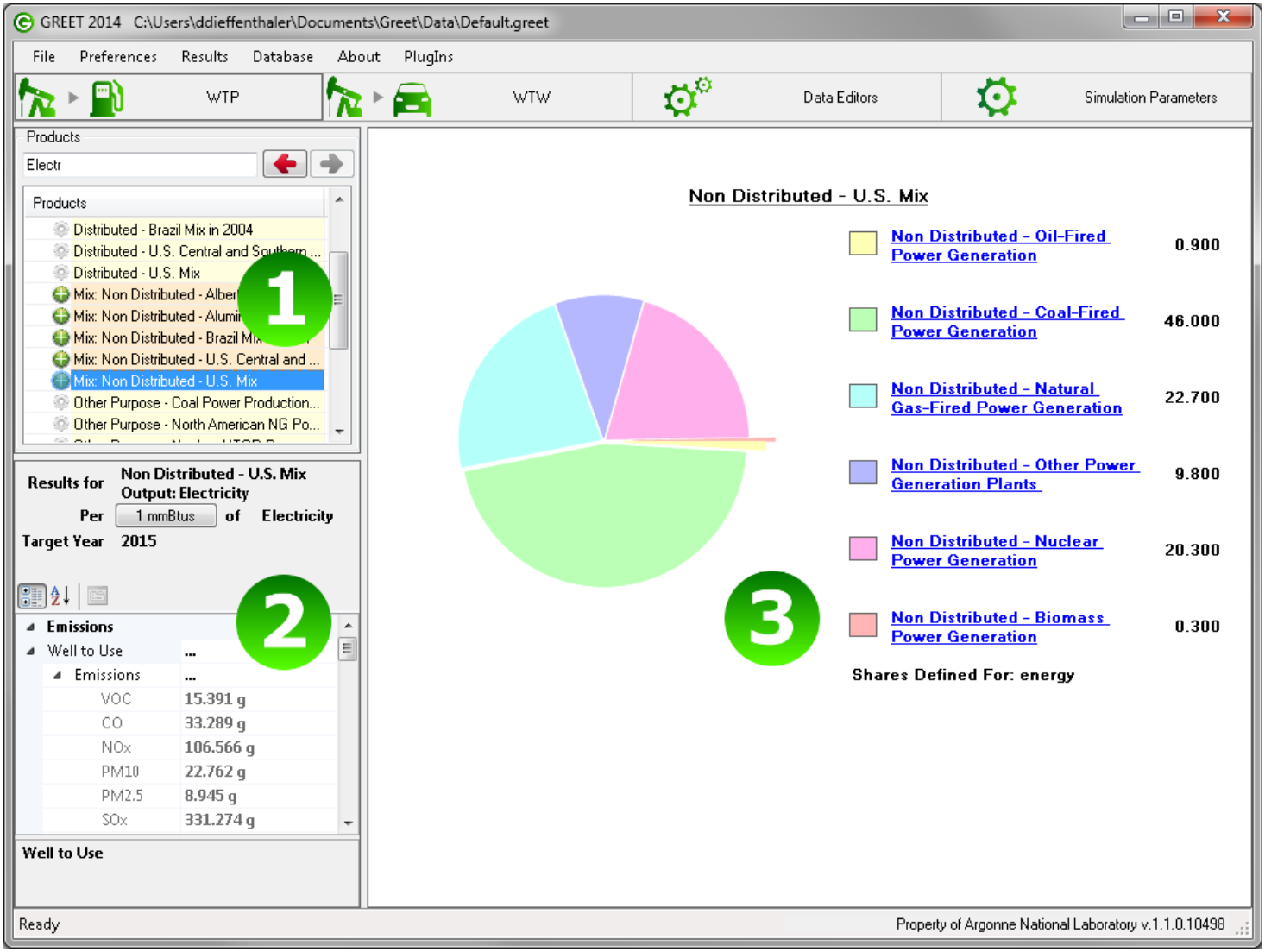


Figure 14: Electricity produced in the United States averaged before distribution by power lines

The zone 3 on 14 represents the shares in a pathway mix. As it is assumed in the GREET model, electricity in the United States, on average, comes from 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 15 shows a pathway after the user clicked on **Non-Distributed - Natural Gas-Fired Power Generation**.

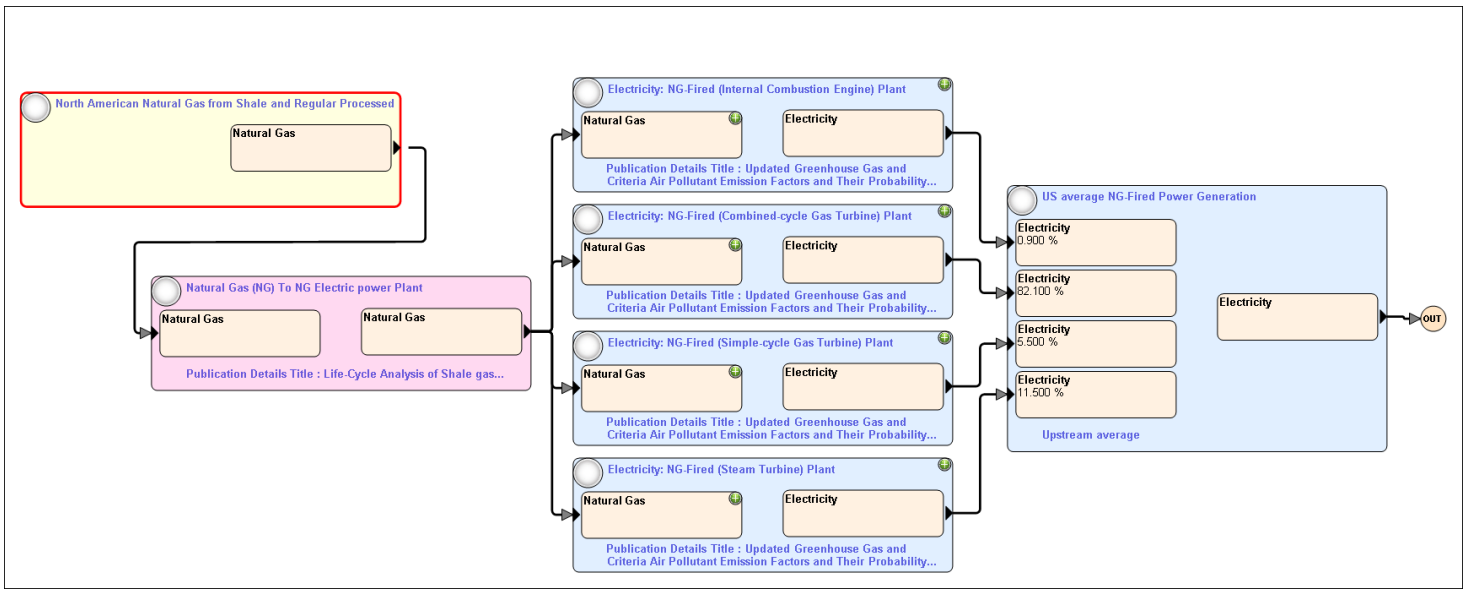


Figure 15: Non Distributed - Natural Gas-Fired Power Generation

The pathway 15 has a feed. The feed, represnted by the yellow box, is another pathway from which the main output is connected to the input of a transportation process (the pink box). In Figure 15, the upstream for this pathway is the natural gas from North American Natural Gas from Shale and Regular Processed. 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.

Some of the processes in Figure 15 are collapsed. That means that we hide the inputs that are not connected to another process in this pathway. To get an expanded view, double click on the + signs on the top right corner of the processes.

3.6.2 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.

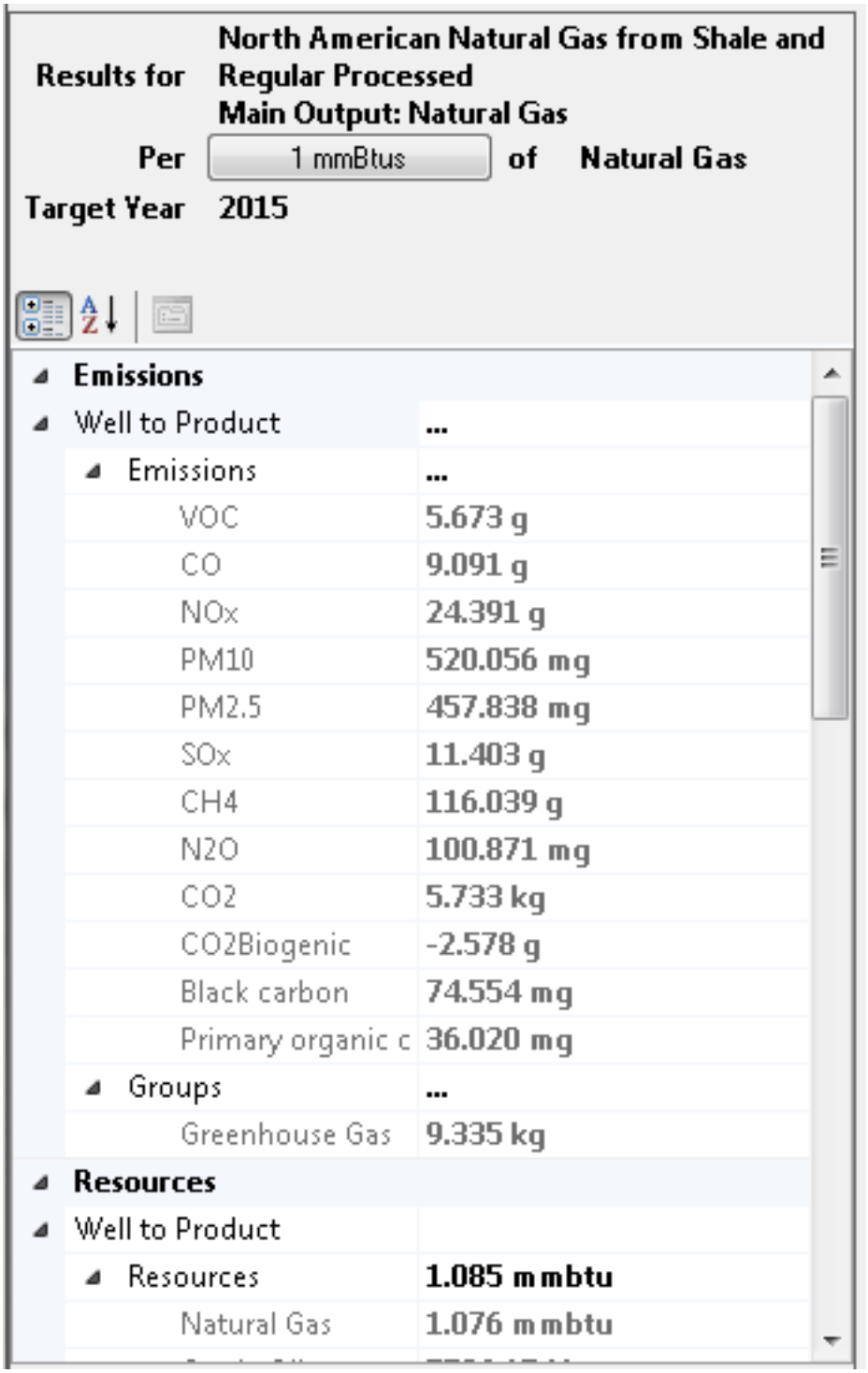


Figure 16: Properties Display for Combined Cycle Electricity Production from Natural Gas

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 13).
- Pathway: When a pathway is selected in the Pathway-Mix tree (Zone 1 of Figure 13).
- Process: When a process is selected in the pathway representation (Zone 3 of Figure 13).
- Step: When a transportation step is selected within a transportation process (Zone 3 of Figure 13).
- Input: When an input is selected in an expanded stationary process (Zone 3 of Figure 13).
- Technology: When a technology is selected in an expanded stationary process input (Zone 3 of Figure 13).

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.

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**. See Figure 17. 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).

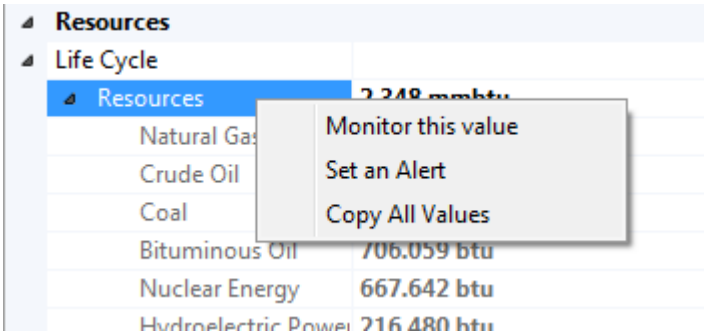


Figure 17: Copying all the Results to the Clipboard

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 17.

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.

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 18.

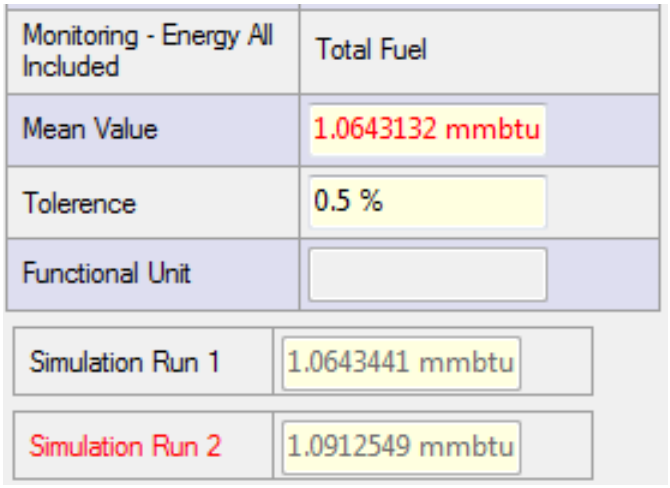


Figure 18: Results after Two Different Simulations, with a Recovery Efficiency Changed on the Second One

When boundaries are defined, a message will appear after the calculation, if one or more values are outside of their limits. Figure 19 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.

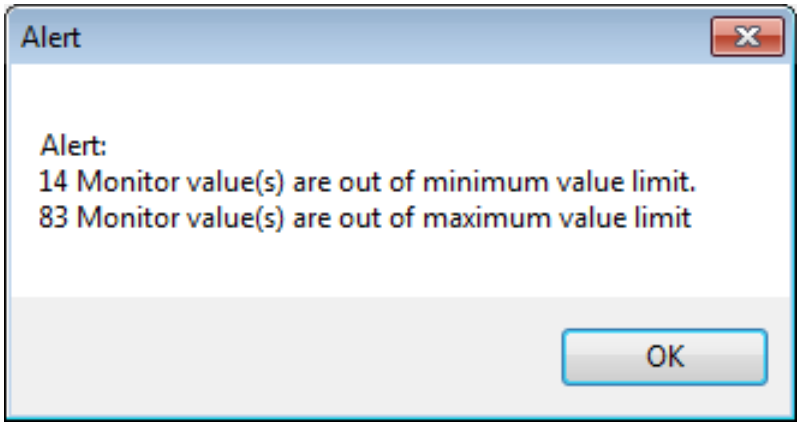


Figure 19: Alert Window

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

3.6.3 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*.

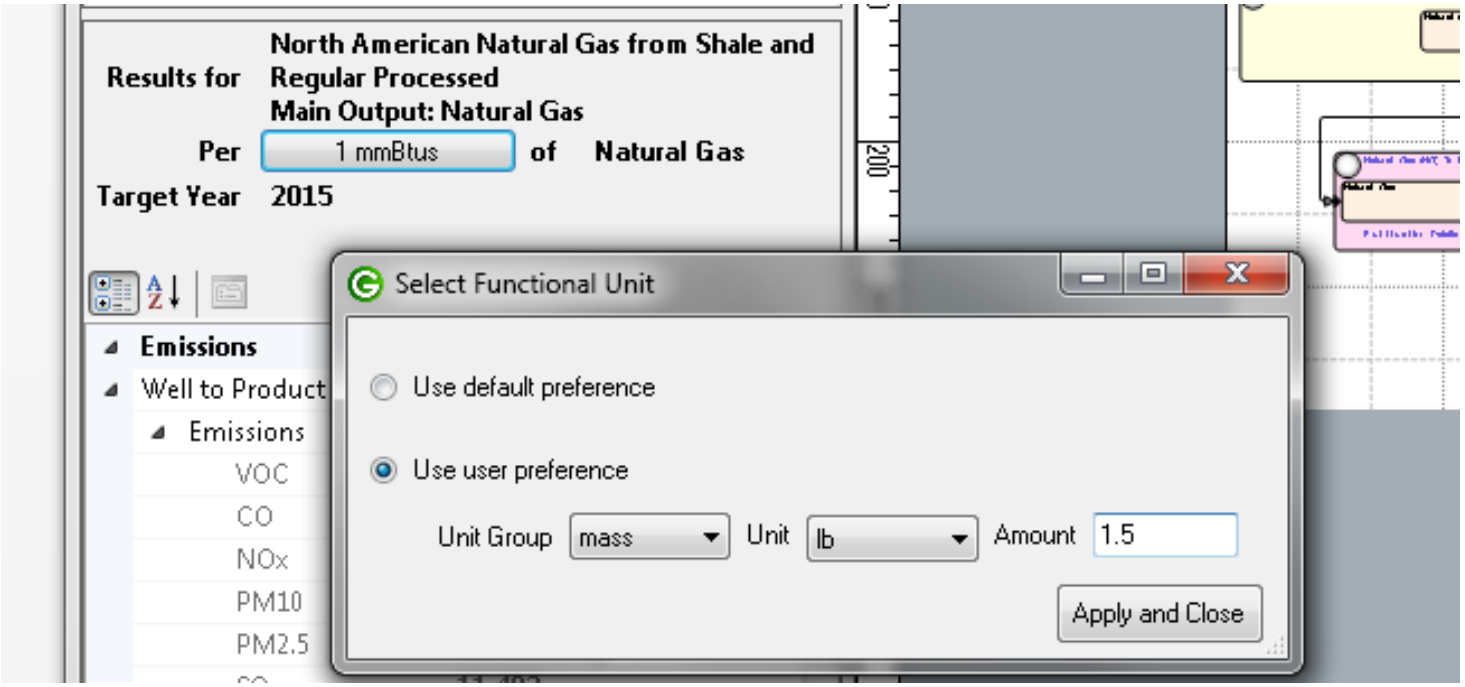


Figure 20: Changing the Functional Unit from the Default 1mmBtu to 1.5 lb

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 20, 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 21.

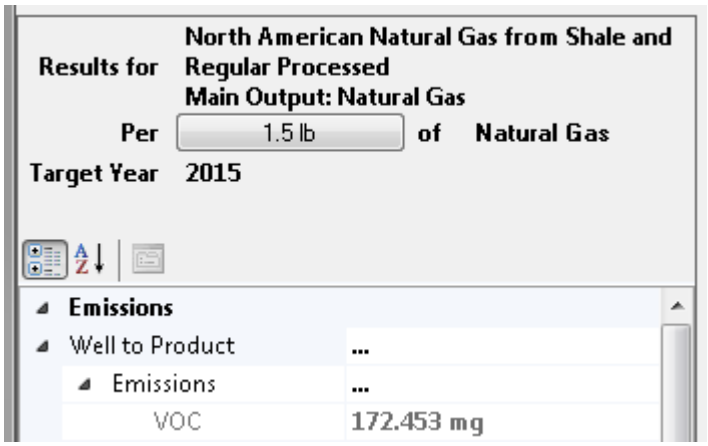


Figure 21: The Results Are Now Displayed Per 1.5 lb of Oil for This Process

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 22. The Well-to-Wheels results represent what the life-cycle impact is of a vehicle

technology in terms of energy and emissions.

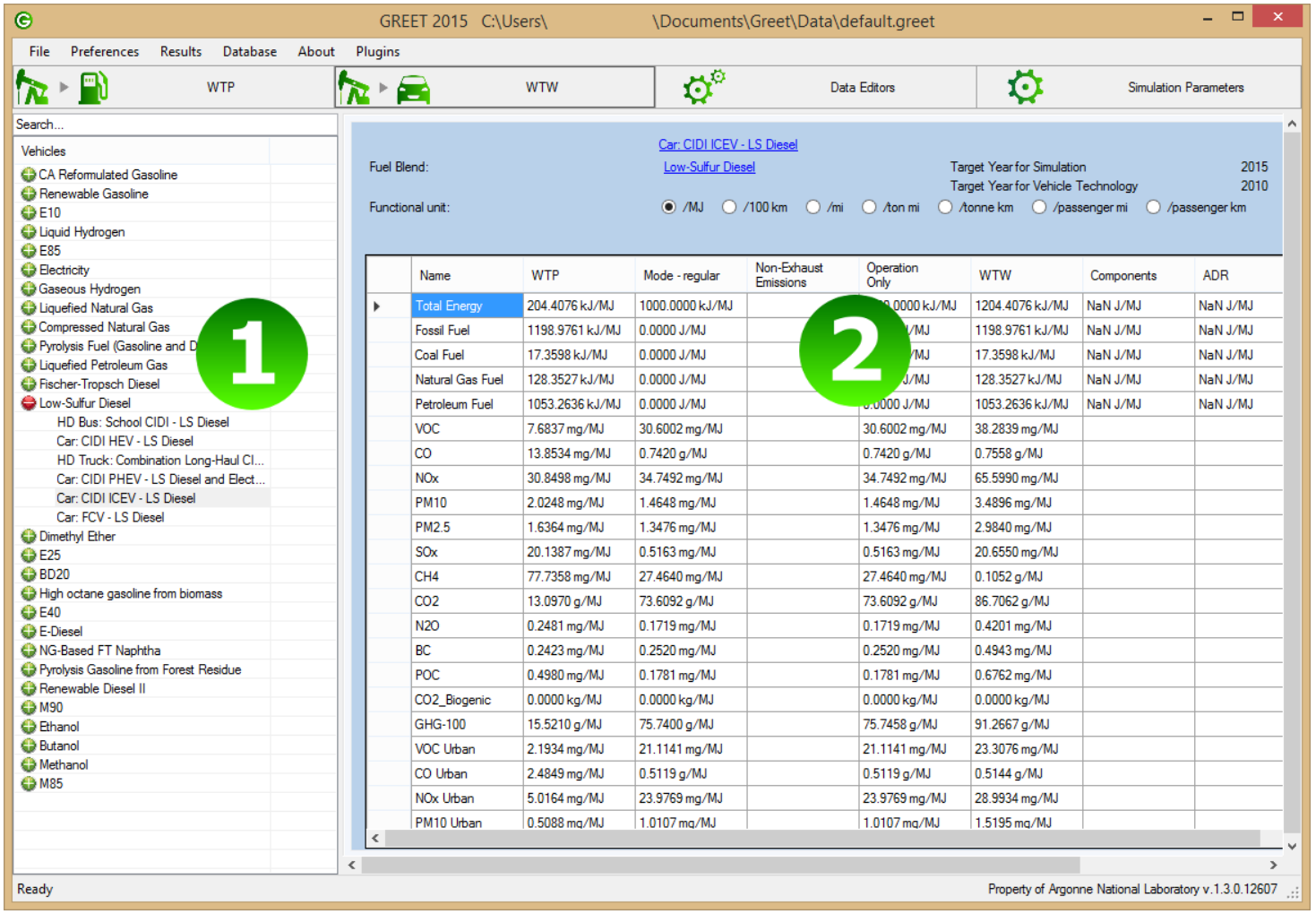
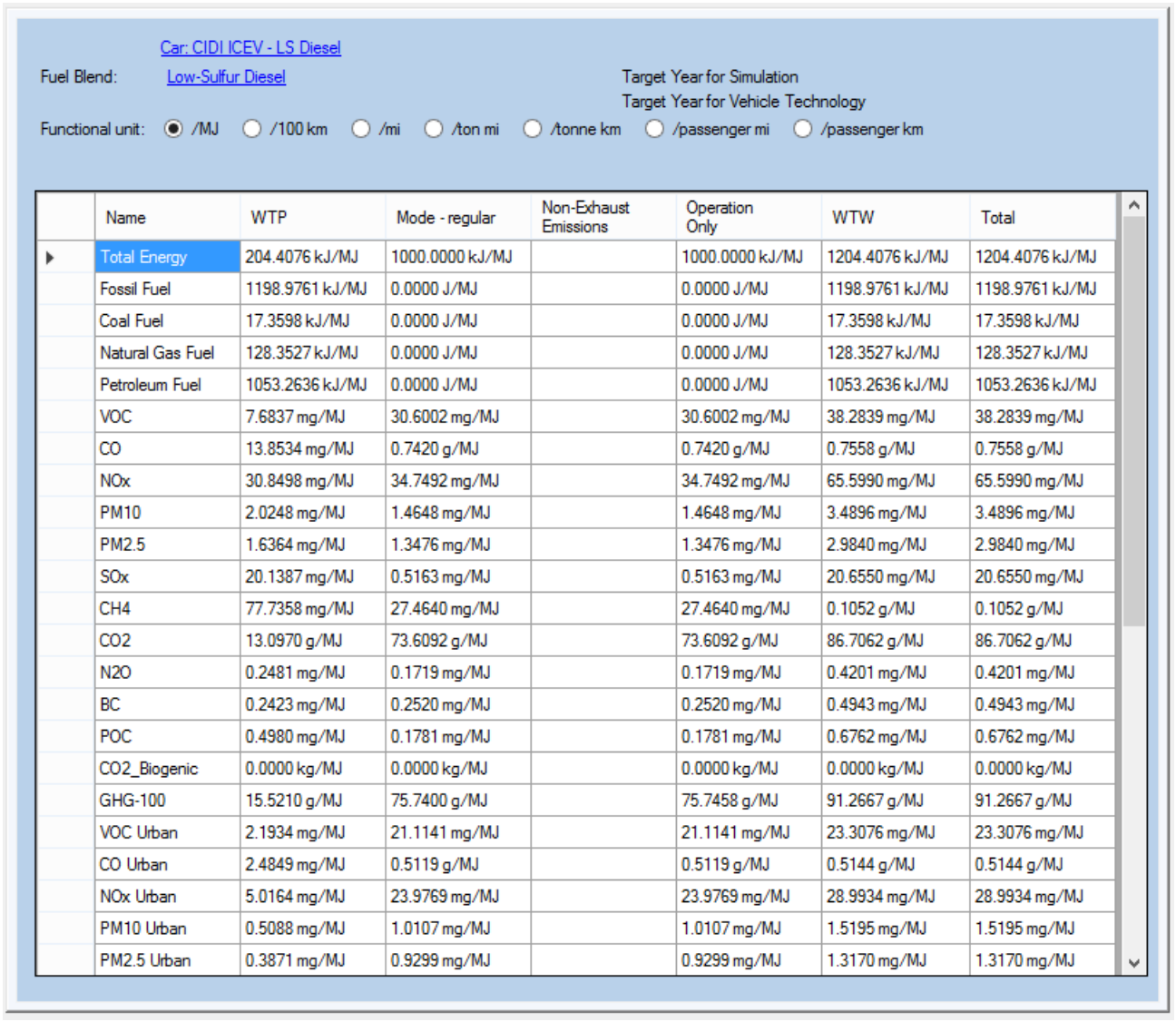


Figure 22: Well-to-Wheels Pane Showing the conventional Diesel vehicle

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.



	Name	WTP	Mode - regular	Non-Exhaust Emissions	Operation Only	WTW	Total
▶	Total Energy	204.4076 kJ/MJ	1000.0000 kJ/MJ		1000.0000 kJ/MJ	1204.4076 kJ/MJ	1204.4076 kJ/MJ
	Fossil Fuel	1198.9761 kJ/MJ	0.0000 J/MJ		0.0000 J/MJ	1198.9761 kJ/MJ	1198.9761 kJ/MJ
	Coal Fuel	17.3598 kJ/MJ	0.0000 J/MJ		0.0000 J/MJ	17.3598 kJ/MJ	17.3598 kJ/MJ
	Natural Gas Fuel	128.3527 kJ/MJ	0.0000 J/MJ		0.0000 J/MJ	128.3527 kJ/MJ	128.3527 kJ/MJ
	Petroleum Fuel	1053.2636 kJ/MJ	0.0000 J/MJ		0.0000 J/MJ	1053.2636 kJ/MJ	1053.2636 kJ/MJ
	VOC	7.6837 mg/MJ	30.6002 mg/MJ		30.6002 mg/MJ	38.2839 mg/MJ	38.2839 mg/MJ
	CO	13.8534 mg/MJ	0.7420 g/MJ		0.7420 g/MJ	0.7558 g/MJ	0.7558 g/MJ
	NOx	30.8498 mg/MJ	34.7492 mg/MJ		34.7492 mg/MJ	65.5990 mg/MJ	65.5990 mg/MJ
	PM10	2.0248 mg/MJ	1.4648 mg/MJ		1.4648 mg/MJ	3.4896 mg/MJ	3.4896 mg/MJ
	PM2.5	1.6364 mg/MJ	1.3476 mg/MJ		1.3476 mg/MJ	2.9840 mg/MJ	2.9840 mg/MJ
	SOx	20.1387 mg/MJ	0.5163 mg/MJ		0.5163 mg/MJ	20.6550 mg/MJ	20.6550 mg/MJ
	CH4	77.7358 mg/MJ	27.4640 mg/MJ		27.4640 mg/MJ	0.1052 g/MJ	0.1052 g/MJ
	CO2	13.0970 g/MJ	73.6092 g/MJ		73.6092 g/MJ	86.7062 g/MJ	86.7062 g/MJ
	N2O	0.2481 mg/MJ	0.1719 mg/MJ		0.1719 mg/MJ	0.4201 mg/MJ	0.4201 mg/MJ
	BC	0.2423 mg/MJ	0.2520 mg/MJ		0.2520 mg/MJ	0.4943 mg/MJ	0.4943 mg/MJ
	POC	0.4980 mg/MJ	0.1781 mg/MJ		0.1781 mg/MJ	0.6762 mg/MJ	0.6762 mg/MJ
	CO2_Biogenic	0.0000 kg/MJ	0.0000 kg/MJ		0.0000 kg/MJ	0.0000 kg/MJ	0.0000 kg/MJ
	GHG-100	15.5210 g/MJ	75.7400 g/MJ		75.7458 g/MJ	91.2667 g/MJ	91.2667 g/MJ
	VOC Urban	2.1934 mg/MJ	21.1141 mg/MJ		21.1141 mg/MJ	23.3076 mg/MJ	23.3076 mg/MJ
	CO Urban	2.4849 mg/MJ	0.5119 g/MJ		0.5119 g/MJ	0.5144 g/MJ	0.5144 g/MJ
	NOx Urban	5.0164 mg/MJ	23.9769 mg/MJ		23.9769 mg/MJ	28.9934 mg/MJ	28.9934 mg/MJ
	PM10 Urban	0.5088 mg/MJ	1.0107 mg/MJ		1.0107 mg/MJ	1.5195 mg/MJ	1.5195 mg/MJ
	PM2.5 Urban	0.3871 mg/MJ	0.9299 mg/MJ		0.9299 mg/MJ	1.3170 mg/MJ	1.3170 mg/MJ

Figure 23: Well-to-Wheel Results for Conventional Diesel vehicle

Operation Only The "Operation Only" column (Figure 23) shows the total energy and emissions associated with vehicle operation only (i.e., the sum of "Mode"(s) and "Non-Exhaust Emissions"); those results do not include any upstream energy or emissions due to fuel manufacturing. "Mode - " columns that are showing the consumption per each operational mode of the vehicle. "Non-Exhaust Emissions" are emissions such as VOC evaporation, PM emissions from brake, tire, and wear, etc. To check or edit the parameters used to calculate those values, please refer to the Vehicles Editor in section 3.8.8.

WTP The upstream column is the "WTP" (Well-to-Pump) column which represents the upstream associated with fuel production and distribution.

Vehicle Construction If the vehicle definition has vehicle construction information (optional) such as transmission, wheels, fluids, battery, etc., these vehicles construction results will be part of the broad Well-to-Wheel results. In order to calculate their impact we use the total vehicle lifetime mileage and the upstream necessary to build each of these items.

Each vehicle construction item has an individual column, and columns "Components", "ADR" (Assembly Disposal and Recycling), "Fluids", "Battery", and "Others" represent the aggregated results of the corresponding groups of items.

WTW and Total The Total column shows the vehicle operation, the upstream associated with the fuel production and distribution, and the vehicle construction results (if any) added together; these are the Well-to-Wheels results in the broad sense. Please note that here we use column "WTW" to represent the Well-to-Wheel results in the narrow sense which is only

the sum of "WTP" and "Operation Only"; and use column "Total" to represent the Well-to-Wheel results in the broad sense which further includes vehicle construction results.

Clicking one of the fuel links will open the pathway or the pathway mix that was used to produce that fuel; and clicking the vehicle link will open the vehicle editor of the current vehicle.

The functional unit can be on of many options:

- "/MJ" which uses 1 Megajoule of fuel used by the engine as the functional unit (thus does not take into account the vehicle fuel economy)
- "/100km" which uses 100 kilometers as the functional unit
- "/mi" which uses 1 statute mile as the functional unit
- "ton mi", this option is only available if a payload is defined for the vehicle and uses short ton*mile as the functional unit
- "tonne km", this option is only available if a payload is defined for the vehicle and uses short-tonne*kilometer as the functional unit
- "passenger mi", this option is only available if a passenger number is defined for the vehicle and uses passenger*mile as the functional unit
- "passenger km", this option is only available if a passenger number is defined for the vehicle and uses passenger*kilometer as the functional unit

Note: All "miles" unit are assumed to be "statute miles" in the WTW results panel.

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 24.

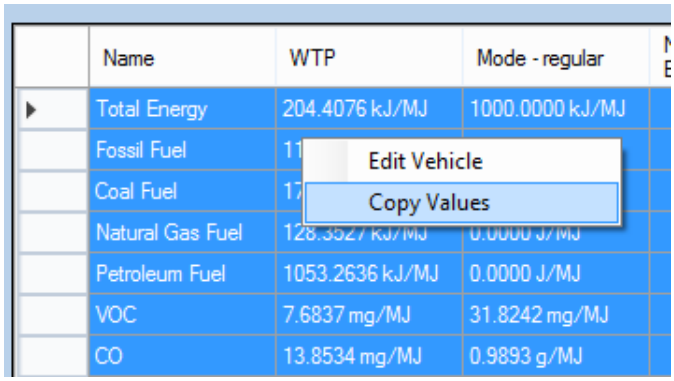
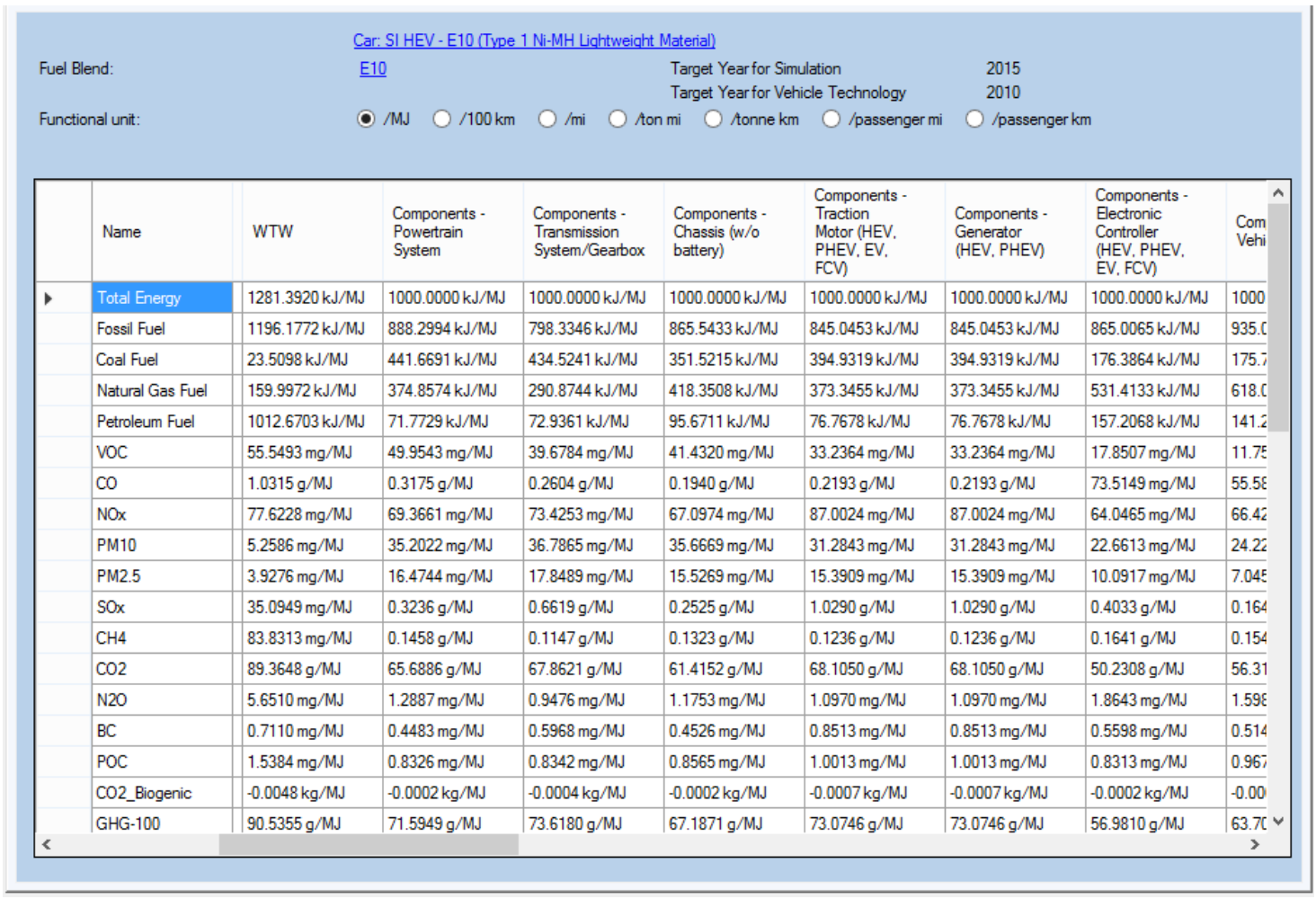


Figure 24: Copying the Results of a Vehicle

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



Name	WTW	Components - Powertrain System	Components - Transmission System/Gearbox	Components - Chassis (w/o battery)	Components - Traction Motor (HEV, PHEV, EV, FCV)	Components - Generator (HEV, PHEV)	Components - Electronic Controller (HEV, PHEV, EV, FCV)	Com Vehi
Total Energy	1281.3920 kJ/MJ	1000.0000 kJ/MJ	1000.0000 kJ/MJ	1000.0000 kJ/MJ	1000.0000 kJ/MJ	1000.0000 kJ/MJ	1000.0000 kJ/MJ	1000
Fossil Fuel	1196.1772 kJ/MJ	888.2994 kJ/MJ	798.3346 kJ/MJ	865.5433 kJ/MJ	845.0453 kJ/MJ	845.0453 kJ/MJ	865.0065 kJ/MJ	935.0
Coal Fuel	23.5098 kJ/MJ	441.6691 kJ/MJ	434.5241 kJ/MJ	351.5215 kJ/MJ	394.9319 kJ/MJ	394.9319 kJ/MJ	176.3864 kJ/MJ	175.7
Natural Gas Fuel	159.9972 kJ/MJ	374.8574 kJ/MJ	290.8744 kJ/MJ	418.3508 kJ/MJ	373.3455 kJ/MJ	373.3455 kJ/MJ	531.4133 kJ/MJ	618.0
Petroleum Fuel	1012.6703 kJ/MJ	71.7729 kJ/MJ	72.9361 kJ/MJ	95.6711 kJ/MJ	76.7678 kJ/MJ	76.7678 kJ/MJ	157.2068 kJ/MJ	141.2
VOC	55.5493 mg/MJ	49.9543 mg/MJ	39.6784 mg/MJ	41.4320 mg/MJ	33.2364 mg/MJ	33.2364 mg/MJ	17.8507 mg/MJ	11.75
CO	1.0315 g/MJ	0.3175 g/MJ	0.2604 g/MJ	0.1940 g/MJ	0.2193 g/MJ	0.2193 g/MJ	73.5149 mg/MJ	55.58
NOx	77.6228 mg/MJ	69.3661 mg/MJ	73.4253 mg/MJ	67.0974 mg/MJ	87.0024 mg/MJ	87.0024 mg/MJ	64.0465 mg/MJ	66.42
PM10	5.2586 mg/MJ	35.2022 mg/MJ	36.7865 mg/MJ	35.6669 mg/MJ	31.2843 mg/MJ	31.2843 mg/MJ	22.6613 mg/MJ	24.22
PM2.5	3.9276 mg/MJ	16.4744 mg/MJ	17.8489 mg/MJ	15.5269 mg/MJ	15.3909 mg/MJ	15.3909 mg/MJ	10.0917 mg/MJ	7.045
SOx	35.0949 mg/MJ	0.3236 g/MJ	0.6619 g/MJ	0.2525 g/MJ	1.0290 g/MJ	1.0290 g/MJ	0.4033 g/MJ	0.164
CH4	83.8313 mg/MJ	0.1458 g/MJ	0.1147 g/MJ	0.1323 g/MJ	0.1236 g/MJ	0.1236 g/MJ	0.1641 g/MJ	0.154
CO2	89.3648 g/MJ	65.6886 g/MJ	67.8621 g/MJ	61.4152 g/MJ	68.1050 g/MJ	68.1050 g/MJ	50.2308 g/MJ	56.31
N2O	5.6510 mg/MJ	1.2887 mg/MJ	0.9476 mg/MJ	1.1753 mg/MJ	1.0970 mg/MJ	1.0970 mg/MJ	1.8643 mg/MJ	1.598
BC	0.7110 mg/MJ	0.4483 mg/MJ	0.5968 mg/MJ	0.4526 mg/MJ	0.8513 mg/MJ	0.8513 mg/MJ	0.5598 mg/MJ	0.514
POC	1.5384 mg/MJ	0.8326 mg/MJ	0.8342 mg/MJ	0.8565 mg/MJ	1.0013 mg/MJ	1.0013 mg/MJ	0.8313 mg/MJ	0.967
CO2_Biogenic	-0.0048 kg/MJ	-0.0002 kg/MJ	-0.0004 kg/MJ	-0.0002 kg/MJ	-0.0007 kg/MJ	-0.0007 kg/MJ	-0.0002 kg/MJ	-0.00
GHG-100	90.5355 g/MJ	71.5949 g/MJ	73.6180 g/MJ	67.1871 g/MJ	73.0746 g/MJ	73.0746 g/MJ	56.9810 g/MJ	63.70

Figure 25: Results per component

In order to help understanding how the columns are summed up, here are the basic equations in use for the WTW results:

$$\sum_{m=1}^{modes} M(m) + NonExhaust = OperationOnly$$

$$WTP + OperationOnly = WTW$$

$$WTW + \sum_{c=1}^{components} C(c) + \sum_{a=1}^{ADR} A(a) + \sum_{f=1}^{fluids} F(d) + \sum_{b=1}^{battery} B(b) + \sum_{o=1}^{others} O(o) = TOTAL$$

3.8 Data Editors

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**.

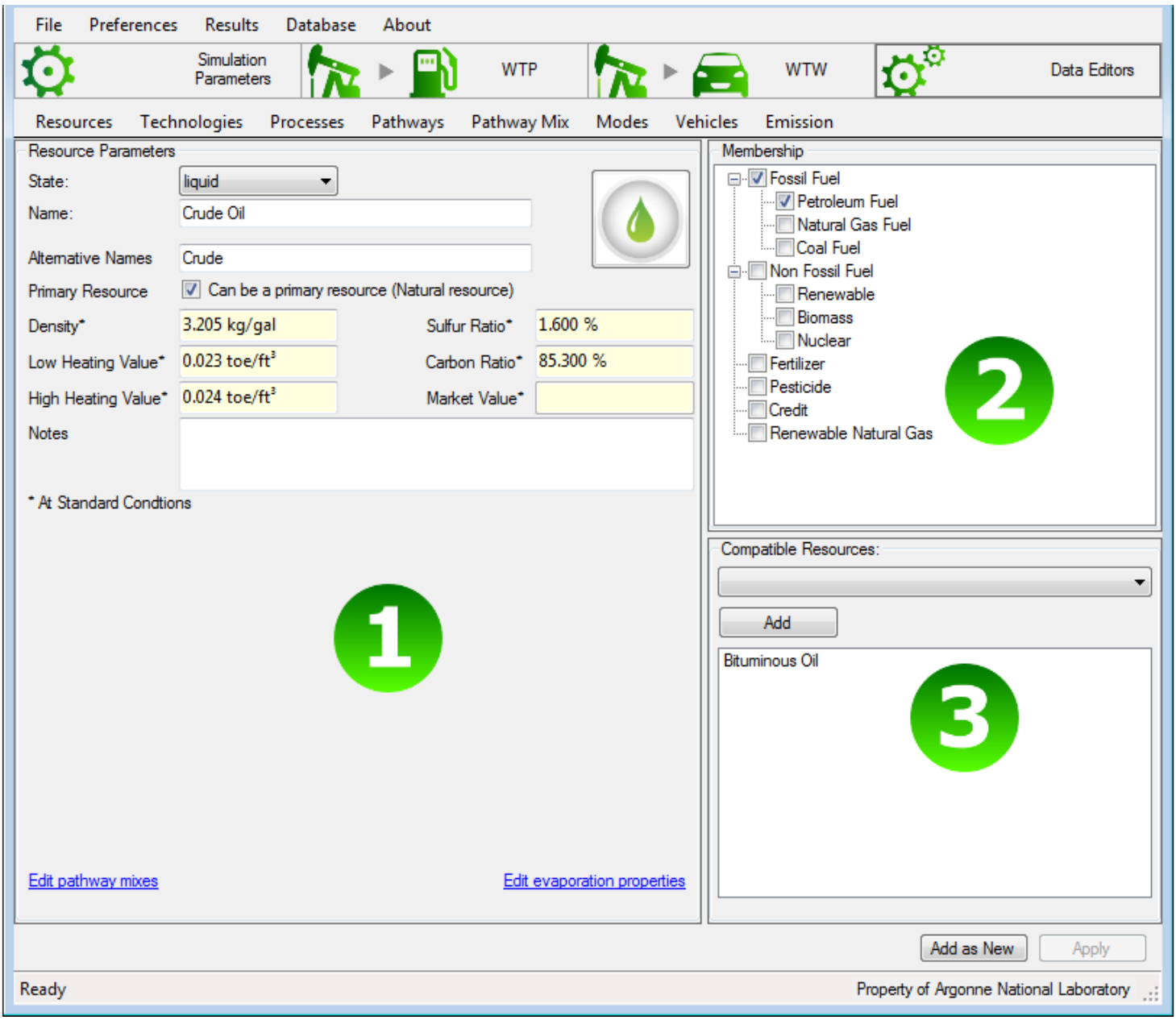


Figure 26: Crude Oil in the Resource Editor

Figure 26 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].

The values for density and heating values can be either removed (Right-click then "Delete value") or set to zero. In this case, the software will not automatically perform unit conversions between mass/volume/energy.

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 form 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.

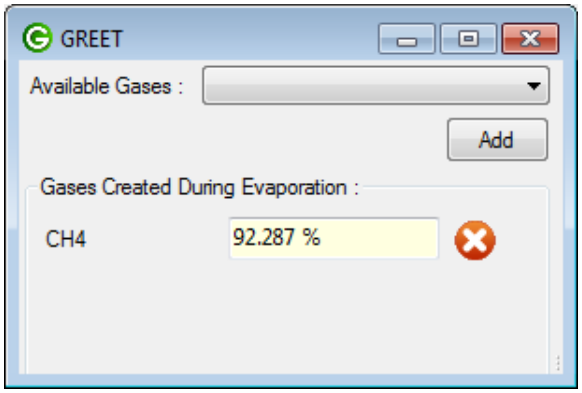


Figure 27: Natural Gas Evaporation Gases Parameters

Figure 27 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 26). 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 28 or in the Well-to-Wheels main pane.

Energy	
Life Cycle	
Resources	**7487.678 btu**
Groups	
Fossil Fuel	**7484.494 btu**
Natural Gas Fuel	**7442.617 btu**
Petroleum Fuel	**28.032 btu**
Coal Fuel	**13.846 btu**
Non Fossil Fuel	**3.184 btu**
Nuclear	**2.082 btu**
Renewable	**1.102 btu**
Biomass	**0.115 btu**

Figure 28: 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 29.

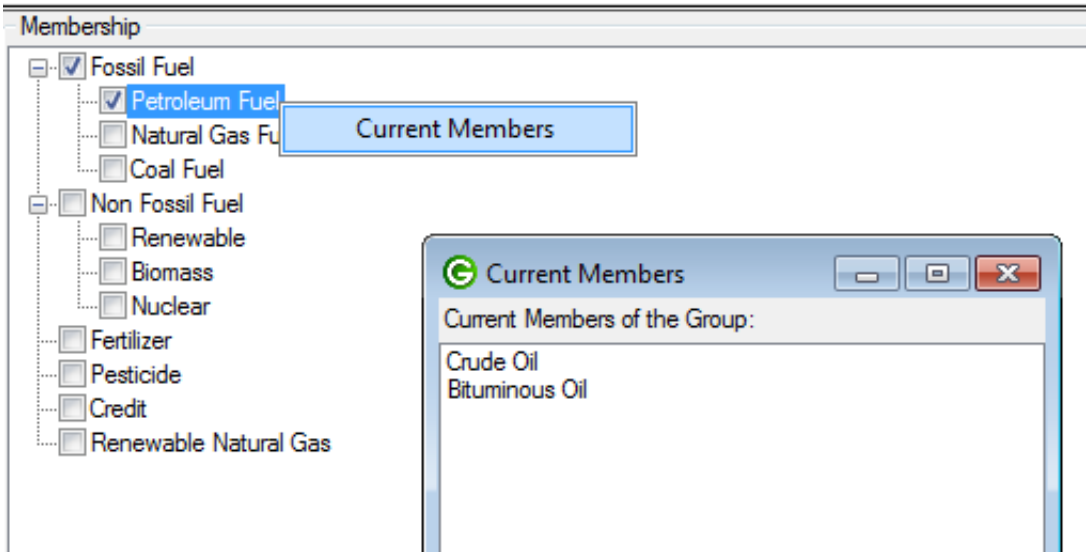


Figure 29: Display Members of a Resource Group

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**.

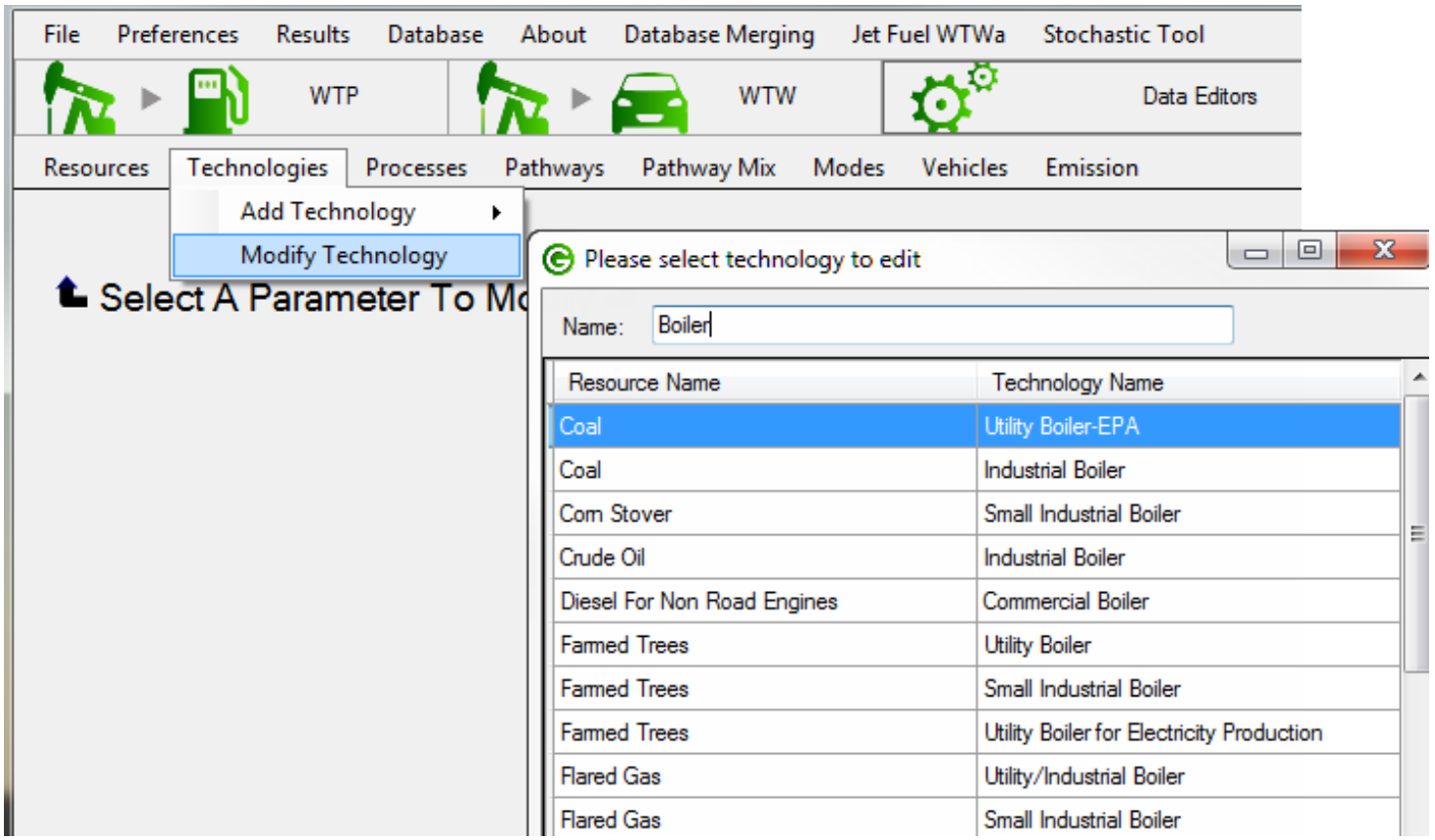


Figure 30: Select the Technology to Edit

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 30. Once the desired technology is found, double-click on it to see the emission factors.

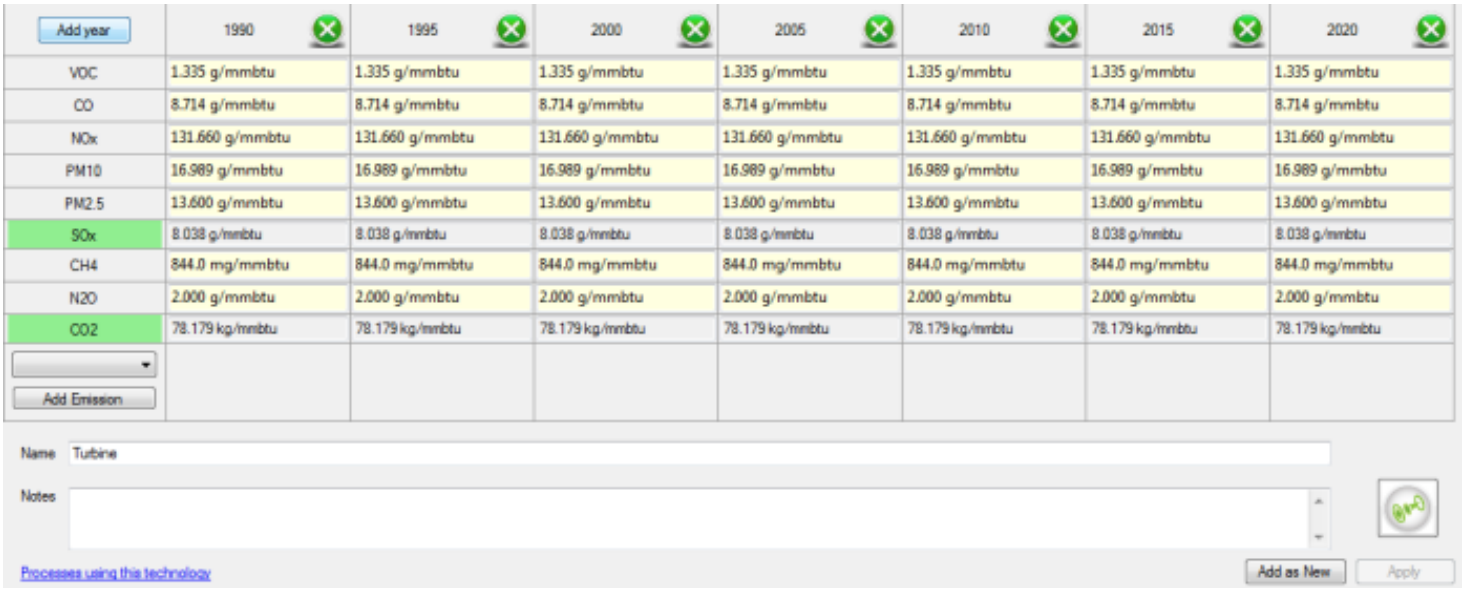


Figure 31: Technology Editor Showing Emission Factors and Properties

The screenshot in Figure 31 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 32. 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 **Switch to calculated value**. This means this emission factor is now calculated using a balancing equation [1].

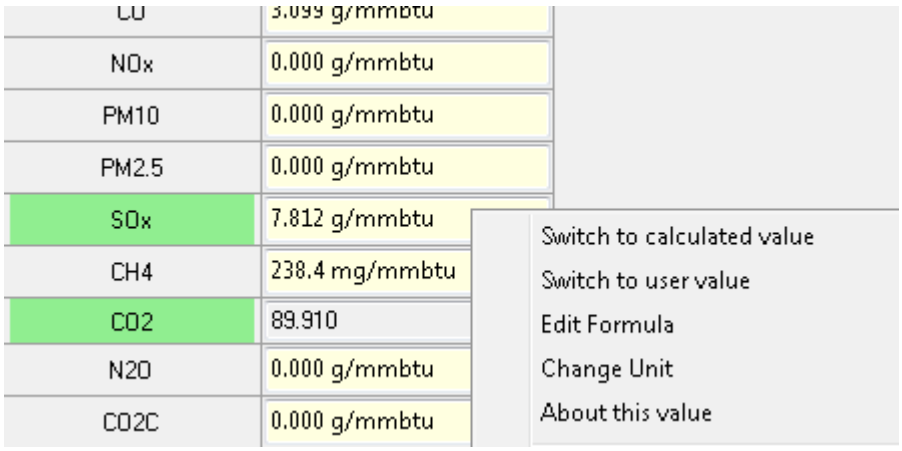


Figure 32: Green Background Indicates Emission Factors Are Balanced

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 33. 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.

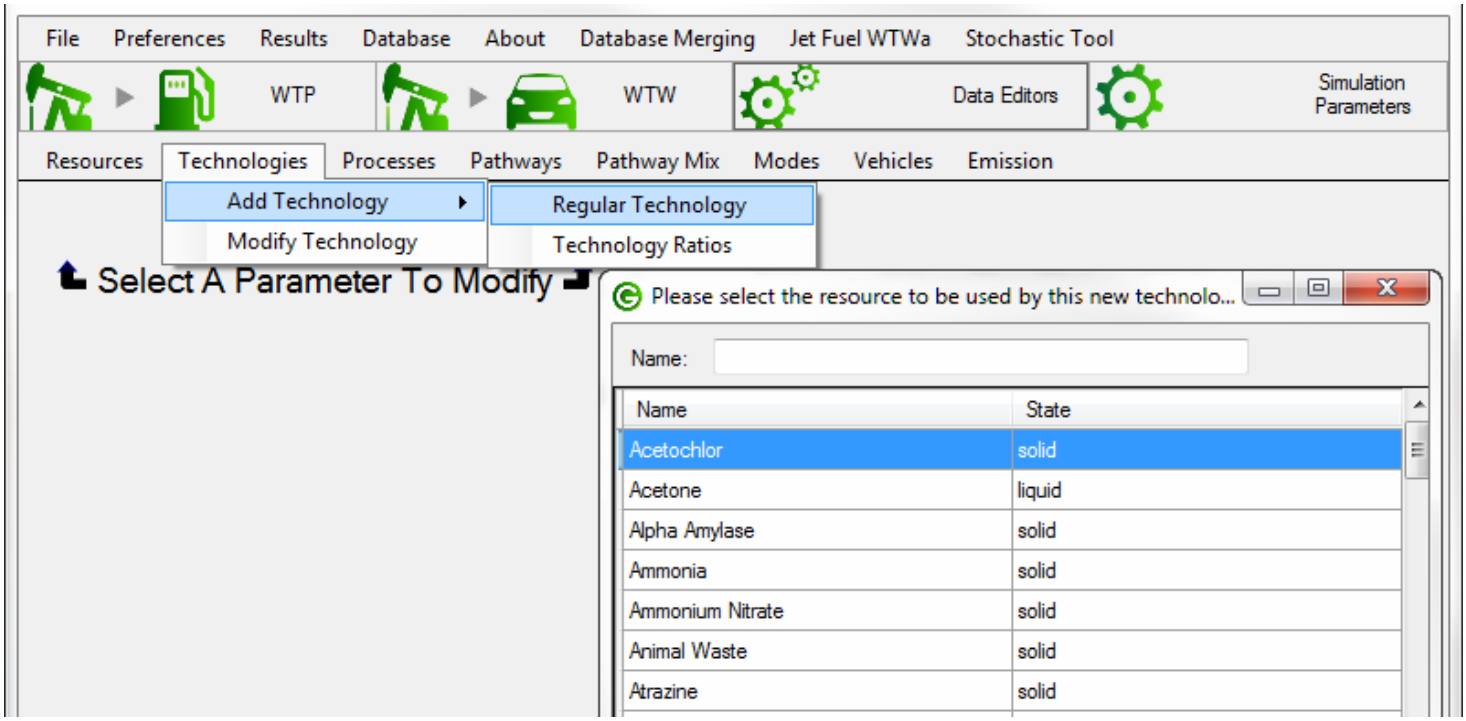


Figure 33: Selecting the Fuel for the New Technology

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 34. 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.

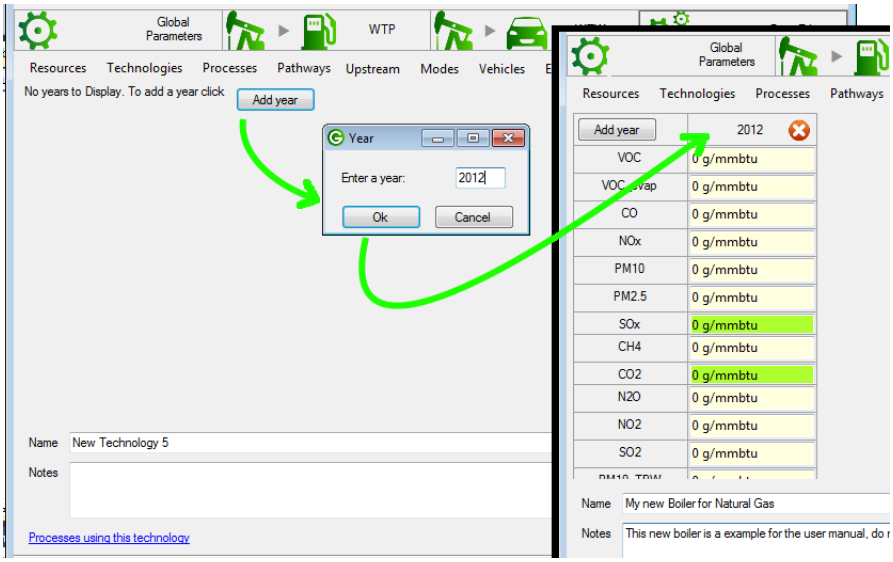


Figure 34: Adding Emissions Factors for Specific Years in the Technology Editor

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

NOTE: Processes in GREET - Stationary Processes versus Non-Stationary Processes defined.

- **Stationary Process** - starts with defined resource GREET called an "Input" and ends by producing a defined resulting product GREET called an "Output". The Stationary Process also stores properties that are basically significant values of the process.
- **Transportation Process** - transportation of basically anything that includes all possible, determined to be significant, modes of transportation. The Transportation Process also stores properties that are basically significant values of the process.

The Process Editor allows you to create and modiify processes. It is accessed by clicking the **Data Editor** main pane button.

Creating a New Stationary Process: Start by clicking **Processes** menu item. Then click **Add Stationary Process**. Review Figure 35 for Stationary Processes interface.

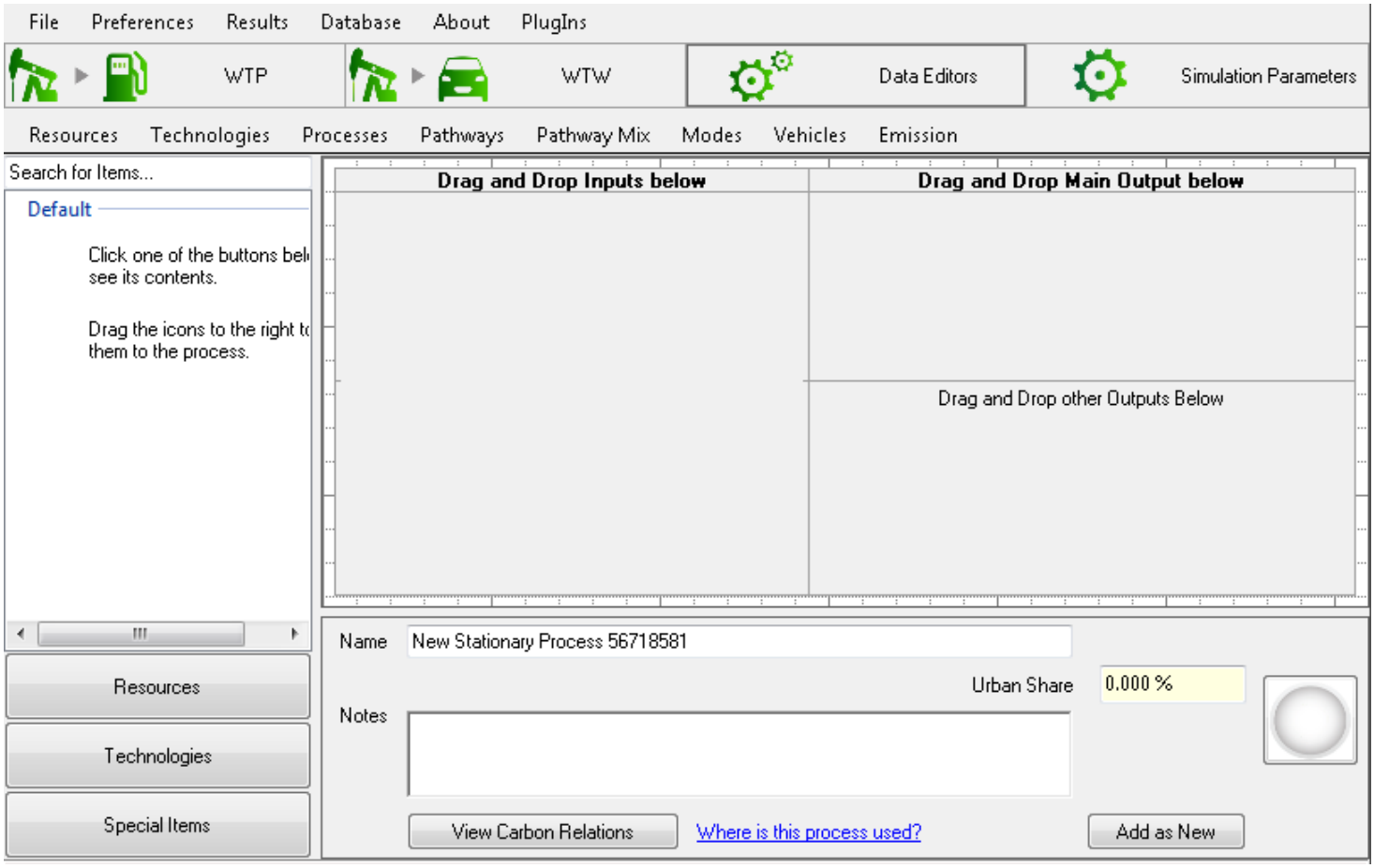


Figure 35: Stationary Process Interface in the Editor

Defining the main output To add a main input or output to a process 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 input or output in the process layout.Review Figure 36 for **Output** drag and drop example, Figure 37 for **Input** Input drag and drop example.

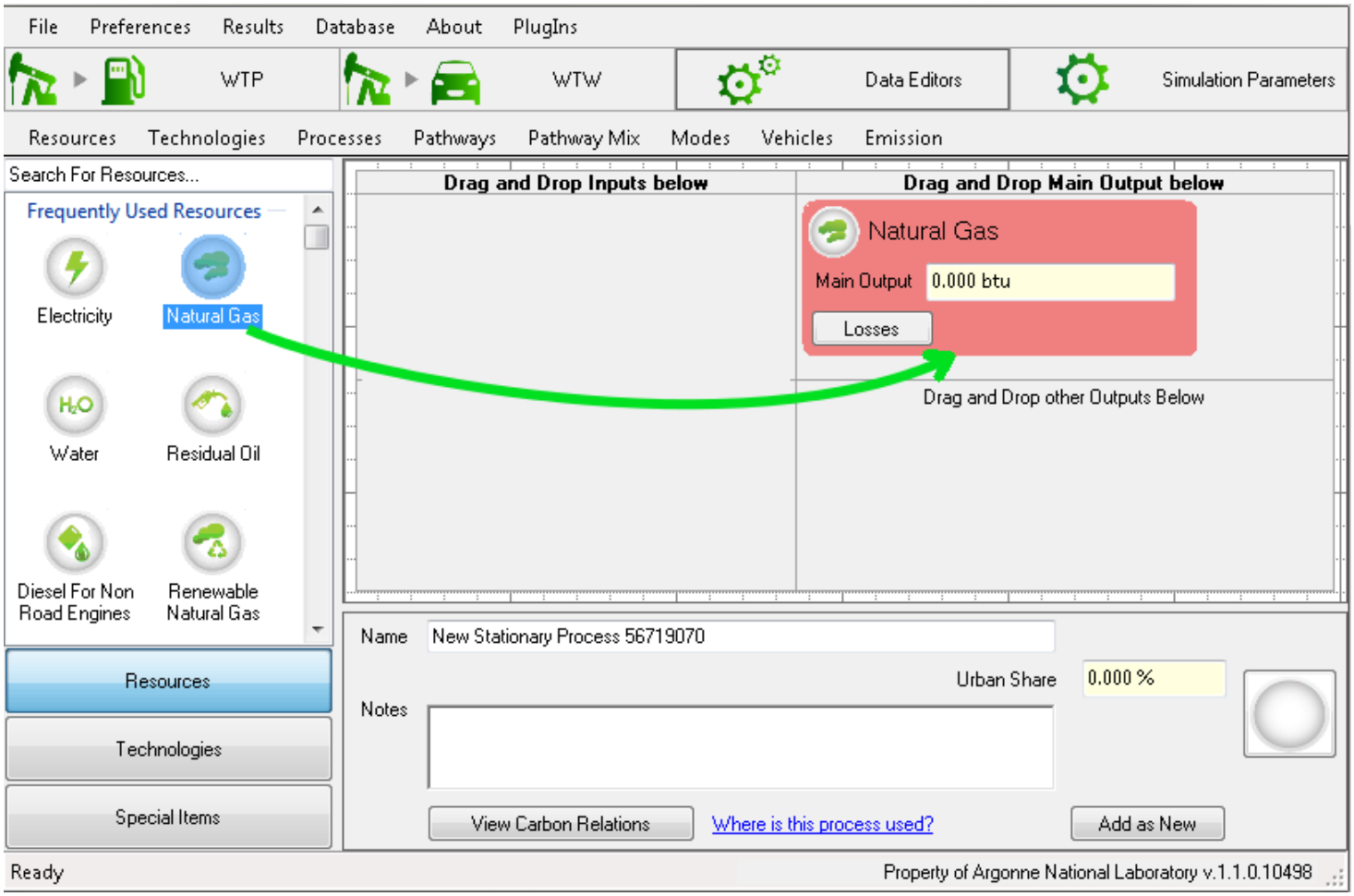


Figure 36: Drag and Drop a Resource for the Main Output Example

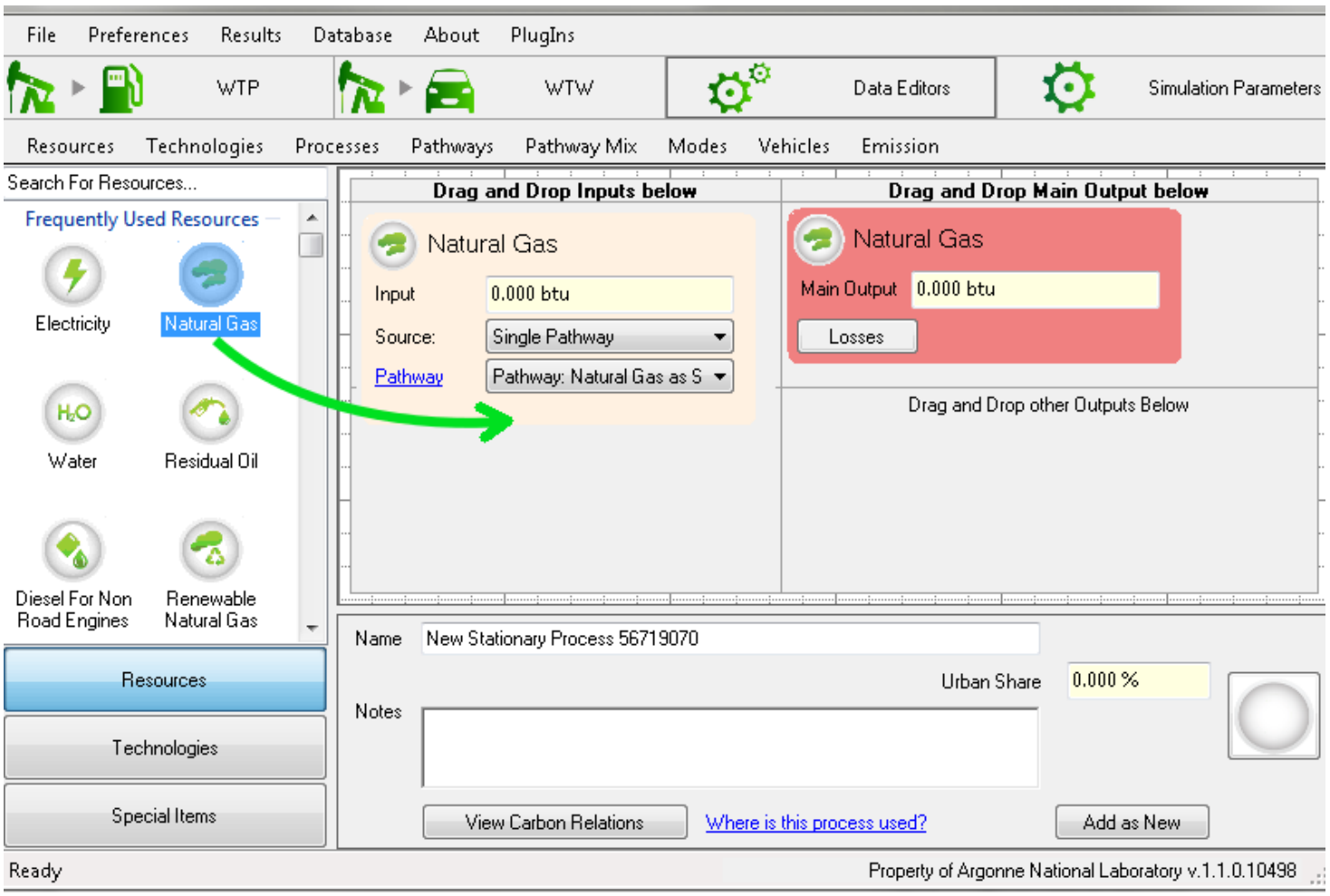


Figure 37: Drag and Drop a Resource for the Main Input Example

To delete input/output right clicking on input/output then select Delete I/O. see Figure 37.

The inputs defined represent the necessary resources to produce the specified amount of Output and the Co-products if a Co-product exists.

- The best practice is to define an amount. Click on the **Resources** category button on the bottom left of the interface. Then drag and drop the resource into the process; into inputs. Then, define the amount by entering a value in the Input box.

  Note that the **Source** and **Pathway** or **Mix** have default values. The source repersents the 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 (for that input).

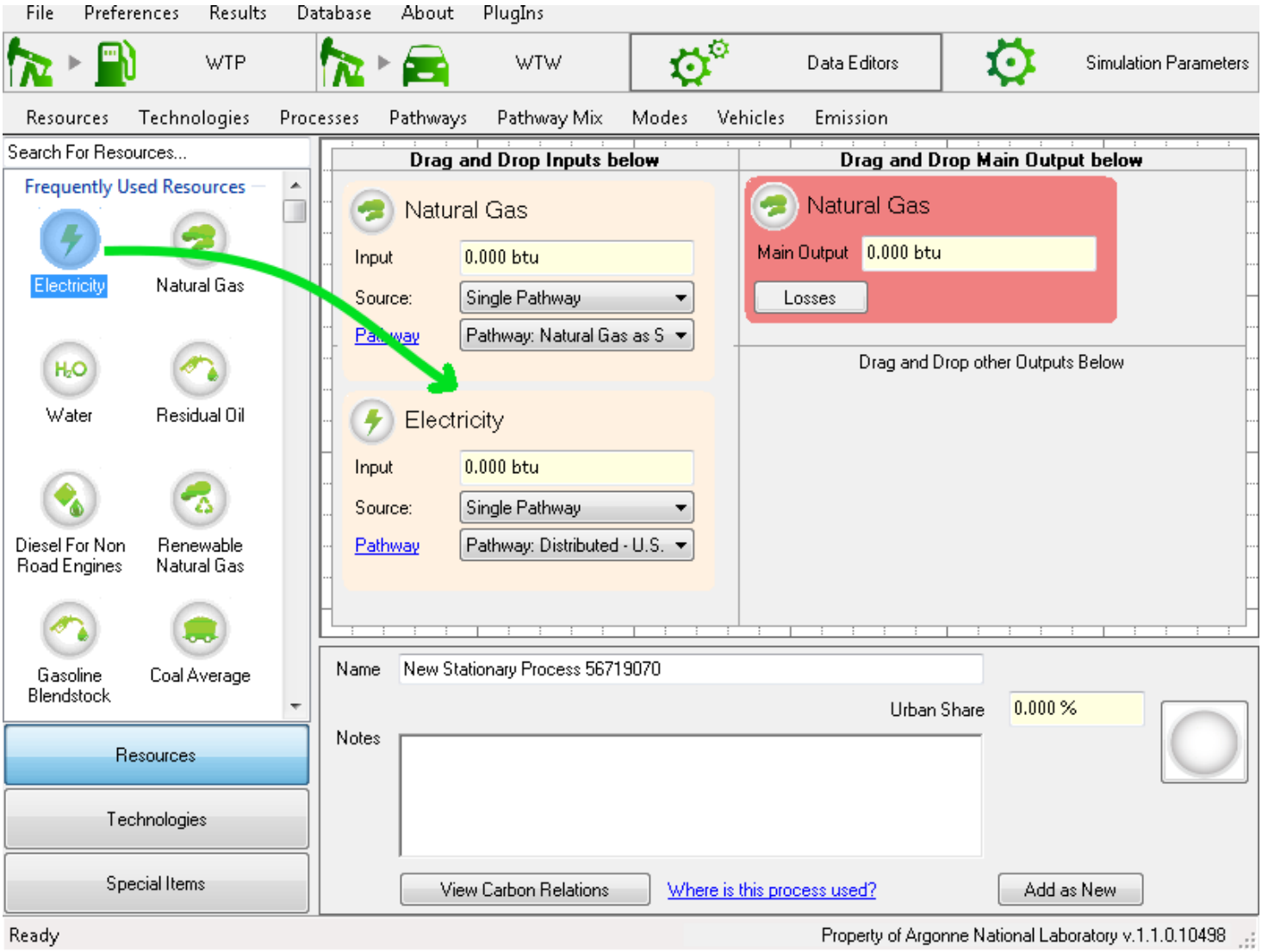


Figure 38: Drag and Drop a Resource as an Input for the Process

- The second way is to create a group and place one or more inputs into that group and assigning them a share, Review Figure 39.

  NOTE: Figures 39 shows what is needed to add inputs to a group.

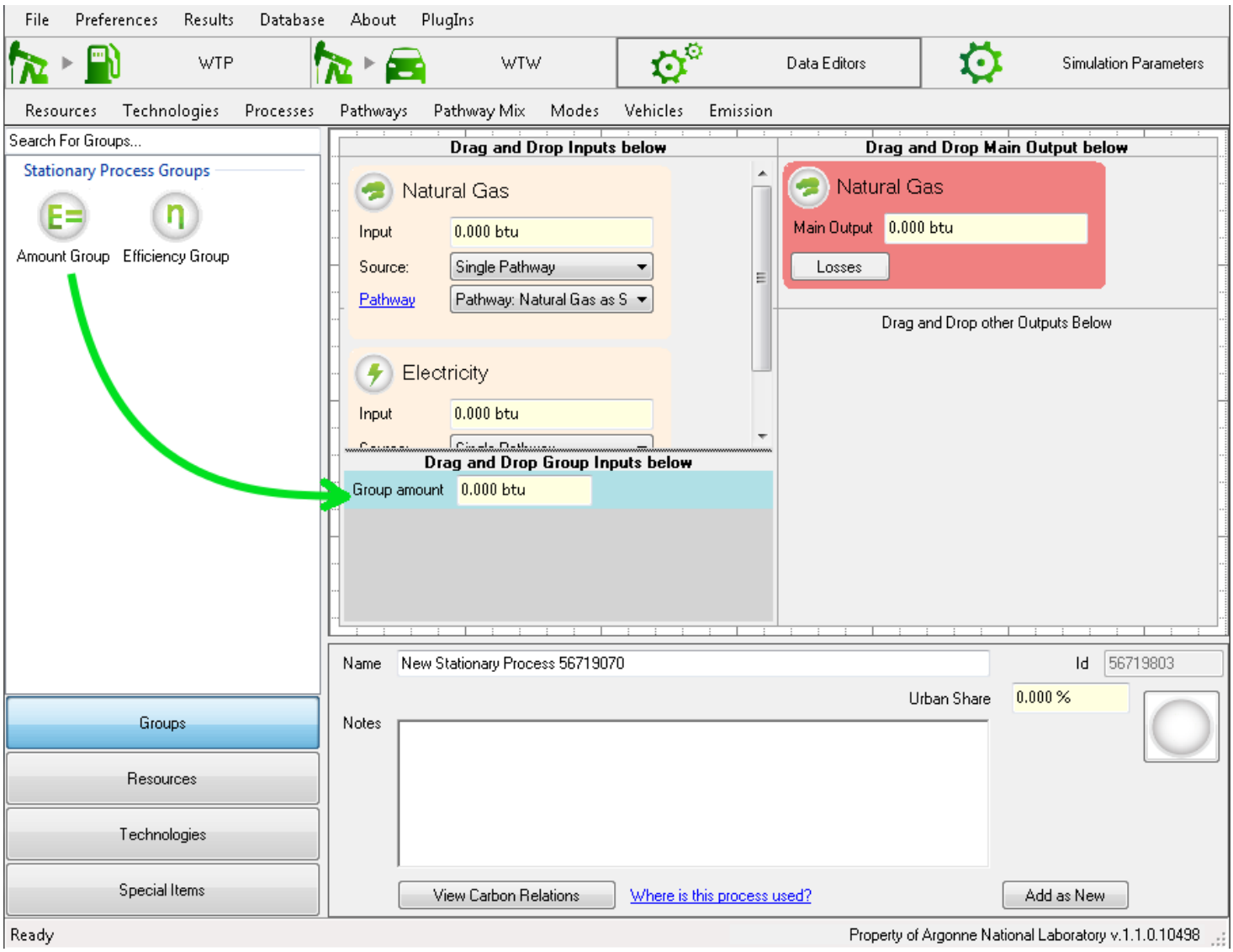


Figure 39: Drag and Drop an Amount Group

the three processes shown below in Figure 40 are **identical**. All three processes will be converted to the same canonical form before calculating the results. For more details see [1].

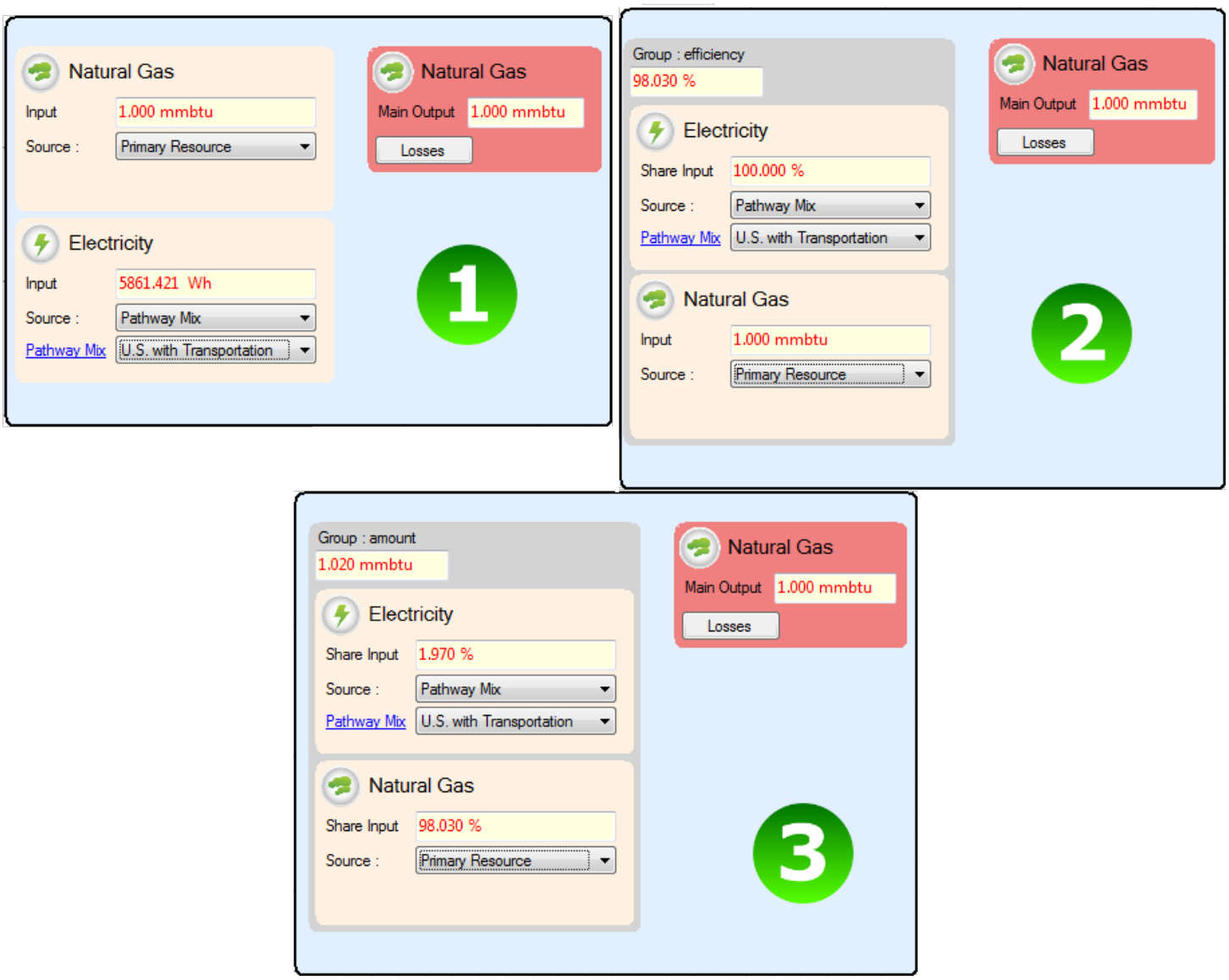


Figure 40: Three Equivalent Stationary Processes

Adding Technologies: Technologies can only be added to stationary processes; Technologies are not needed with transportation processes and thus not allowed to be added. Technologies are added to stationary processes in the same drag and drop way Resources are added to processes. Figure 41 shows dragging and dropping a utility boiler and a simple cycle gas turbine onto a natural gas input.

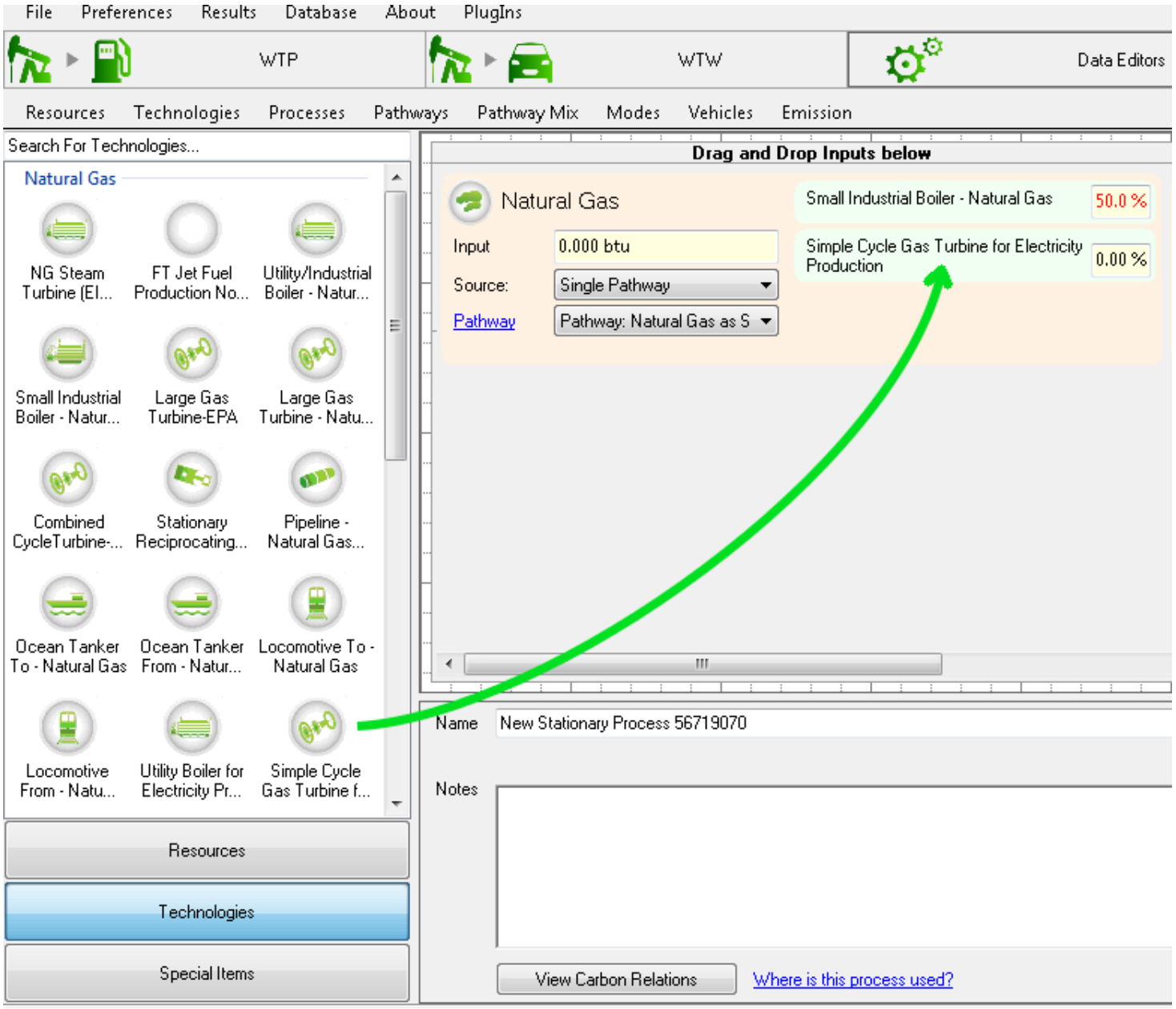


Figure 41: Adding a Technology Over an Input and Assigning Shares to It

When you right click on a **Technology** the menu shown in Figure: 42 is displayed. Actions of each menu item are described next.

- **Delete Technology** deletes the Technology you are currently on.
- **About this Technology** menu 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.

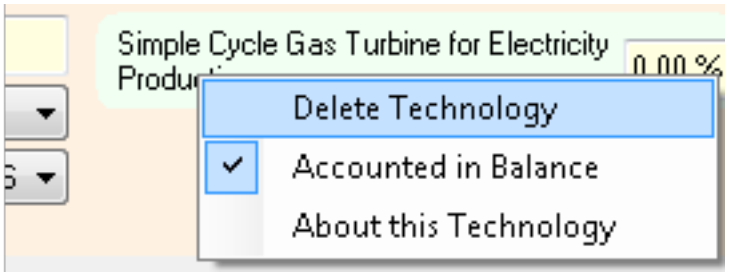


Figure 42: Technology Right Click Menu

Co-Products: You can add CoProduct directly by draging and droping a Resource from the Resource display area into "Drag and Drop other Outputs Below" area. Or by, converting an input to a CoProduct by right clicking on the input then selecting **Set as CoProduct**. See Figure 43.

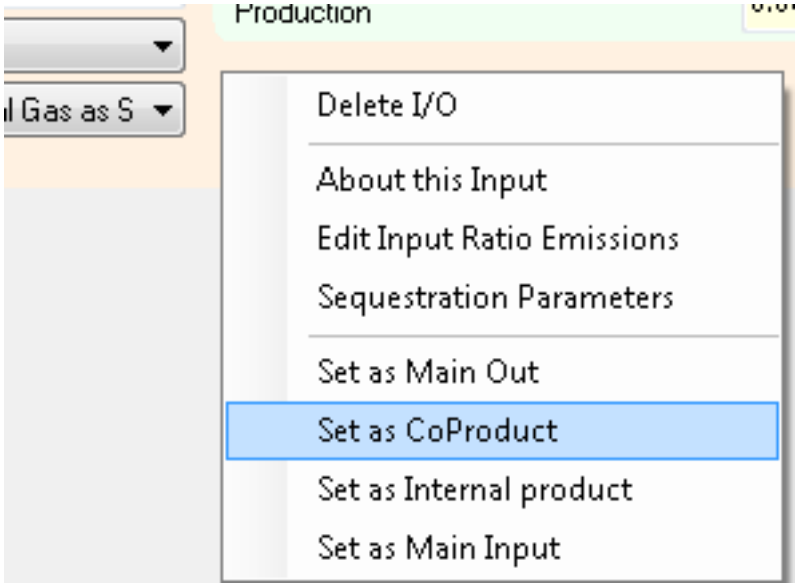


Figure 43: Input Right Click Menu

Parameters for coproducts are accessed by right clicking one of the coproducts; The pop-up window appears as shown in Figure 44.

The treatment method can be Displacement or Allocation. Both are calculated at the process level. Pathway level treatment methods aren't available as of yet with this current version.

If the allocation method is selected as shown on Figure 44 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 below, 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 need to be defined. In Figure 45, 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. A displaced resource can be added by clicking the **Add displacement resource** button and an existing one can be removed by clicking .

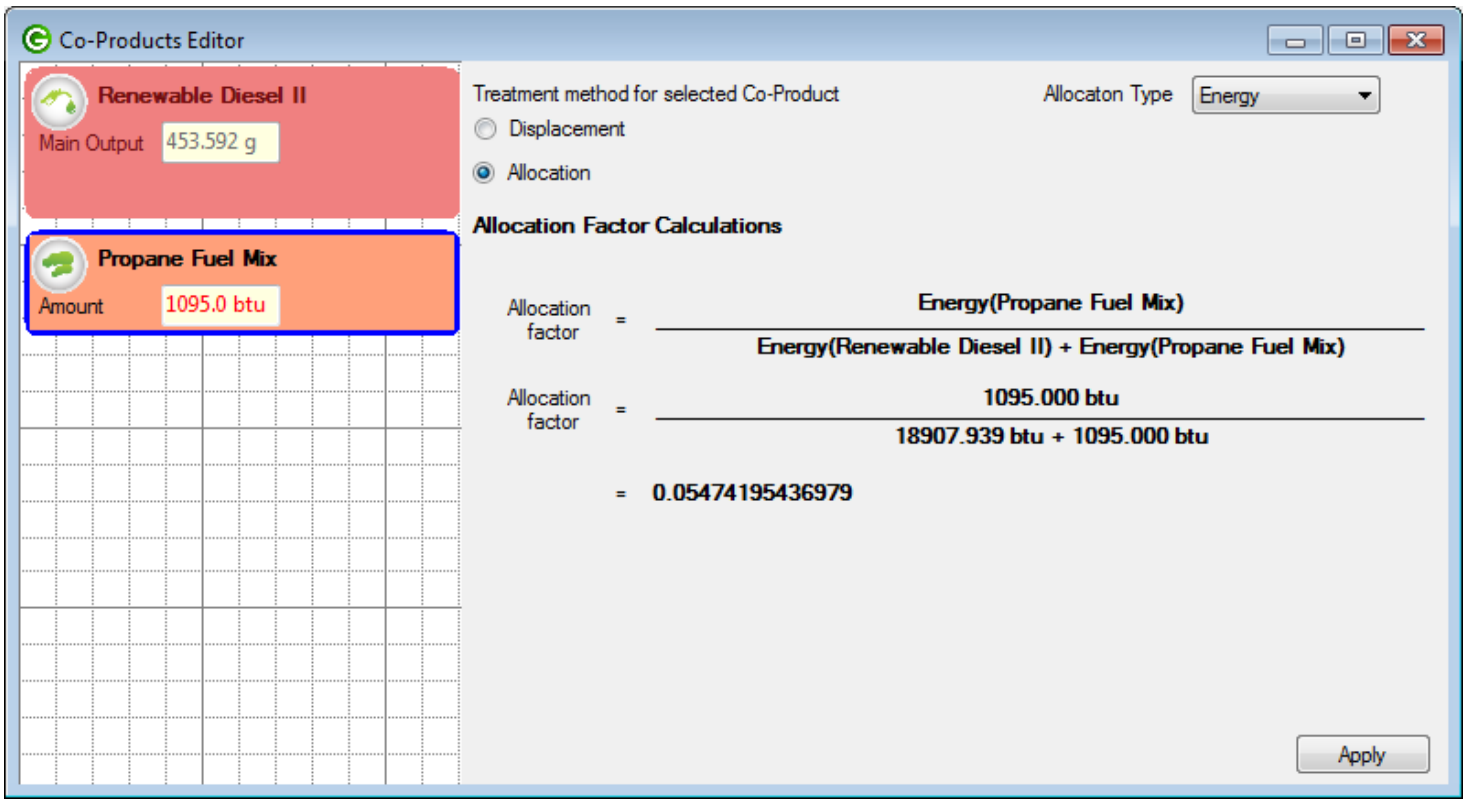


Figure 44: Coproducts Editor

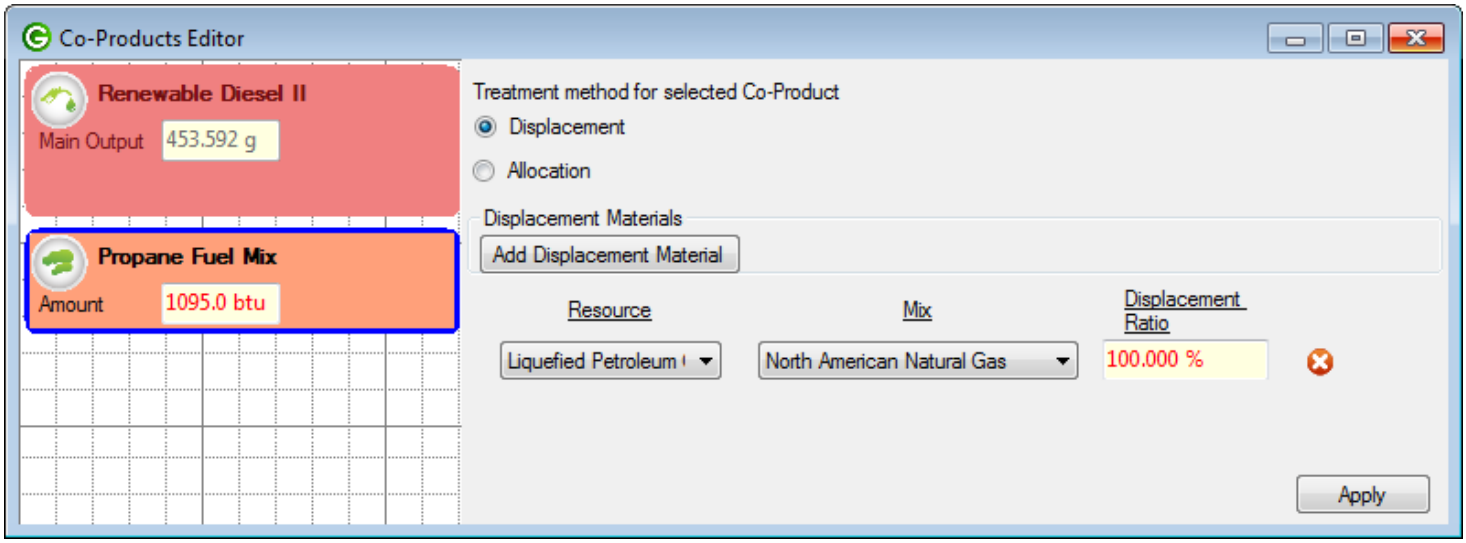


Figure 45: Coproducts Editor with Propane Displacing LPG

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.

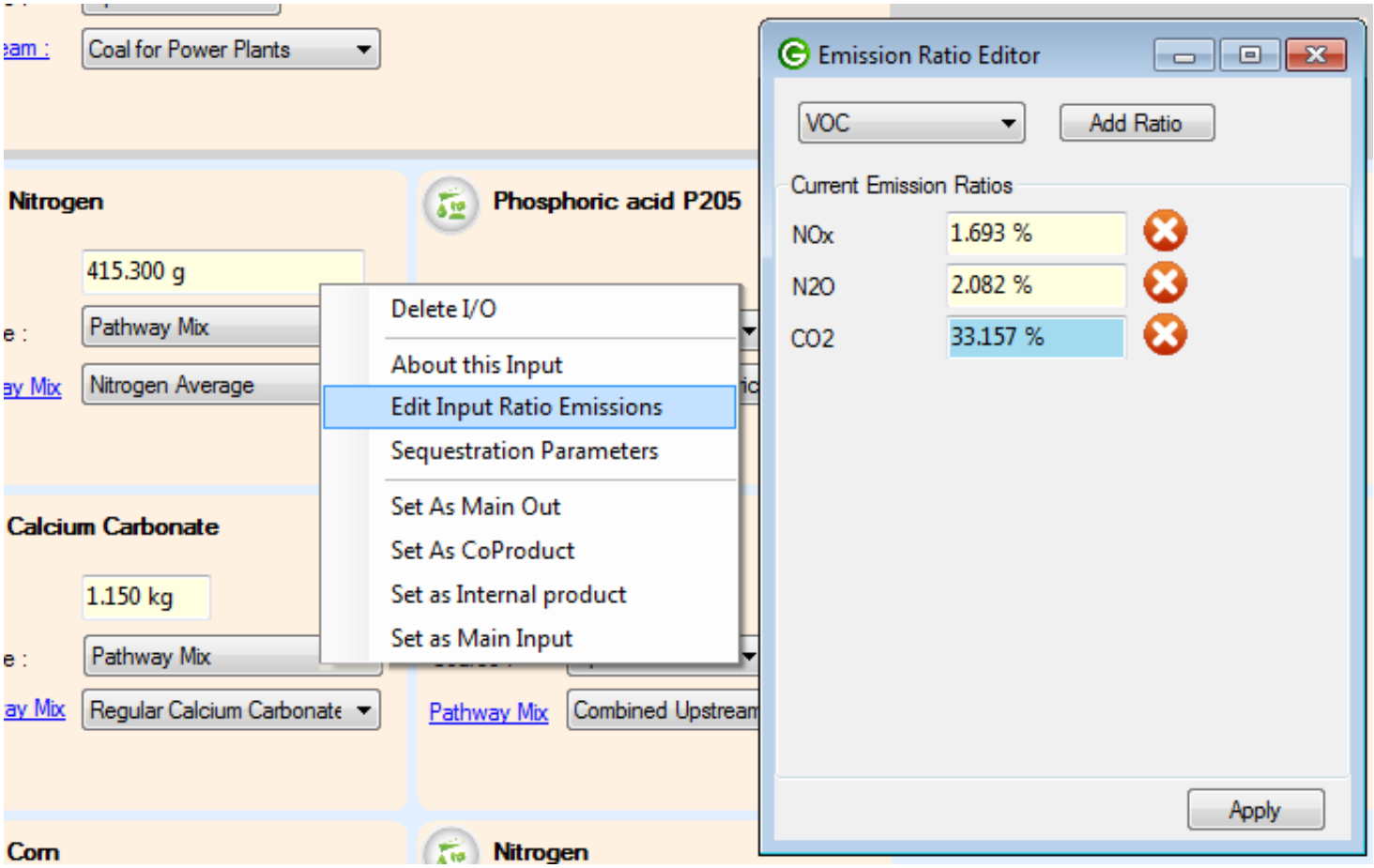


Figure 46: Input Emission Ratios

In Figure 47, 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 3.6.2).

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.

View Carbon 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.

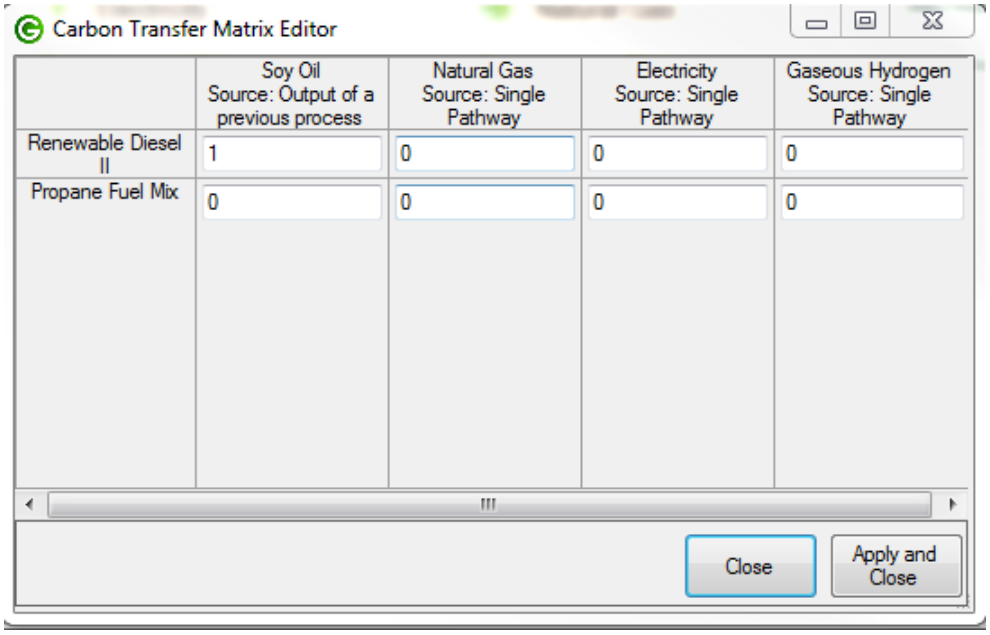


Figure 47: View Carbon Relations

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.

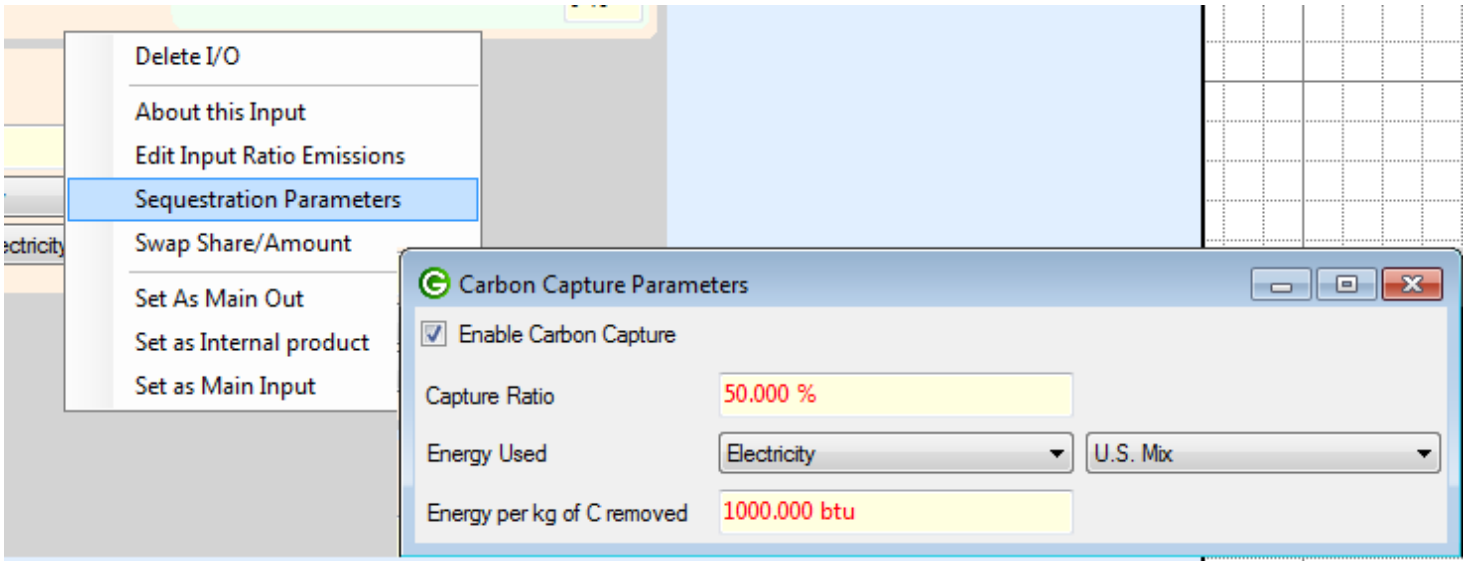


Figure 48: Carbon Sequestration Parameters Accessed from the Stationary Process Editor

The carbon sequestration parameter will be shown as in Figure 48. 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**.

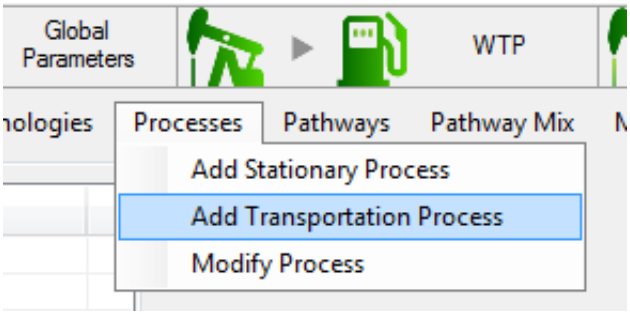


Figure 49: Adding a new transportation process to the database

Creating a New Transportation Process Start by clicking **Processes** menu item. Then click **Add Transportation Process**. Review Figure 50 for Transportation Processes interface.

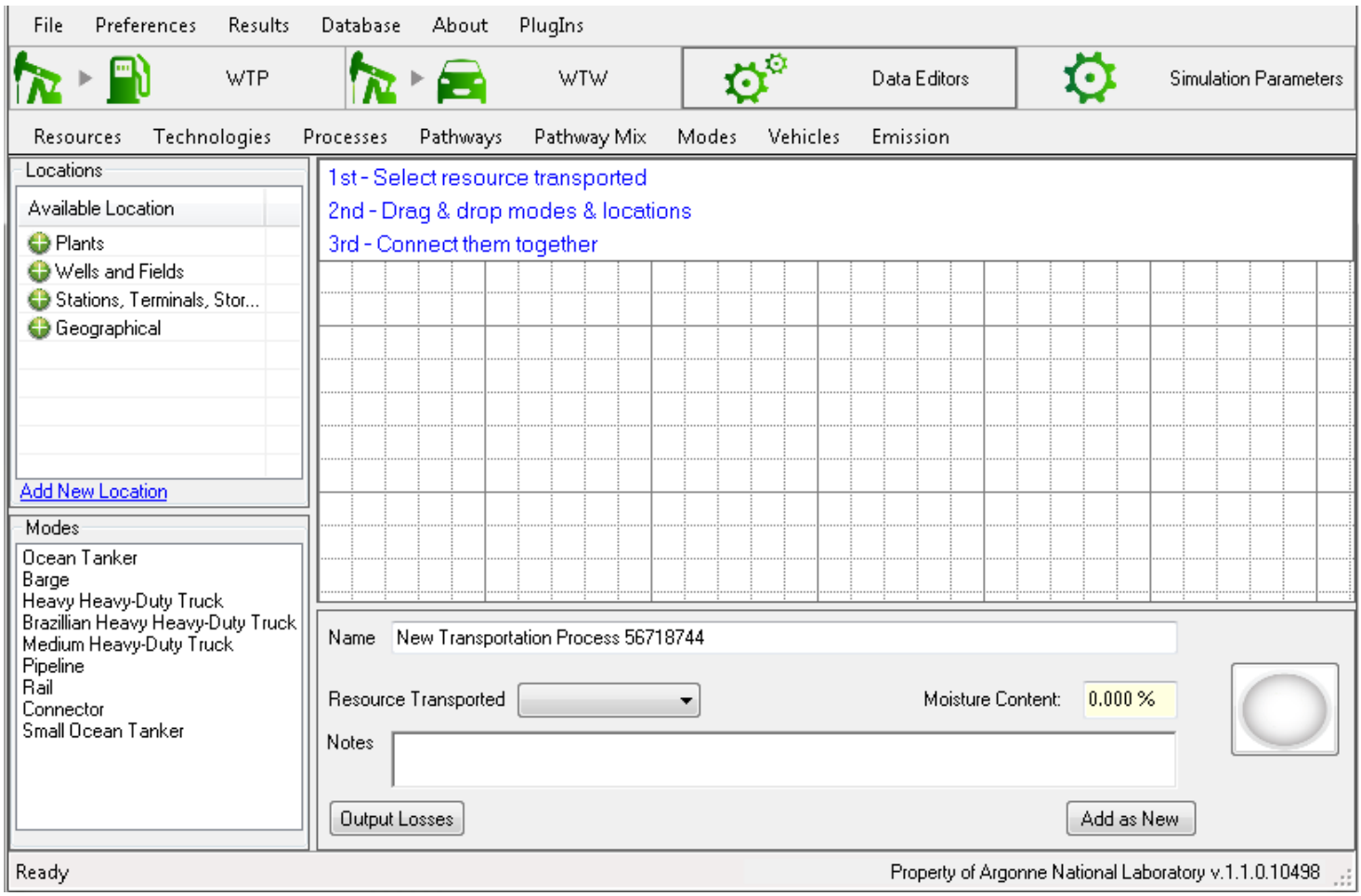


Figure 50: Transportation Process Interface in the Editor

Then, building a transportation process is based on dragging and dropping the transportation steps you need and linking them together. See Figures 51. 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).

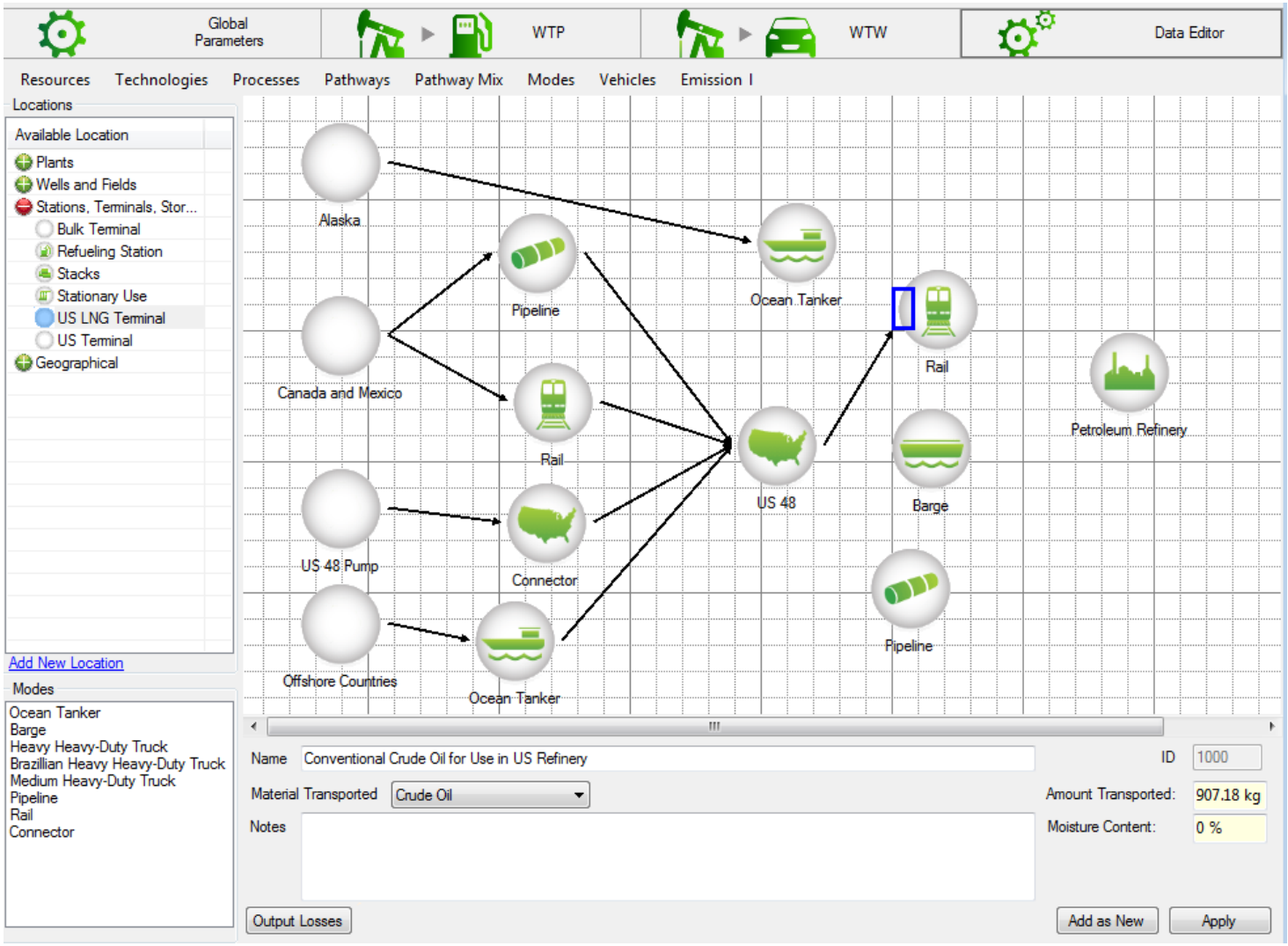


Figure 51: Linking Transportation Steps and Locations in the Transportation Editor

Select the material transported: When building a new transportation process, the first thing to do is to select the material transported. See Figure 52. 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.

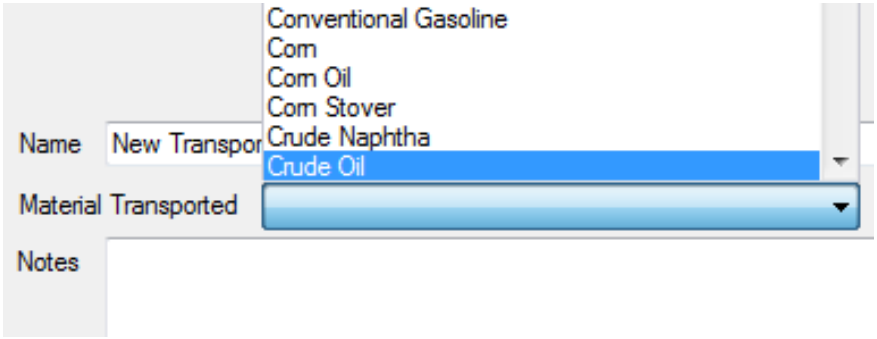


Figure 52: Selecting the Material Transported

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 53 and 54.

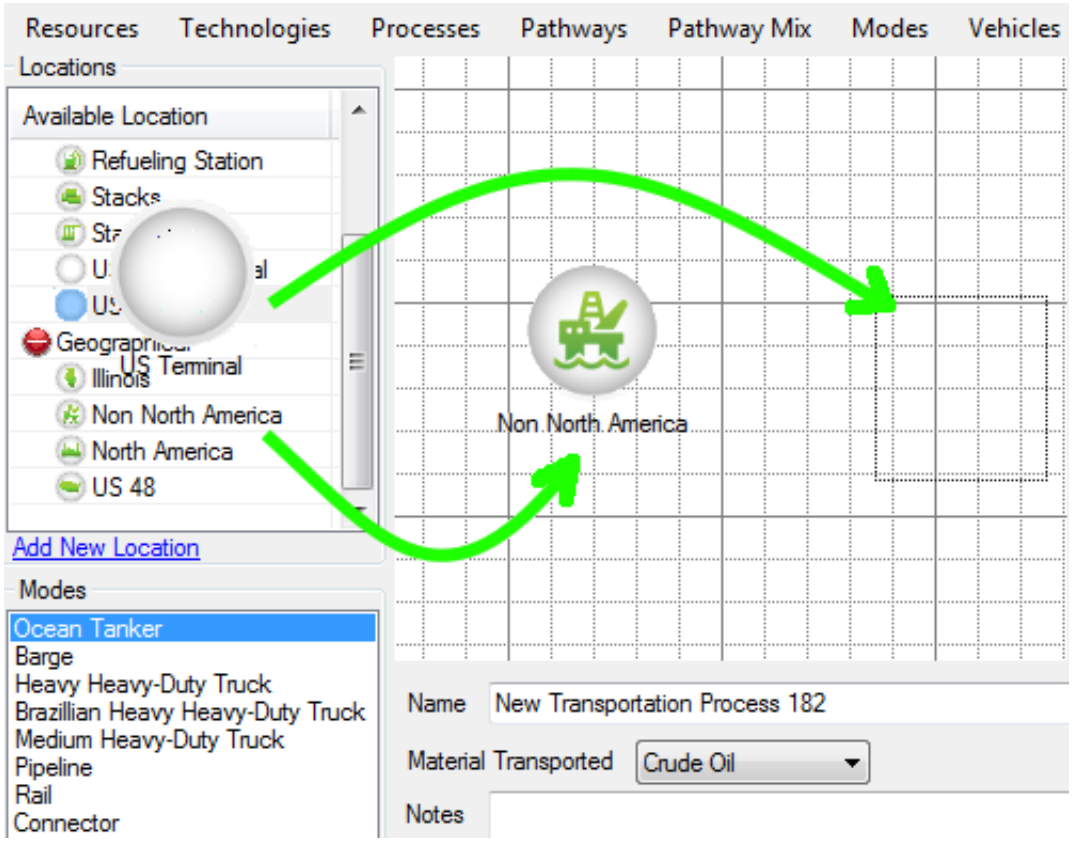


Figure 53: Dropping Locations for the Transportation Process

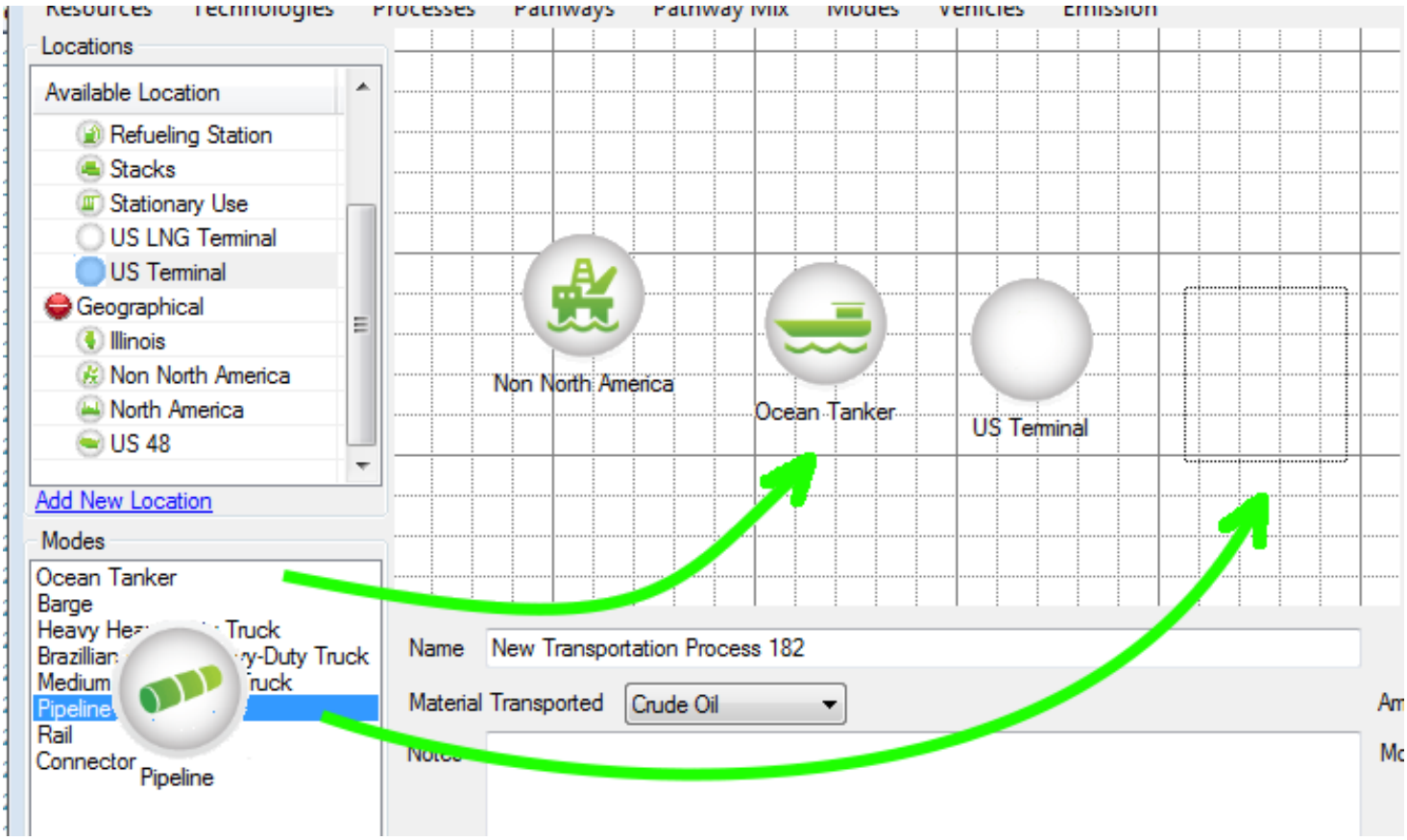


Figure 54: Dropping Modes in the Transportation Process

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 51. 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 Step / Delete location** 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 55. 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.

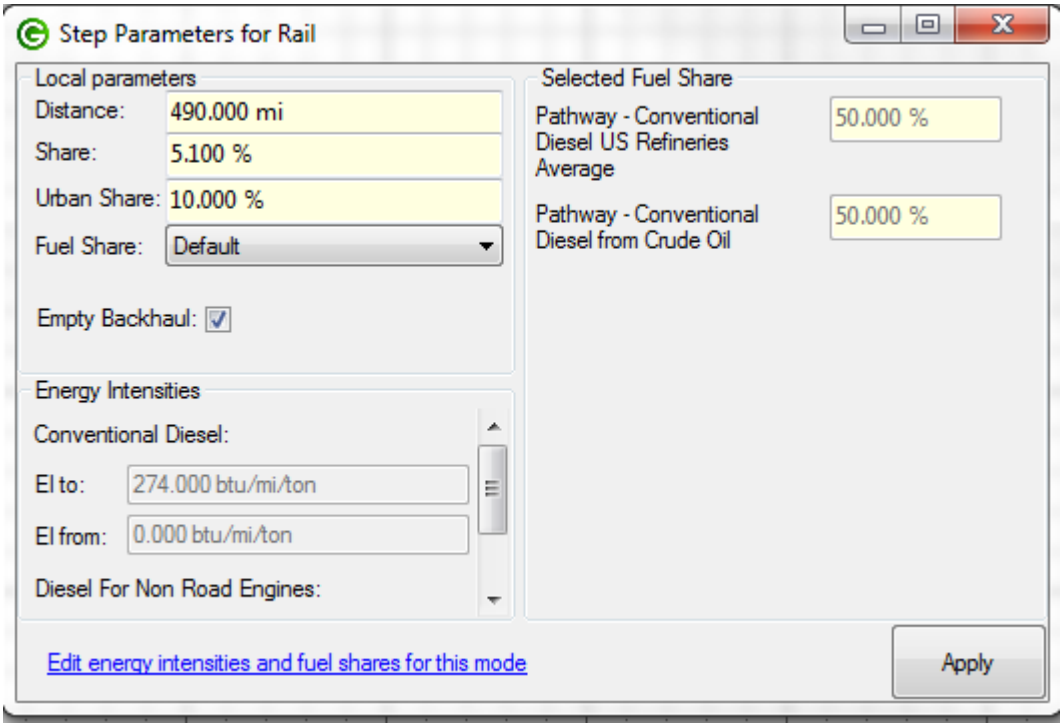


Figure 55: Mode Parameters and Mode Fuel Shares

Multiple sets of fuel shares are predefined and can be selected using the drop-down box. In the example in Figure 55, 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**.

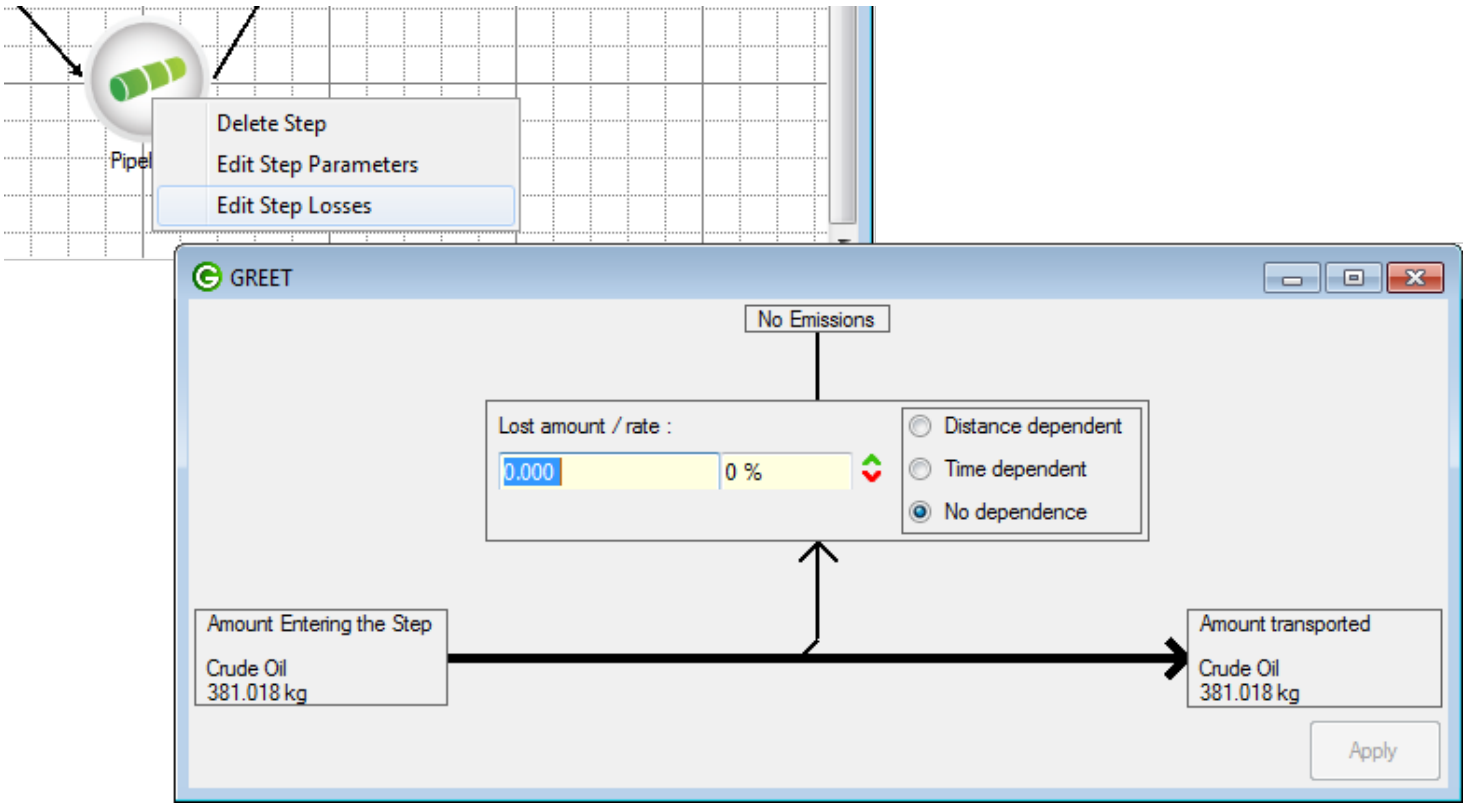


Figure 56: Transportation Step Losses

A new form will pop up, see Figure 56. 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 Editors** 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 57.

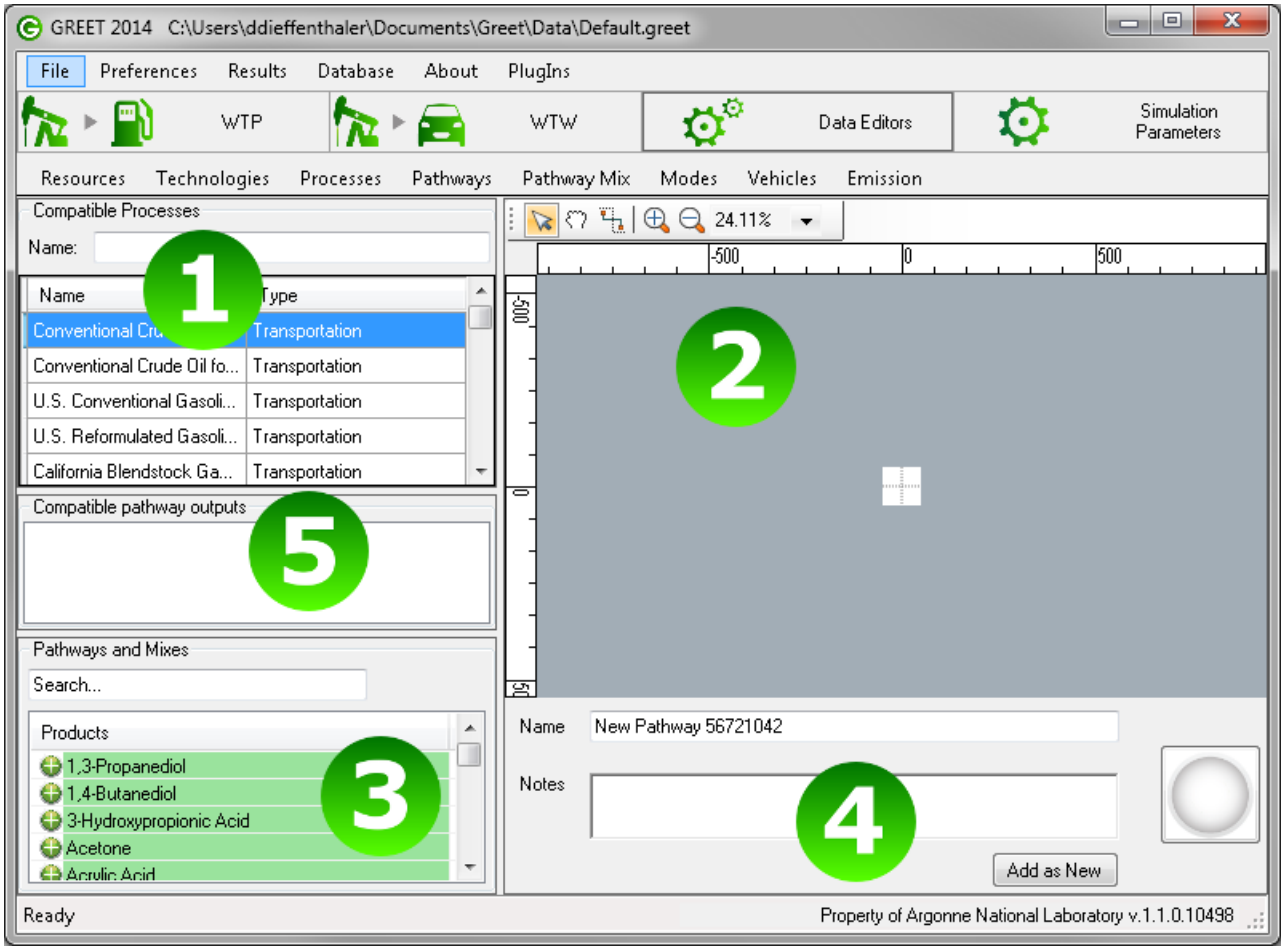


Figure 57: Creating a New Pathway

The pathway editor is designed in 5 main zones. See Figure 57

- 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 pathway such as name and notes.
- Zone 5 shows the resources available for outputs.

A process is selected then dragged and dropped to the building area.

Building a pathway consists in drag and dropping processes to the building area and connect them using the connector tool to define flows. Flows can only be built from an output to an input if they share the same resource. The amounts are normalized for each outputs during the calculations, so the outputs and inputs amounts do not need to match.

The next figure, figure 58, shows how to drag and drop the first process.

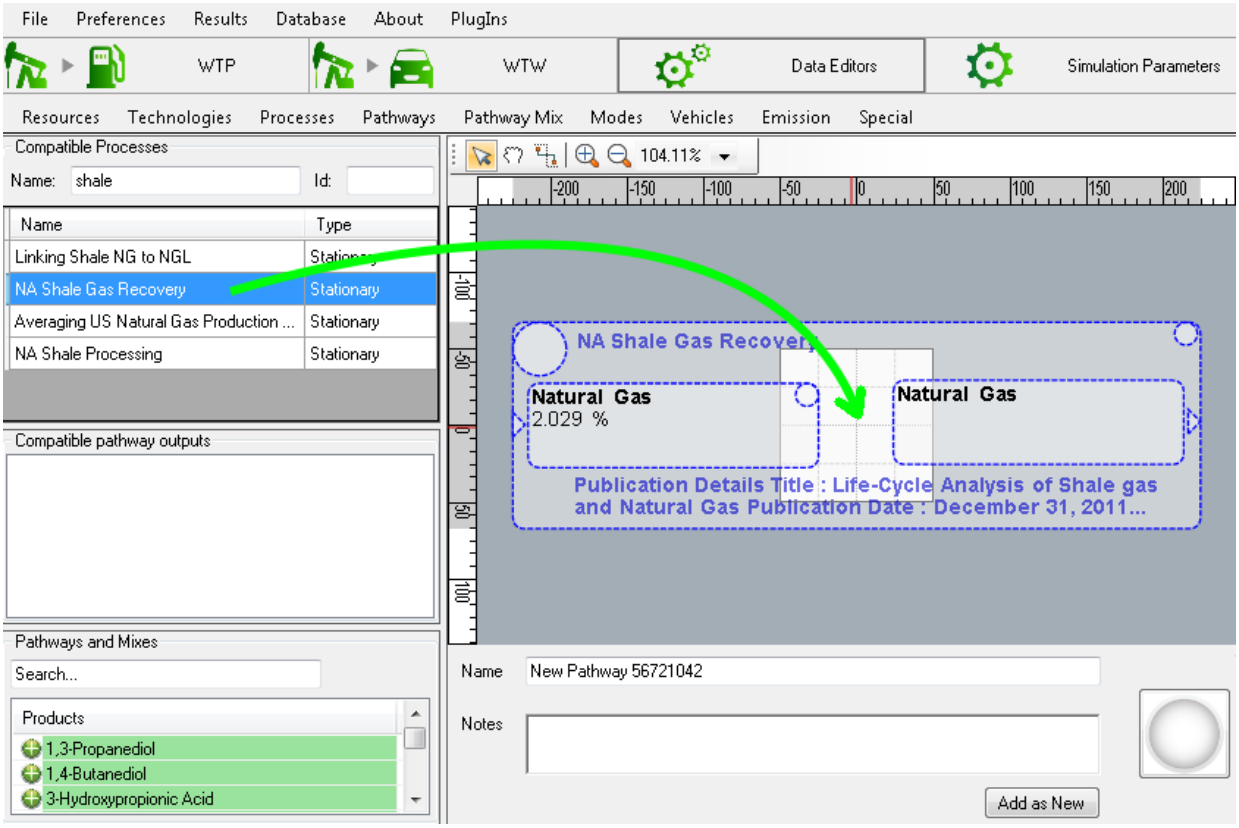


Figure 58: Dropping the First Process

As a result of dropping a process model, we can now see it in the building area. This process model has on input from **Output of Previous Process** and a single **Main Output** this is why we observe one input and one output that can be connected to other processes in figure 59

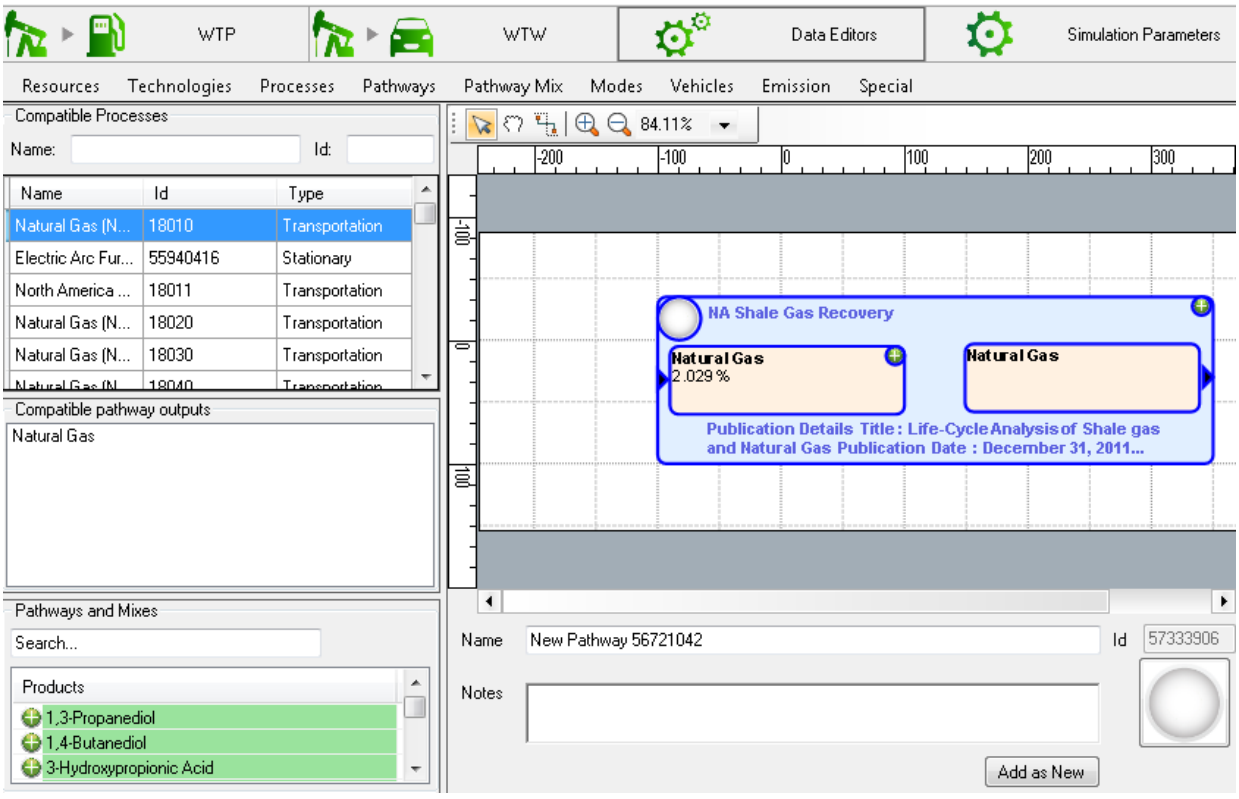


Figure 59: A stationary process has been added to this pathway

Many other processes can then be dragged and dropped to this pathway. In the next figure we dropped a natural gas processing process. Once both processes are there, we need to connect their inputs and outputs to define flows. In order to connect processes, we need to use the connector tool:

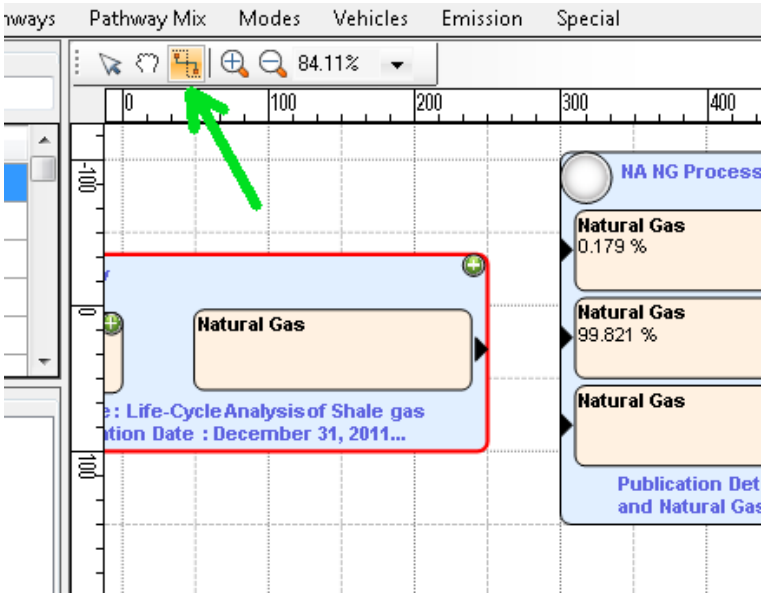


Figure 60: Selecting the connector tool to create a link between the feedstock and the process

Using the connector tool, one can draw flows between the output of the first process towards the inputs of the second one. By doing this we create a pathway by specifying that the upstream to use for these three inputs is the output of the first process.

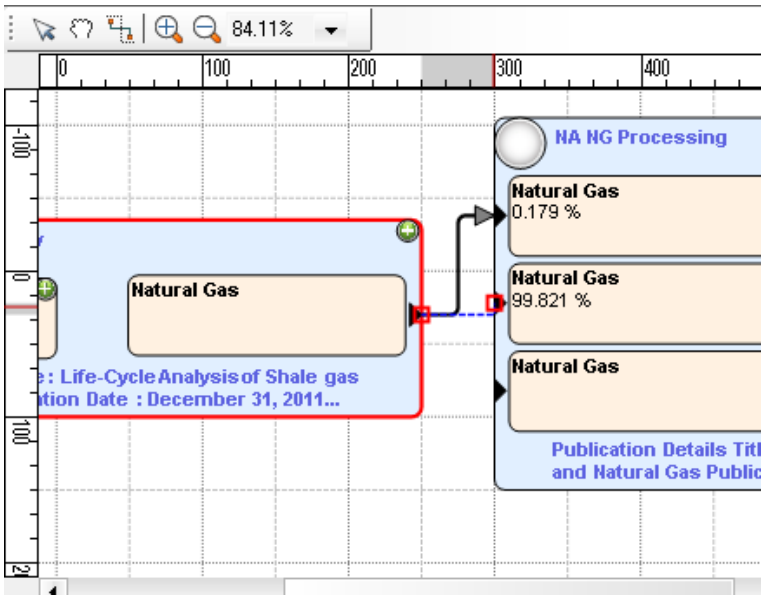


Figure 61: Connecting processes by drawing a line between the output of the first process to the input of the second one

When all processes are connected we need to define outputs for the pathway. A pathway need to define at least one main output. This main output is going to be used by default for all other items using that pathway as an upstream. Other outputs can be created, but they can only be re-used into other pathways.

In the folowing example we're going to demonstrate this by dropping two outputs for this pathway. In order to do that, drag and drop the "Natural Gas" output from the list on the left : textbfCompatible pathway outputs

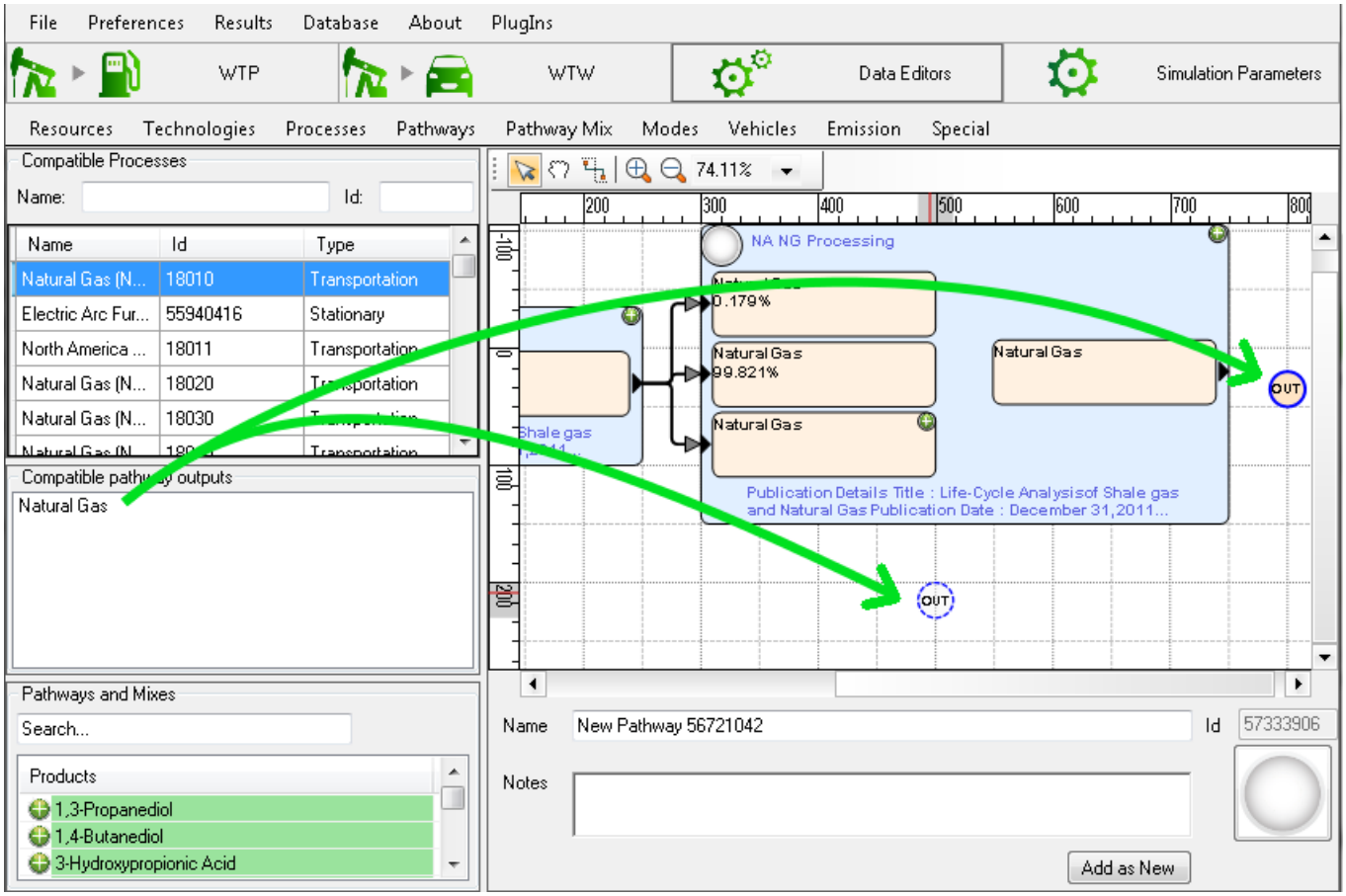


Figure 62: Dropping an output for the pathway, which will be the Main Output of this pathway

After dropping the outputs for the pathway, we need to link them using the connector tool.

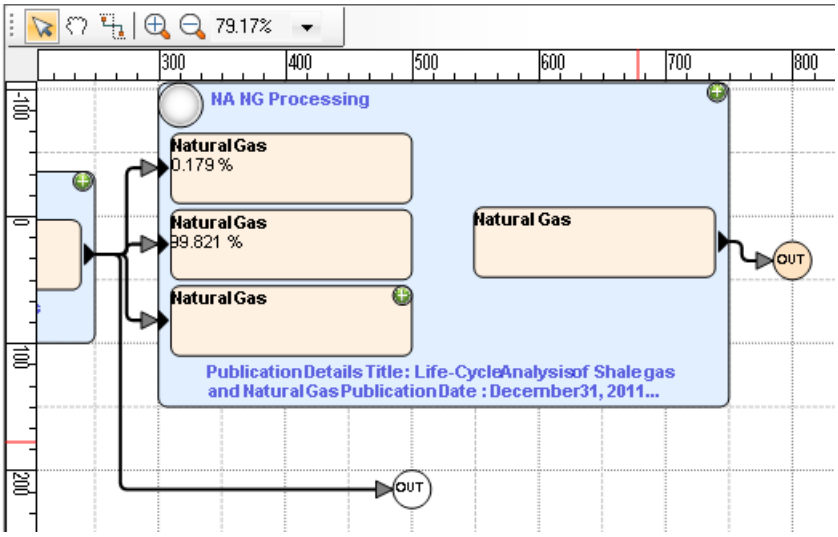


Figure 63: Both outputs of the pathways are connected to processes outputs

Now the only remaining connection to do is the feedback to the first process, in order to do that, again, use the connector too and connect the output of the last process to the first one.

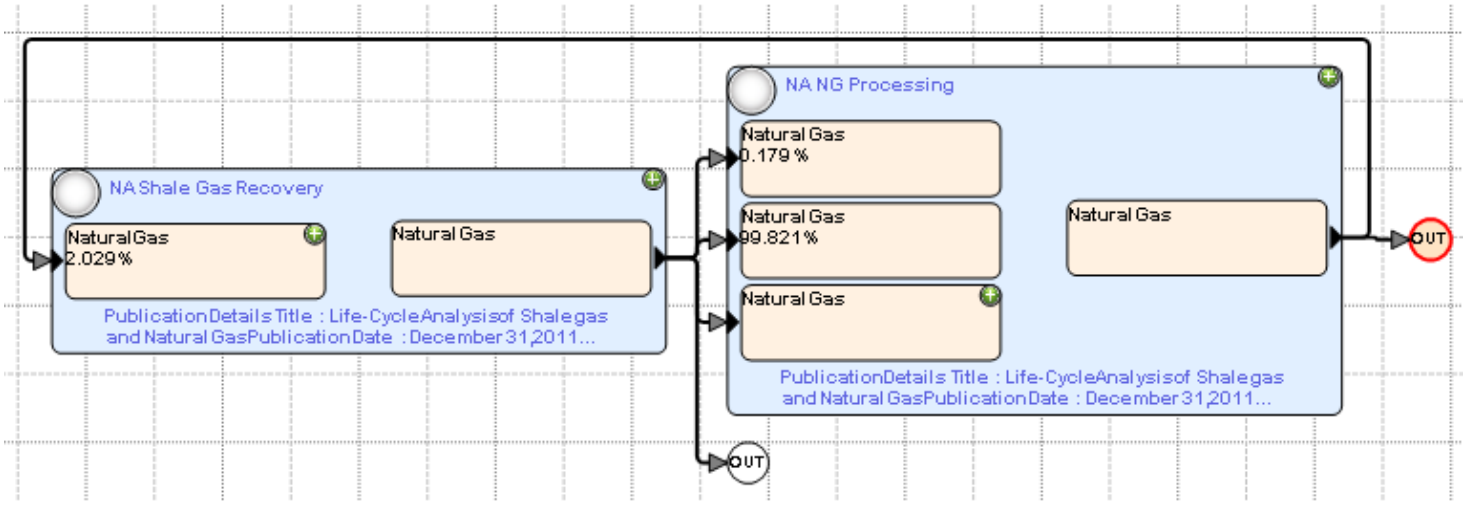


Figure 64: Pathway Is Done

Now that the pathway is done we can enter a name for it, notes related to the assumptions used in that pathway, and finally click the **Add as New**

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 65.

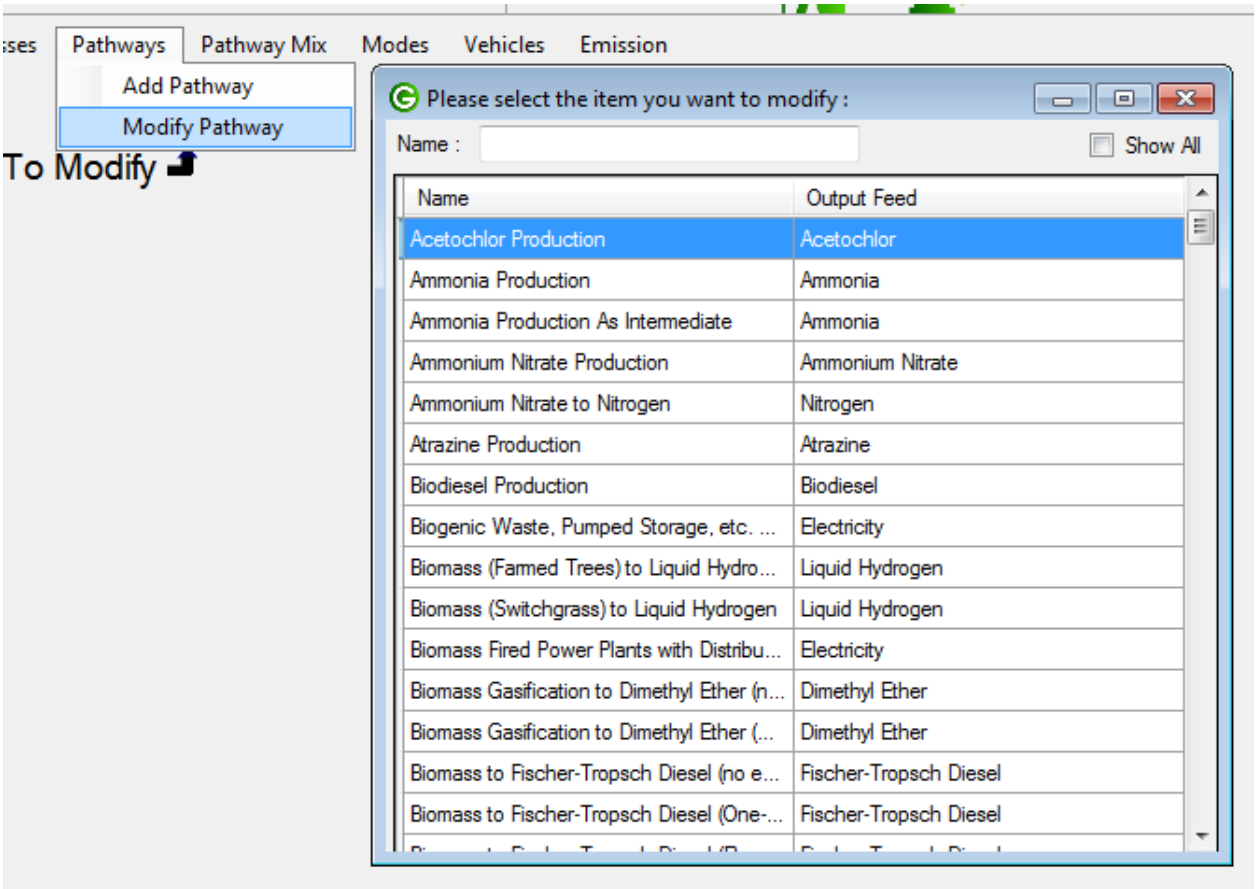


Figure 65: Select an Existing Pathway for Modifications

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

To delete a process, select it by clicking on the process, then hit the textbfDelete key on your keyboard. Keep in mind that deleting a process might break the integrity of the pathway and warning messages might be shown.

Using pathways and mixes as feedstock In order to take the full advantage of the GREET data model, pathway and mixes that have already been created can be reused. They can be reused in the process editor when the textbfsource of an input is set to Pathway or Mix, but they can also be reused in a pathway when the textbfsource of an input is set to tetbfOutput of a previous process.

In order to do that we're going to reuse the pathway created two paragraphs ago, and use it as a feedstock for another process in a new pathway.

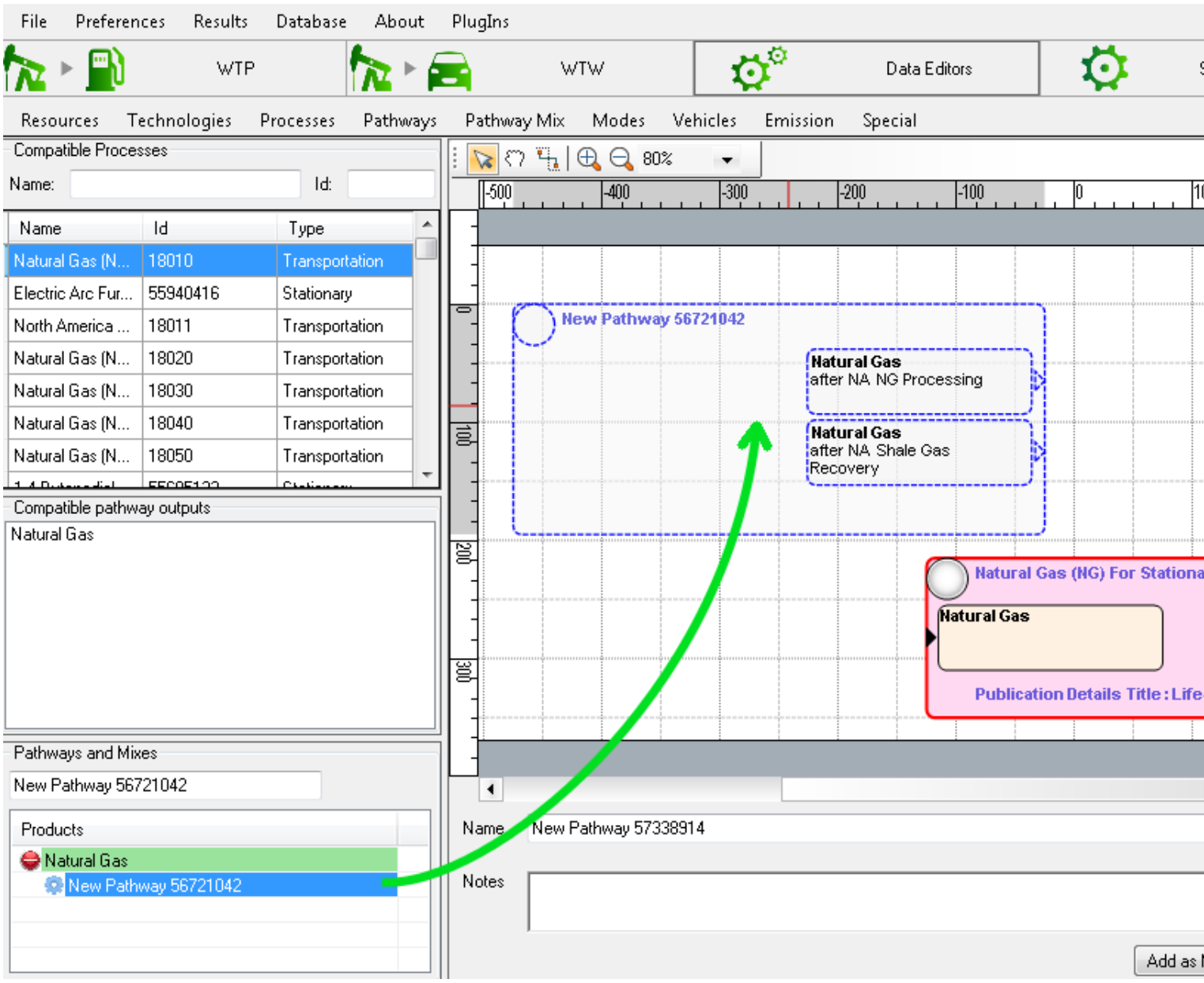


Figure 66: Using a pathway as a feedstock for another pathway

As it can be seen in figure 67 a pathway can be dragged and dropped from the **Pathways and Mixes** zone to the building area. By doing this the entire pathway appears to be encapsulated in it's own system. As we defined two outputs for this pathway, it also has two outputs available now. The main concern here is to chose the correct output. In order to do so, one may use the labels in the outputs to help finding the desired output, or use one, run the simulation and click on each of the outputs to compare the results and find the one that is desired.

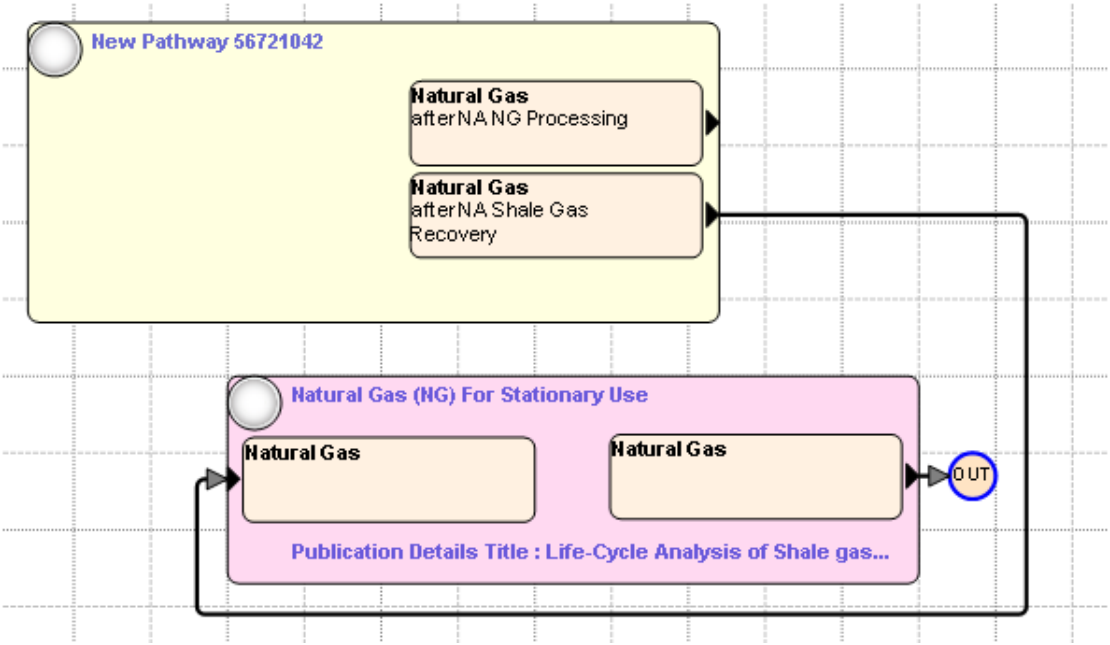


Figure 67: Using a pathway as a feedstock for another pathway

The same technique can be used to use previously defined textbfmixes as a feedstock for other pathway. Just as by drag and dropping a pathway into another pathway, a mix can be dropped into a pathway and it's input connected to the input of a process. The only difference being that mixes have a single output, while pathways may have one or more.

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 68. 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**.

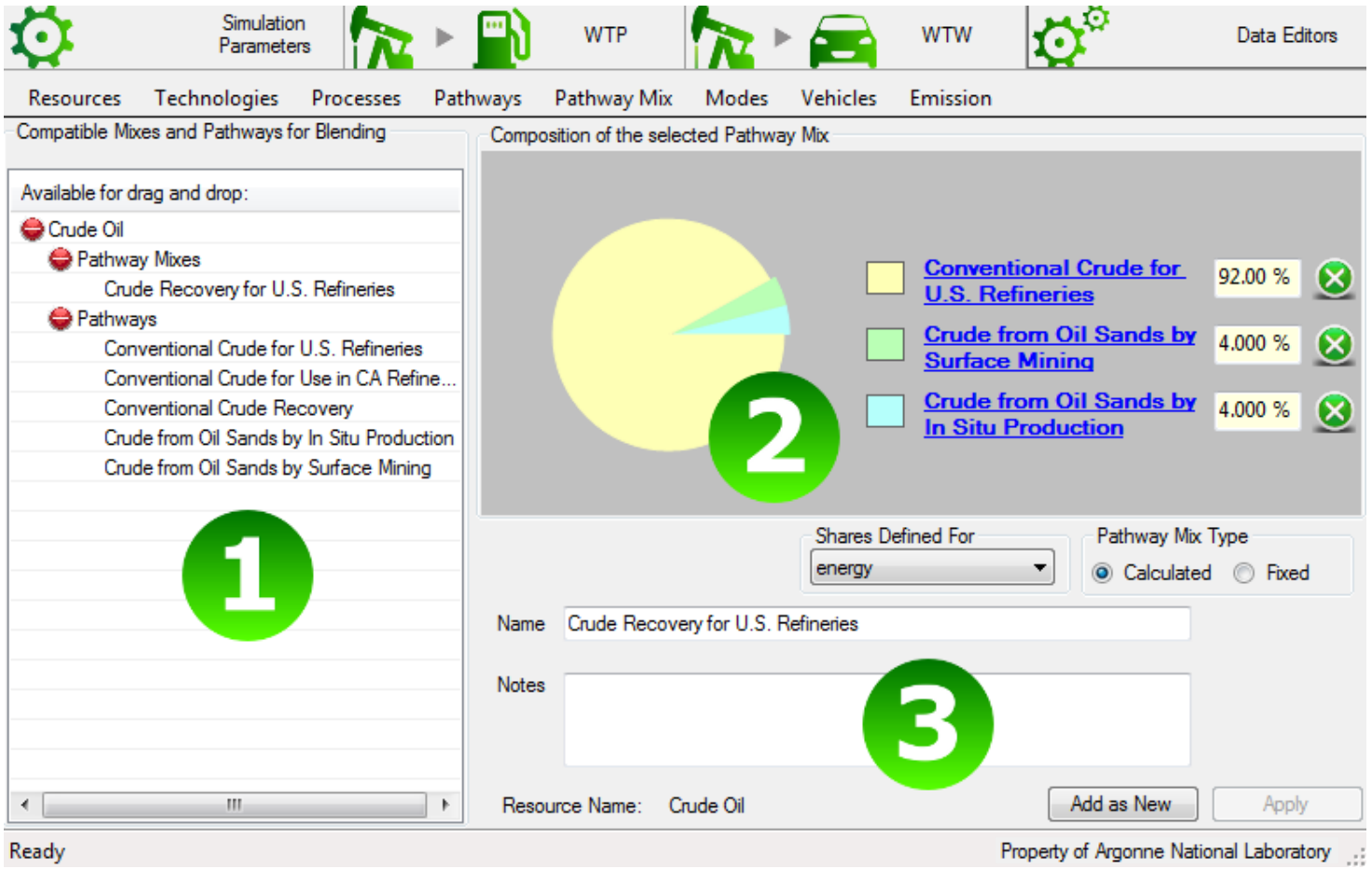


Figure 68: pathway mix Showing the Crude Recovery for U.S. Refineries

Create/Modify a calculated pathway mix: A calculated pathway mix represent an upstream defined by a weighted average of multiple pathways. The figure 68 represents such a pathway mix as seen in the editor.

The "Pathway Mix Type" option is set here as "Calculated".

In order to add new pathways to a mix, drag and drop them from the list of available items on the left (labeled 1) to the pie chart on the right. Then define shares for each of the pathways. The shares can be defined relatively to the energy content of the product, it's mass or it's volume. This option is particularly important when mixing different resources.

Create/Modify a fixed pathway mix: If detailed information for building a pathway is not available but the upstream energy and emission associated with the production of a resource is known, it is possible to define a "Fixed" pathway mix. In order to define a fix pathway mix, please chose the "Fixed" option in the "Pathway Mix Type" option on the right.

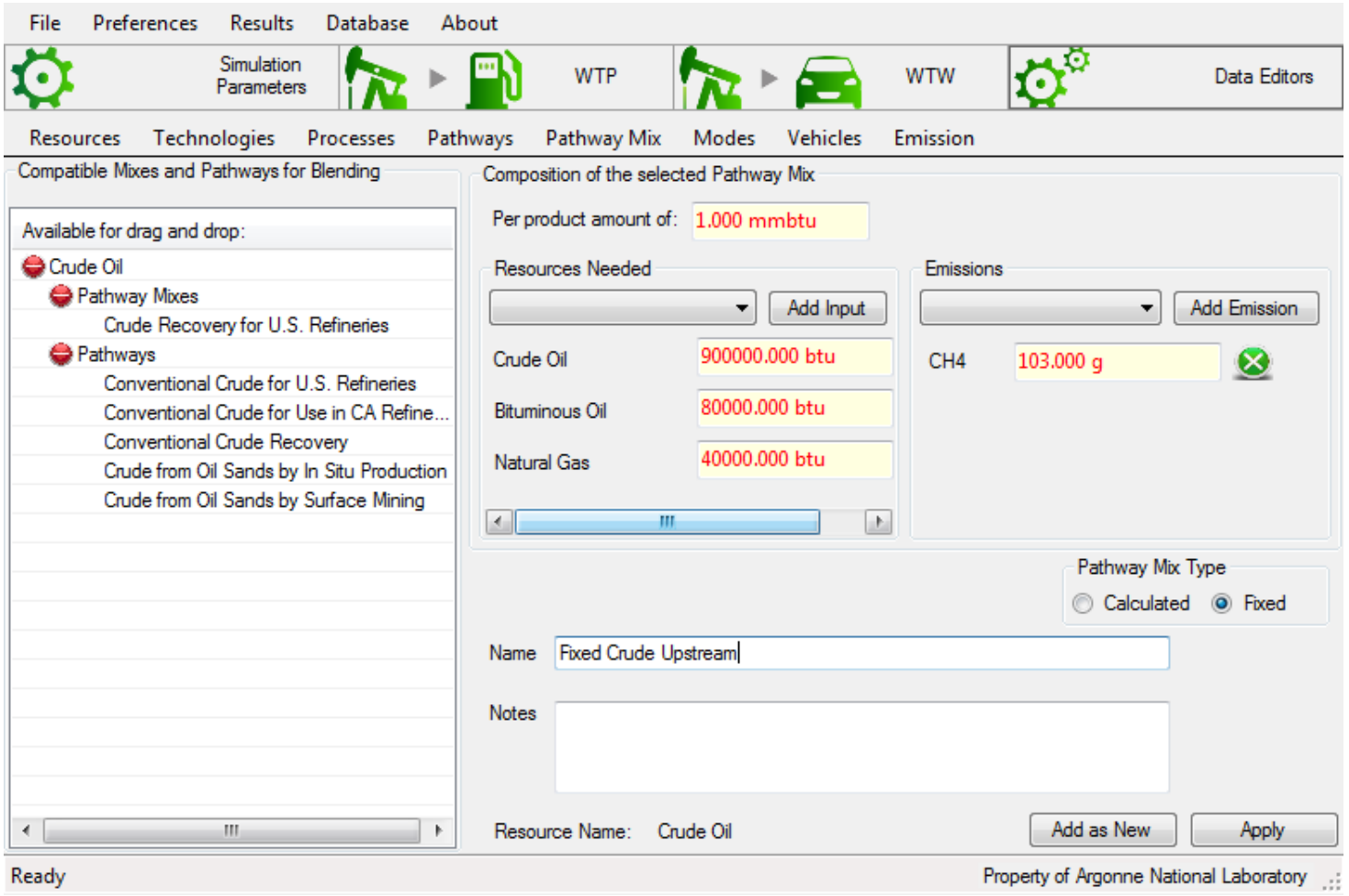


Figure 69: pathway mix with fixed upstream

In order to add resources and pollutants emissions, select the desired resource or pollutant in the drop down boxes and click "Add Input" or "Add Emission". Then fill up the amounts for these species and save you fixed upstream by clicking "Apply" or "Add as New".

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 **Modify Mode**.

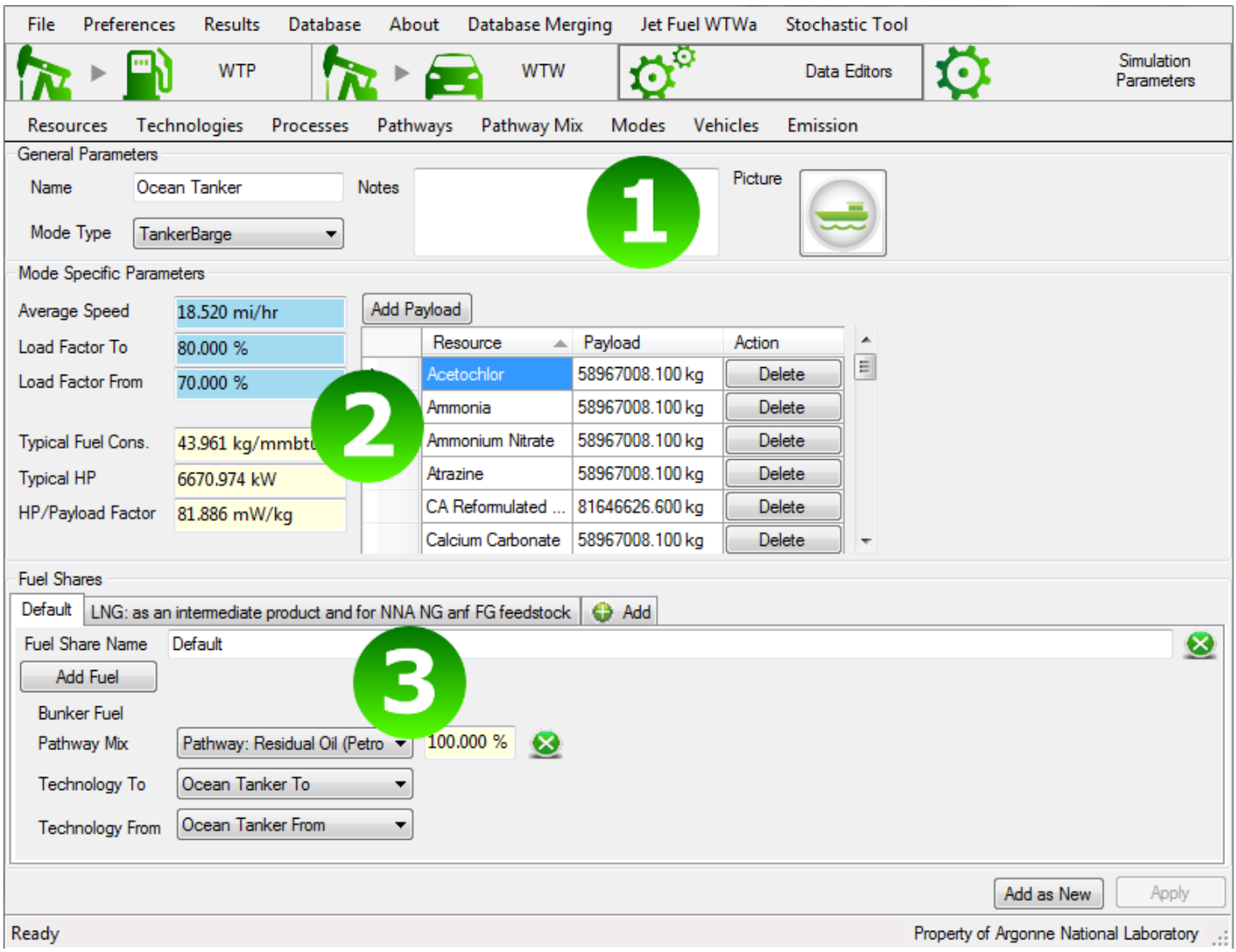


Figure 70: Mode Editor Showing the Parameters for the Ocean Tanker

The Modes Editor is composed of three different zones. See Figure 70. Zone 1 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 2 contains the specific parameters of a mode; these might change depending on which mode type is being used for the currently edited mode. Zone 3 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.

- Typical fuel consumption, Typical HP, Typical HP/Payload factor: These three parameters are used to estimate the energy intensity of the barge or ocean tanker using the following formulas:

$$ei(f_t, f) = \frac{ec(f) \times hp(f_t) \times LoadFactor}{PayLoad(f_t) \times Speed}$$

$ec(f)$ = Energy Consumption using a specific energy source (or fuel) f
hp = horse power required
f_t = resource transported
f = energy source (or fuel) used to power the barge or ocean tanker
$LoadFactor$ = engine load factor while cruising
$hv(f)$ = Heating value of the energy source $\rho(f)$ = Density of the energy source Speed = average travel speed while cruising
hp is calculated as follows:

$$hp(f_t) = TypicalHP + HPPayloadFactor \times Payload(f_t)[g]$$

$$ec(f) = \left(\frac{14.42 \left[\frac{g}{Wh} \right]}{LoadFactor} + TypicalFuelConsumption \right) \frac{hv(f)}{\rho(f)}$$

- 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 transported 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 71. A pop-up window will open asking you to select a resource transported.

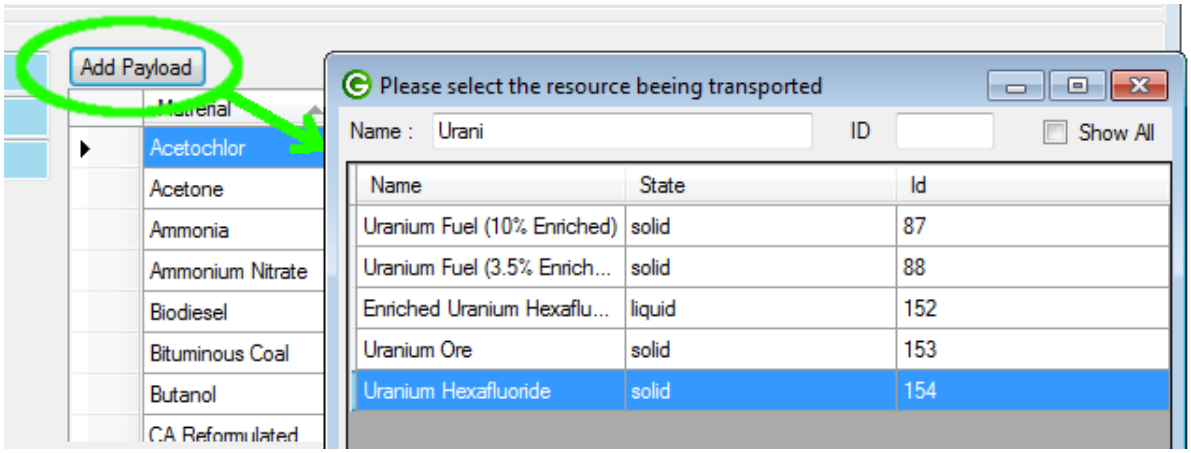


Figure 71: Selecting a Resource Being Transported after Clicking on the Add Payload Button

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 72.

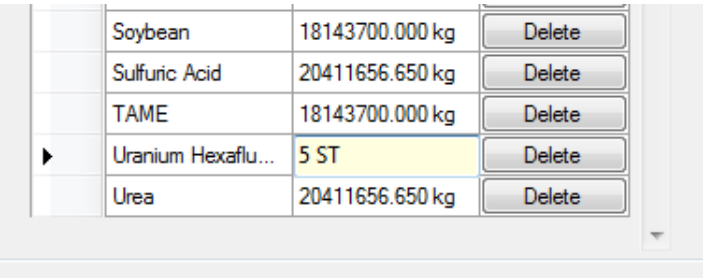


Figure 72: Entering Five Short Ton as a Payload for Uranium Hexafluoride

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.

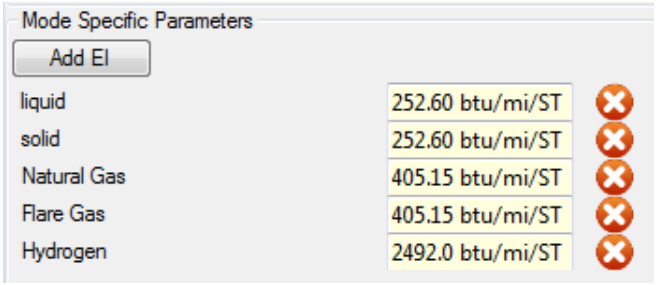


Figure 73: Energy Intensities for a Pipeline

Energy intensities (EI) can be added by clicking the **Add EI** button and then selecting the resource being transported. See Figure 73. 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.

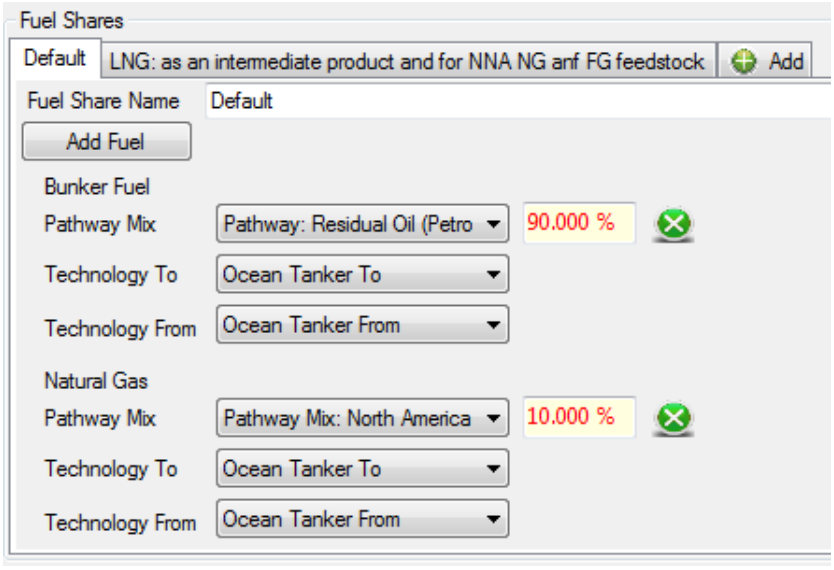


Figure 74: Fuel Shares for a Pipeline Mode

Figure 74 shows, as an example, the different fuel shares for an hypothetical ocean tanker. This assumes that the energy necessary to operate the ocean tanker is coming, on average, from bunker fuel and natural gas. Note: This is not the default data provided with GREET but just an example to illustrate the possibility of mixing different sources of energy for a mode.

Multiple 'configurations' can be saved for the mode, each tab in the fuel shares editor represent another energy source configuration that can then be used in any transportation process.

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. ;

3.8.8 Vehicles Editor

The vehicles editor was largely remodeled in the 2015 version of GREET. It allows you to set parameters of a vehicle such as fuel used, energy consumption, and emissions. See Figure 75.

To create vehicles, open the **Data Editor** main pane; then open the **Vehicles** menu and select **Add Vehicle**.

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

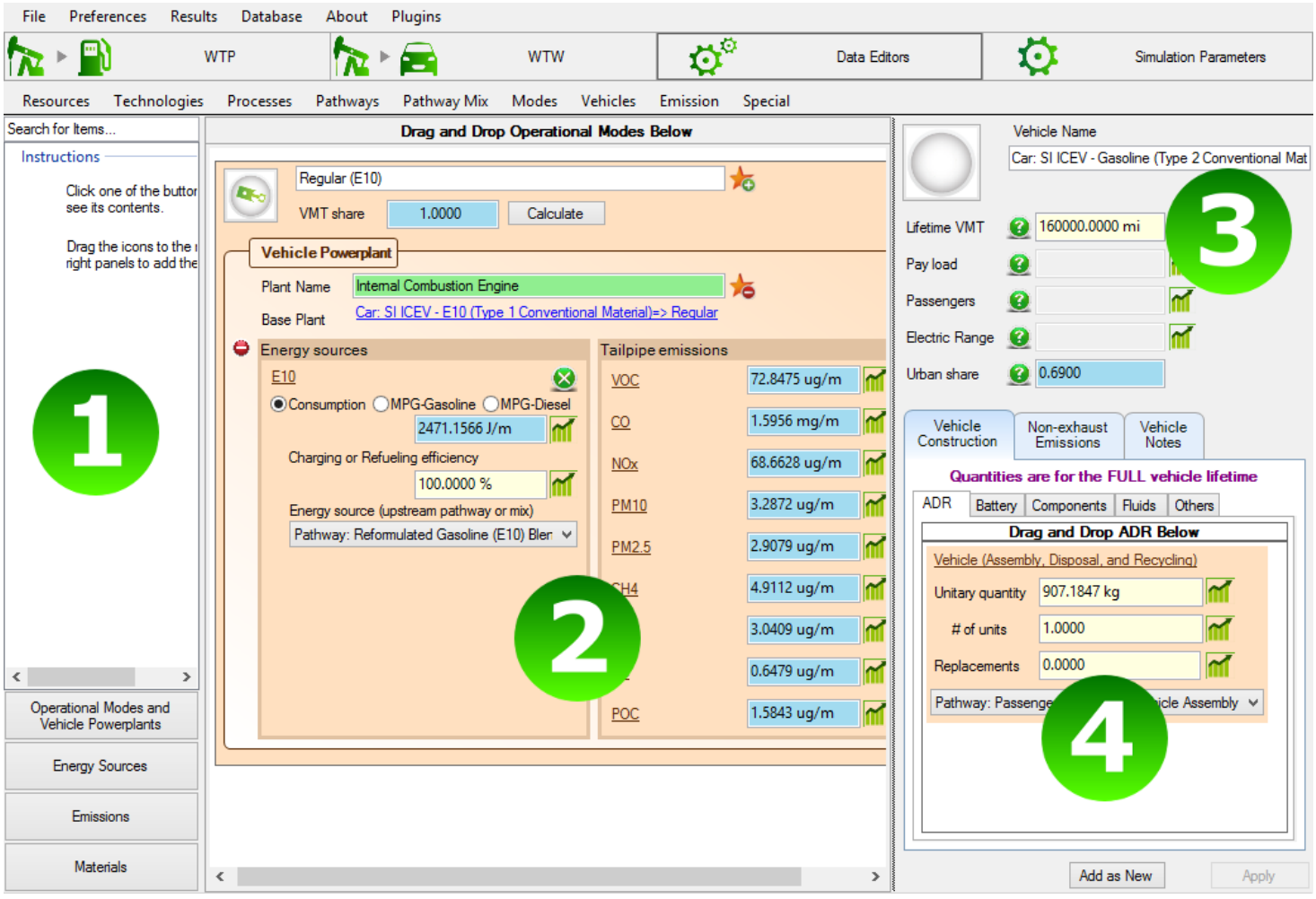


Figure 75: Vehicle Editor Showing the Properties for the Baseline Gasoline Vehicle

The vehicle editor is organized in 4 different zones.

- Zone 1 holds the items that can be used by the vehicle. These items are "Operational Modes" and "Vehicle Powerplants" that are used to represent the technology used and state of the control in the vehicle. The "Energy Sources" are used as fuels for the vehicle, or in order to provide a source of energy such as electricity. "Emissions" are to define the tailpipe and non-exhaust emission factors of the vehicle and finally "Materials" are used in the definition of the vehicle construction items for the vehicle (body, gearbox, engine oil, battery, ...)

- Zone 2 represents the vehicle modes, powerplants, energy used and emission factors. We'll explore this zone in more details in the next paragraph.
- Zone 3 defines the generic parameters that are available for all vehicles. Name, lifetime vehicle mile traveled, payload, urban share...
- Zone 4 stores the optional data for vehicles which can be vehicle construction information, non combustion emissions such as particle matters from brakes, and notes associated with that vehicle.

Items in the Zone 1 can be dragged and dropped onto Zones 2 and 4. This editor is fashioned to be used in a similar way as the stationary process editor. See the Figure 76 to see what items are can be dropped to where.

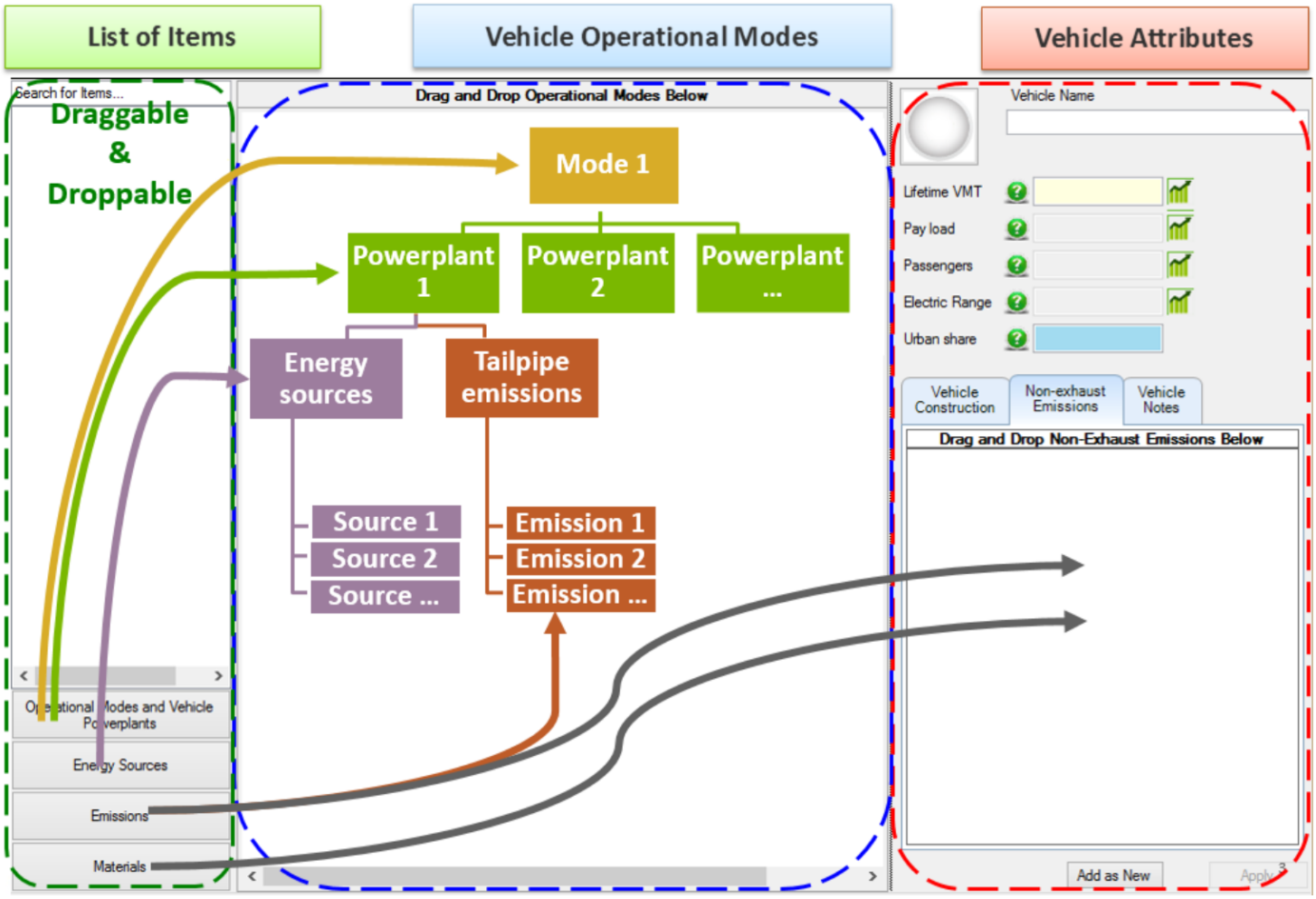


Figure 76: Drag droppable items can be used to build or make changes to a vehicle

As shown above the Zone 2 is used to build a tree structure where the mode encapsulates a powerplant, which itself encapsulates energy used and chemicals emitted.

Vehicle Operational Modes A vehicle may be controlled using one or more modes of operation. For example, a conventional vehicle with only one internal combustion engine has usually one mode of operation. On the other hand, a plugin hybrid vehicle has at least two modes of operation: the internal combustion engine and the electric motor (or both together). In GREET, we usually model these vehicles using two modes: Charge-depletion (CD) and charge-sustaining (CS) mode.

- 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.

Vehicle parameters are characterized for each of the mode. Because emissions and fuel consumption are different for powerplants depending on how they are utilized.

When multiple modes are used for a vehicle they show as "blocks" on the Zone 2. See Figure 77.

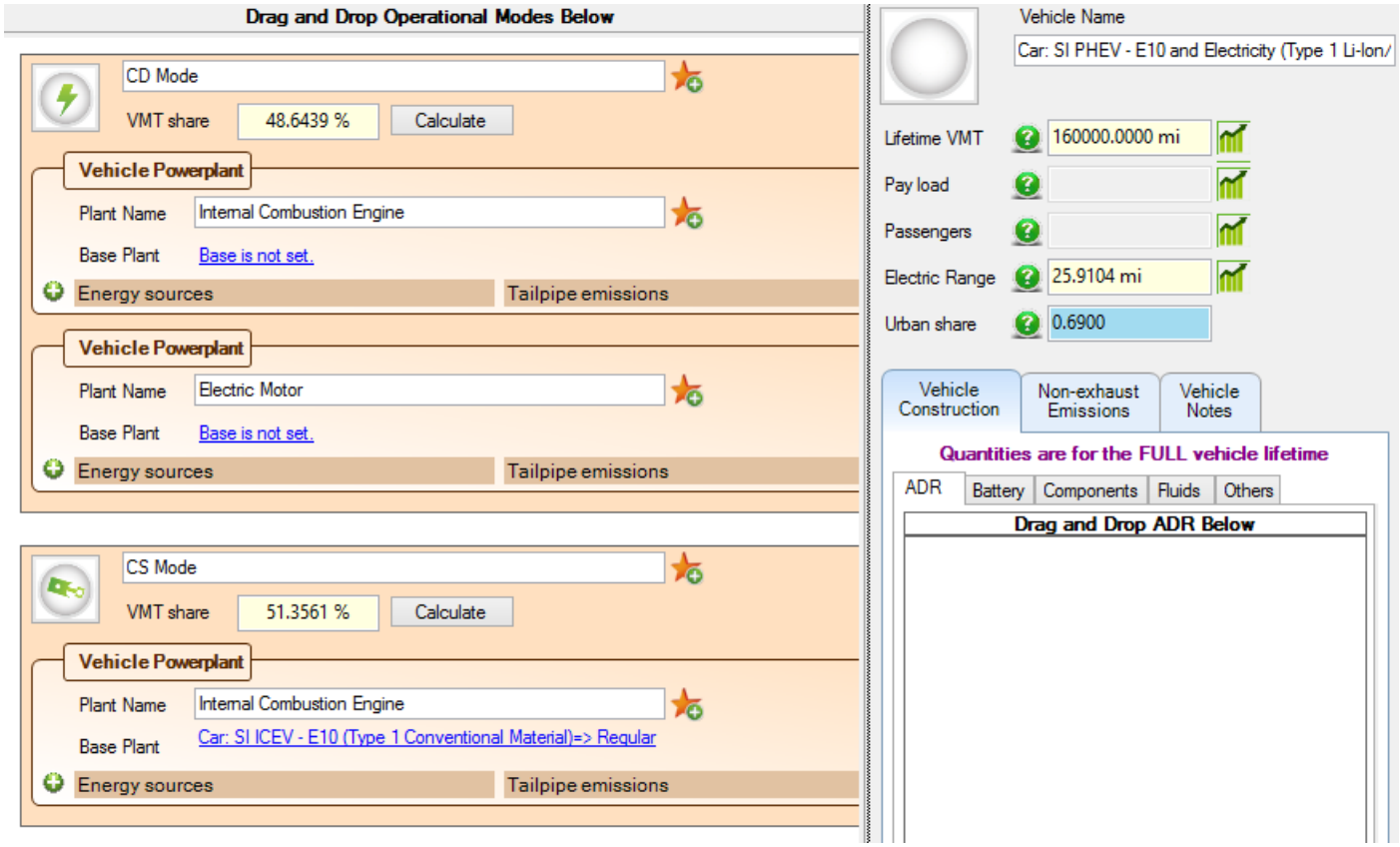


Figure 77: Charge Depletion and Charge Sustaining Modes Showing as Blocks in the Vehicle Editor. Note that for the CD mode we define two different powerplants. For both operational modes we also define VMT share which represent the share of distance over which the vehicle was running in that mode. VMT shares can be entered by the user or calculated automatically by clicking "Calculate" button based on utility factor and electric range.

Powerplants The concept of powerplants was introduced in GREET.net 2015 and did not exist in earlier models. It was created to better represent the technology used by each vehicle and to dissociate energy and emission for powerplants. In the case of vehicles one can imagine powerplant as being an internal combustion engine, a fuel cell, an electric motor, or maybe a gas turbine.

The powerplant offers us the possibility to include hypothetical vehicles and measure their impact on a WTW basis.

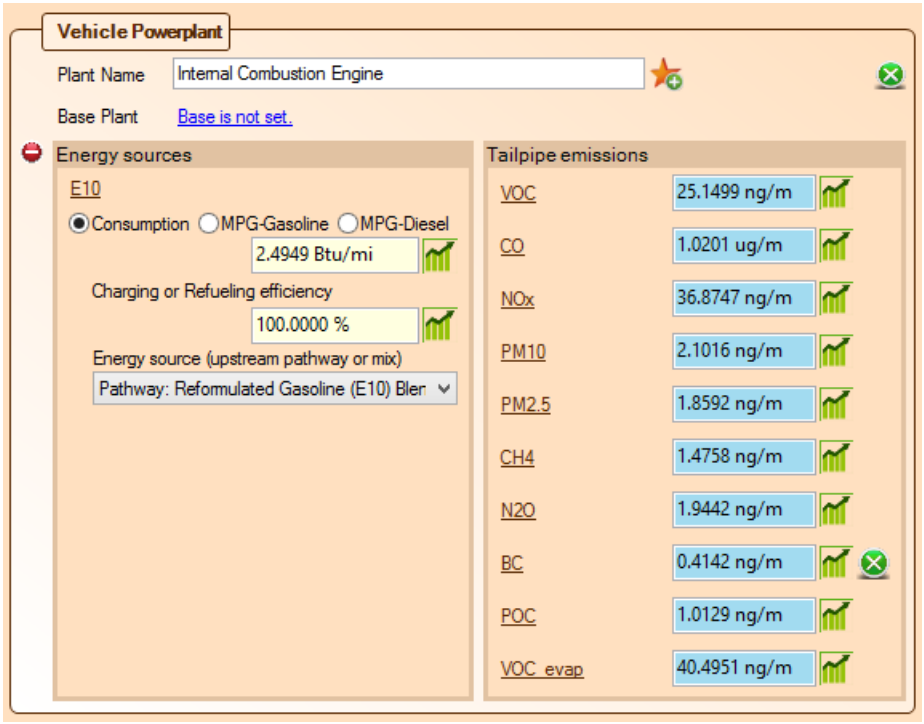


Figure 78: Example of an internal combustion engine powerplant within a mode

Powerplants - Energy use Powerplants must define energy consumption and emissions. As seen in Figure 78 a powerplant has a name. It may or may not have a base plant, the base plant will be used to calculate ratios for energy consumptions and emissions between the base and the current plant.

Energy consumption or MPG gasoline equivalent or MPG diesel equivalent must be given for each energy used. In the case of E10 in Figure 78, the charging efficiency is an abstract value that represents how much fuel is transferred between the pump and the tank of the vehicle. In the case of electricity we account for the charger losses between the wall and the vehicle battery. Each of the energy sources must be coming from a pathway or a mix, this is used to calculate the upstream results associated with the vehicle energy consumption.

Powerplants - Tailpipe emissions The exhaust emissions define the average emissions per mile. See Figure 78.

Time series Time series are available for all parameters of a vehicle. These are optional but creating many parameters for different years allows to see the progression over the years for certain results.

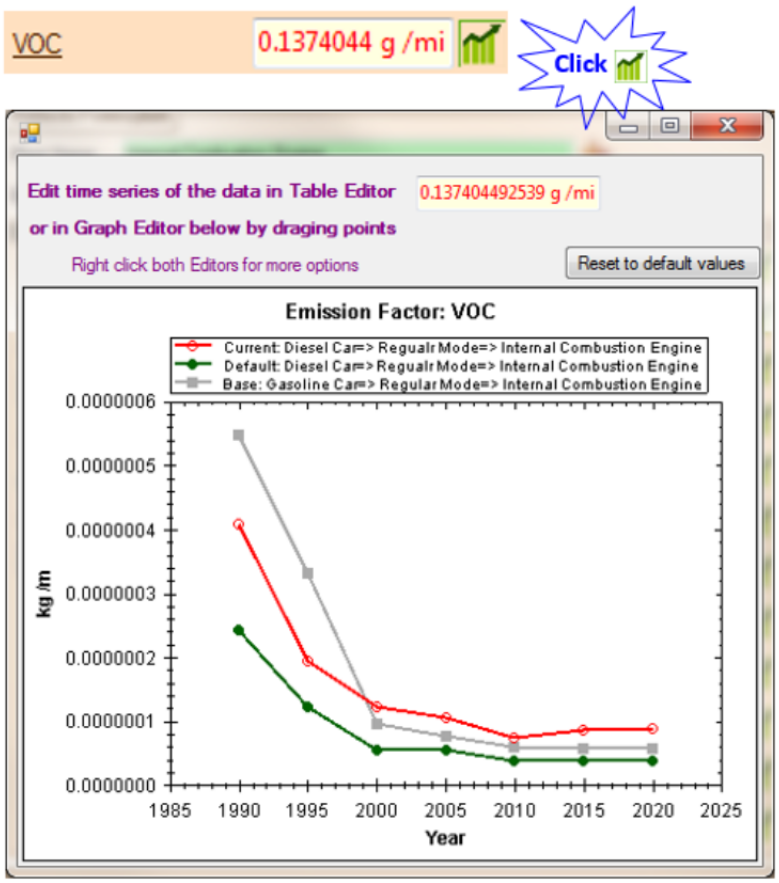


Figure 79: When clicking any of the chart icons, the time series for that parameter appears and can be edited

Vehicle attributes Attributes characterize the vehicle but most are optional. The name is required. However the payload is optional, if a payload is defined, then the results can be normalized using that payload and displayed in X/(ton mi) where X represents either an energy or a mass. The number of passenger is also used in the same manner, if passengers are defined then results can be expressed per passenger mile.

The Lifetime VMT is specific to the use of vehicle construction components. If we account for components, we must then know how many times they are going to be used or replaced over the lifetime of the vehicle in order to account for upstream energy and emissions associated with them.

Vehicle construction Example ADR, gearbox and engine oil. Most of the vehicle construction items defined by default in the GREET database are in the mass units. These masses are then multiplied by the corresponding pathways or mix used for the production of these resources.

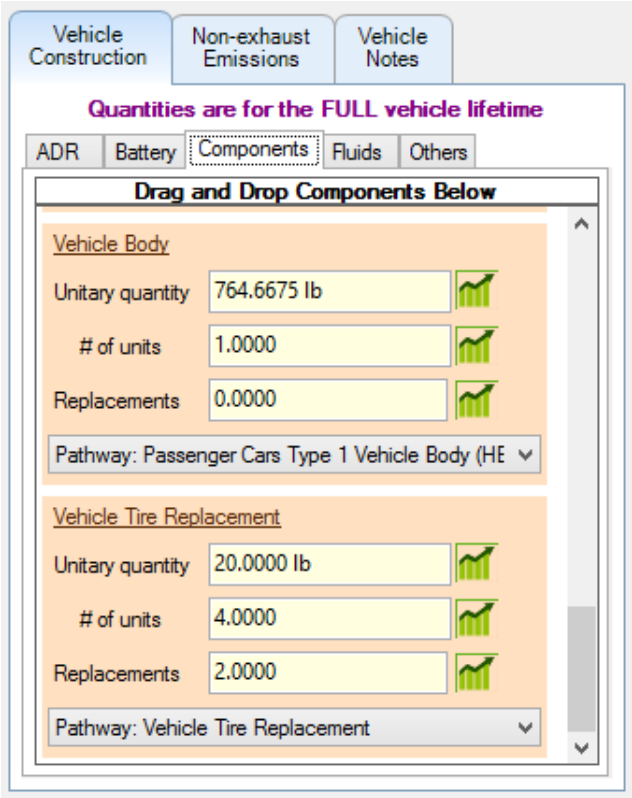


Figure 80: Vehicle Components

The vehicle components are defined by:

- "Unitary Quantity", the mass of a single item of this component
- "# of units", the number of units or number of unitary quantities on this vehicle
- "Replacements", the number of replacement for the all units on this vehicle over it's lifetime.

For example above in Figure 80, the Vehicle Body is never changed or replaced over the lifetime of the vehicle, thus replacements are set to 0. However we have 1 body per vehicle and it's mass is 764 lb for the "Car" SI HEV - E10". For the tires on that vehicle we have 4 of them, and we assumed that they will be changed twice over the lifetime of 160000 miles for that vehicle. Each tire weights 20 lb in the case shown above.

For each of these vehicle construction items, a pathways is associated so that the algorithms can use the upstream for the production of these components in the WTW results.

Other emissions: The non-exhaust 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 81.

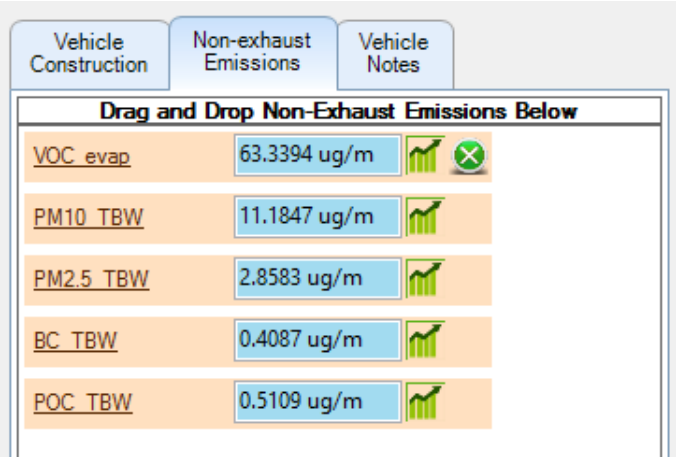


Figure 81: Vehicle Other Emissions

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.

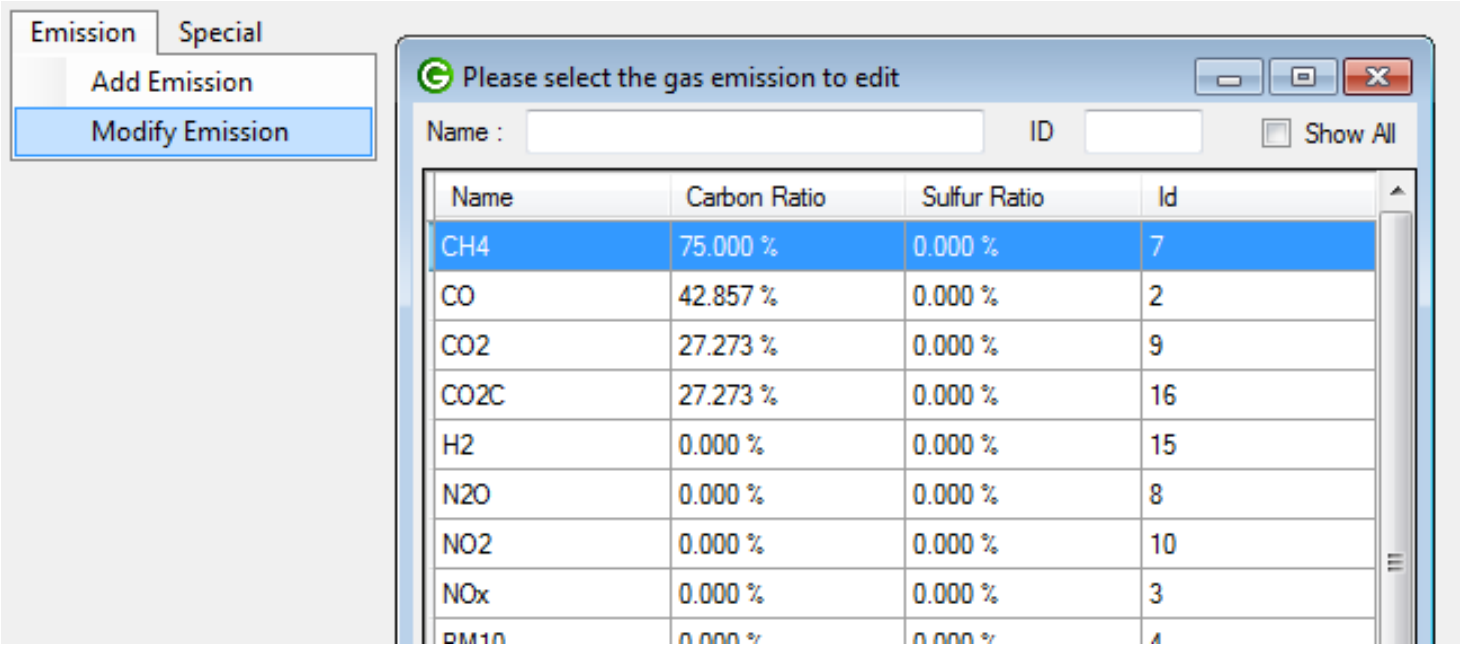


Figure 82: Selecting a Gas to be Opened in the Emissions Editor

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

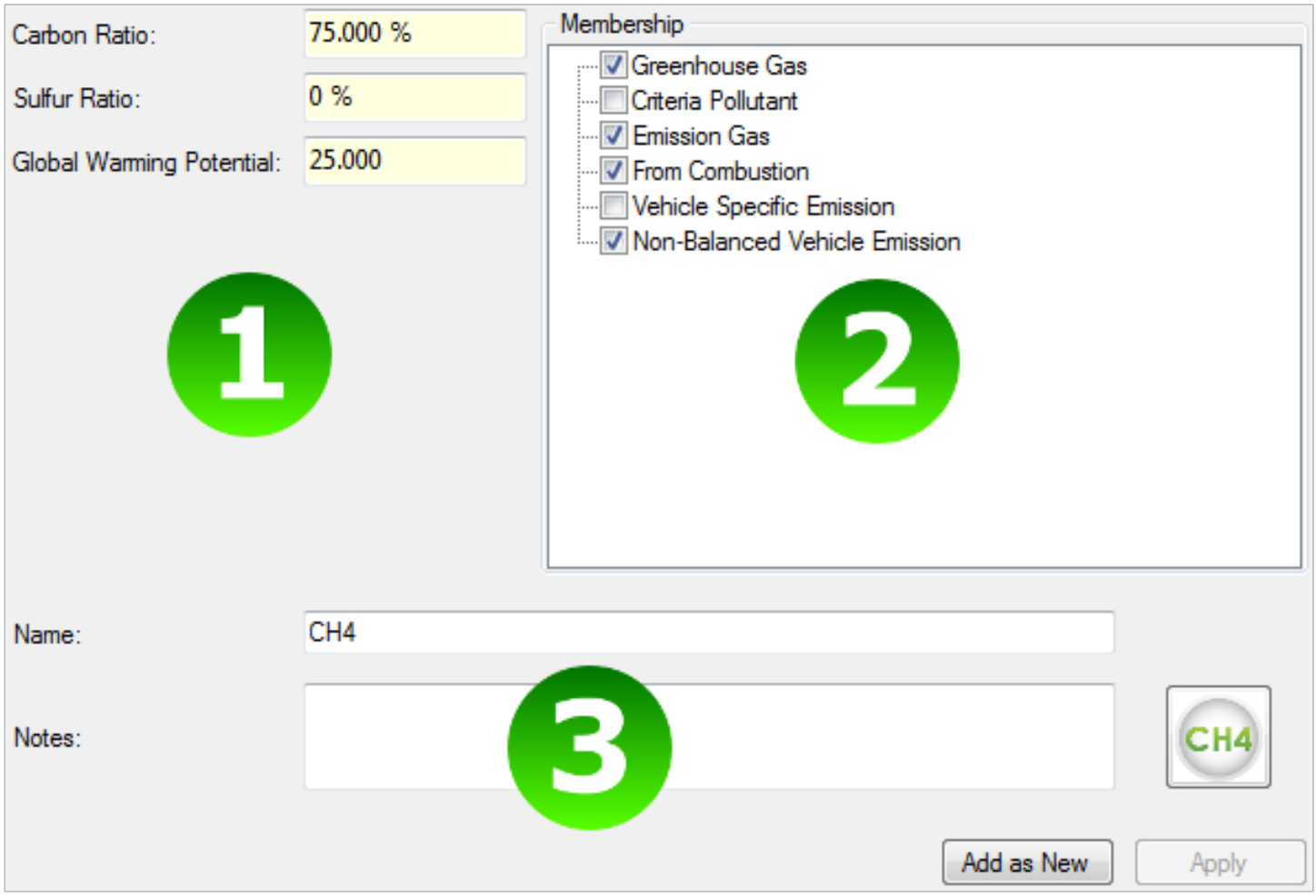


Figure 83: Emissions Editor

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.

3.9 Scenario Manager

The Scenario Manager allows the user to set up and save different values for the parameters in the model and associate these different values with "Scenarios". The graphic user interface can be used to create, manage, and save scenarios. Alternatively, the user may find it more efficient to work directly with the CSV file that stores all the parameters values for each scenarios.

Accessing the Scenario Manager
To access the Scenario Manager, click on the vertical bar on the very right edge of the GREET main window. The Scenario Manager tab should expand. The program will automatically look for a scenario file in User\Documents\Greet\Data\Scenarios\. If the default GREET database file Default2015.greet is loaded, then the scenario manager will try to load Default2015.csv in the Scenarios folder. If the file exists, its file path will be displayed in the text box at the top of the Scenario Mangager, under "Loaded Scenario File."

If no matching file is found, another file may be loaded with the "Load" button, or a new scenario can be created by clicking the "Save" button and typing in a file name. When the program is closed, it will automatically save the scenario .CSV file.

An individual scenario file can contain many scenarios, each with their own set of parameters and values for each of the scenarios.

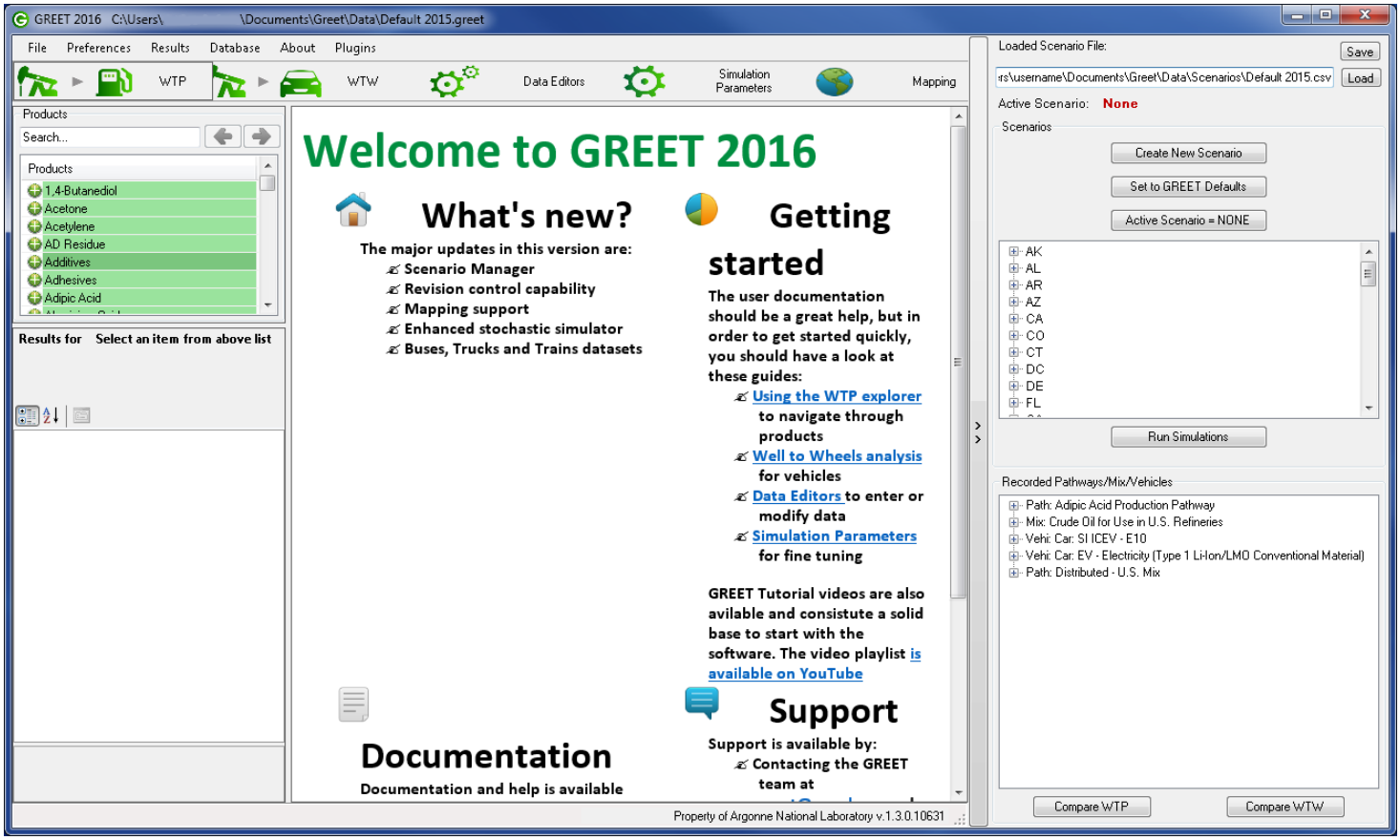


Figure 84: GREET main window with Scenario Manager expanded on the right

Setting parameters for a Scenario
Once a scenario file has been loaded or saved, the scenario manager will track all inputs that are changed from the loaded .greet database file. GREET can be used normally and inputs on pathways, mixes, and vehicles can be altered as necessary. Once the complete set of inputs has been changed for an individual scenario, clicking the "Create New Scenario" will save these inputs to a scenario named "New Scenario [date] [time]". It is recommended to rename this new scenario by right clicking on it and selecting "Change Scenario Name." Another scenario can be created by changing changing the necessary inputs to their new values, then once agian clicking "Create New Scenario." This process can be repeated until all scenarios have been created.

In the list of scenarios, each scenario can be expanded to show the pathways, mixes, and vehicles that have been modified. Each modified input can be listed and right clicking on an input will allow the user to jump to it in the editor. By default,

the scenario manager saves the parameter ID of each input. For more convenient input editing and tracking, the user can change the name of each parameter to be more easy to read.

Recording Results

Each scenario .CSV file can track a set of pathway, mix, or vehicle results that are compared for all scenarios. To add results, navigate to a pathway or mix in the WTP pane, or a vehicle in the WTW pane. In the editor pane, right click and select "Scenario Manager Record This Pathway." It will be added to the list of Recorded Pathways/Mix/Vehicles in the Scenario Manager.

General Use

Double clicking a scenario in the list will select it as the Active Scenario. The Active Scenario's inputs will be loaded and used for GREET calculations. Clicking on the "Active Scenario = NONE" button will stop saving modified inputs to a particular scenario, but will keep the modified values that were loaded by the previously selected scenario. Clicking on the "Set to GREET Defaults" button will also clear the active scenario, but will also return all inputs to default values.

Clicking on the "Run Simulations" button will run all GREET calculations for each scenario. Depending on the number of scenarios and the speed of the computer, this may take a few minutes to process.

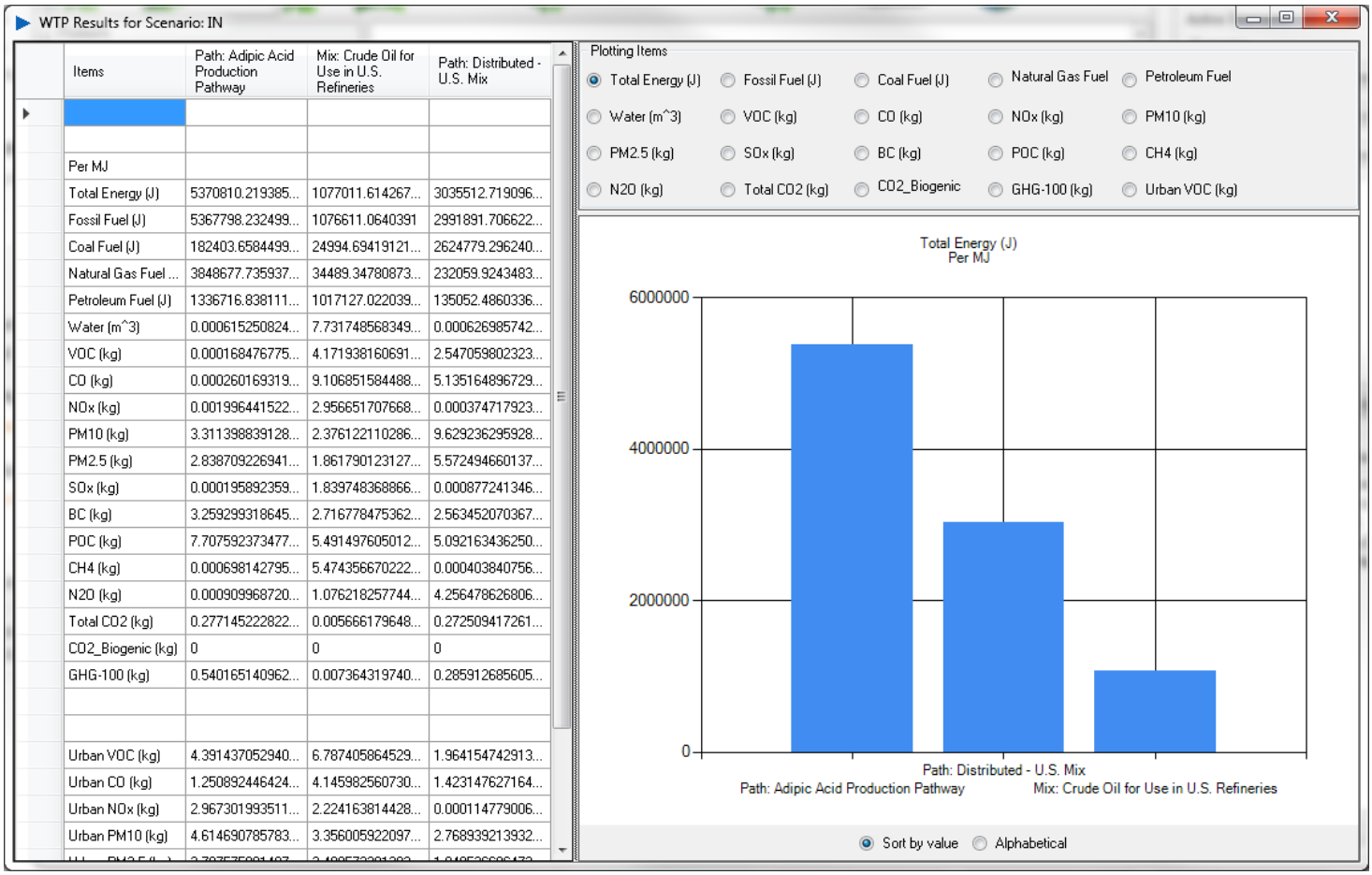


Figure 85: Compare WTP window showing a bar graph to compare three results in a scenario

Comparing Results

Once GREET has run all simulations, calculated results are recorded for each scenario. The "Compare WTP" or "Compare WTW" buttons at the bottom of the Scenario Manager will load a scenario selection window. This window allows the user to select which scenarios they would like to display results for.

A new WTP or WTW Results window will open for each selected scenario. The results window displays a table with all results for each pathway (WTP), mix (WTP), and vehicle (WTW) recorded by the scenario file. A bar graph is also shown to visually compare the results, and plotting items such as Total Energy (J) or Total CO2 (kg) can be displayed.

Scenario modification detected This warning message may appear when using the scenario or the mapping feature of GREET. When scenarios simulations or mapping simulations are run, the results are saved in a .scnres file in the same folder as the .csv file. This file stores all the results that were calculated previously so you do not need to recalculate everything when you stop and re-open GREET.

However imagine that between the last time you saved the scenario or mapping results and now, one of the parameter values has been modified. Then, the results you see may not be up to date anymore and you would need to recompute the results by clicking the button "Run all simulations"

The software automatically detect changes when loading a results file and may warn you that calculations in the scenario or mapping modules needs to be recomputed. The image below illustrate such a warning message:

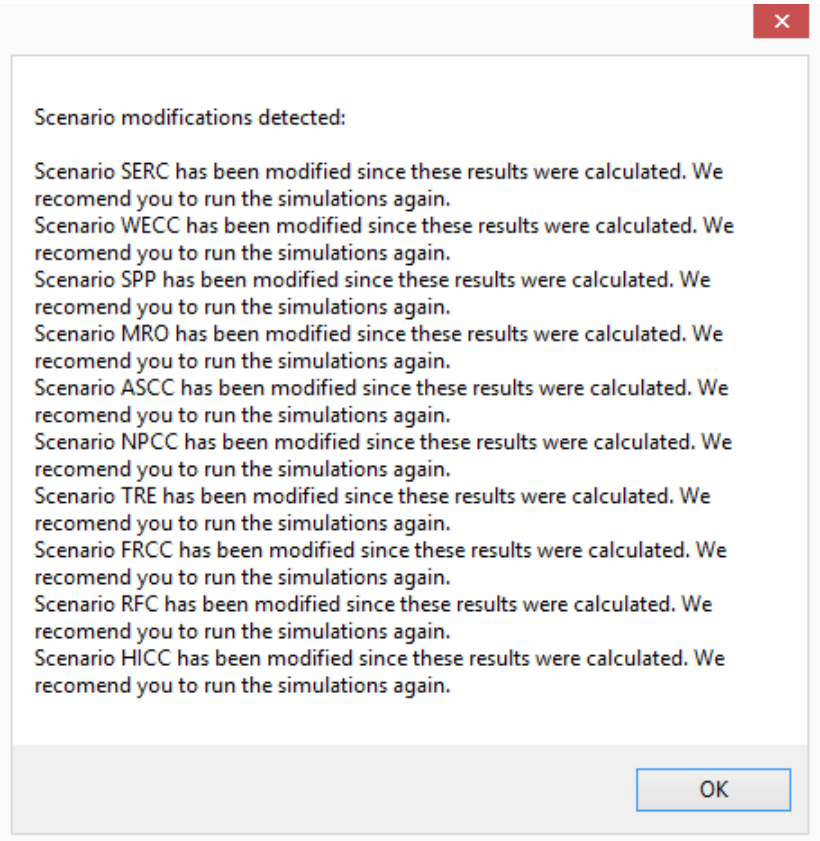


Figure 86: Warning when results needs to be recomputed for scenario or mapping modules

In order to fix this issue, simply go to the scenario or mapping modules and click "Run all simulations"

3.10 Scenario Mapping

GREET's mapping feature works in a very similar way as the Scenario Manager, but then applies each scenario to a particular state, region or country. By default, GREET includes a map for all 50 United States. Users can add more regions or nations by providing GIS shapefiles for custom scenarios.

Accessing the Mapping Feature
Mapping is featured as one of the five primary panes of GREET near the top of the main program window. Click on the Mapping button to begin. Similar to the Scenario Manager, the mapping feature will look for a .CSV file with the same name as the loaded .greet file in the User\Documents\Greet\Data\Mapping\folder. A different file may also be manually loaded or saved.

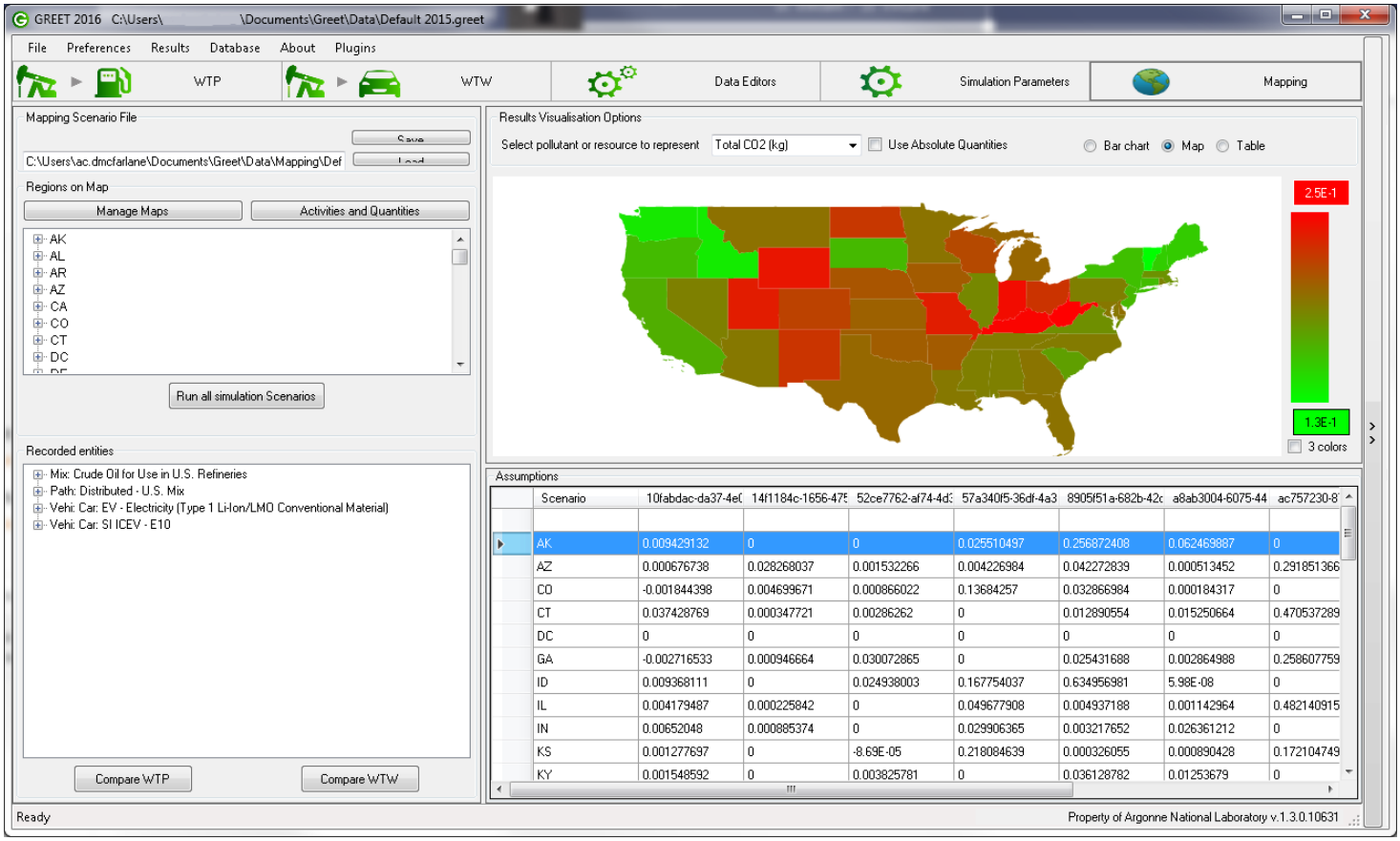


Figure 87: The Mapping pane in the GREET main window

General Use
Before using the mapping feature, users should already have created a scenario file using the Scenario Manager. This file may be copied into the \Greet\Data\Mapping\folder or simply loaded from its existing location. Each scenario needs to be matched with a location, state or region that will be mapped.

A GIS shapefile consists of geolocated *features* that identify components of the map such as individual states, regions, or countries. By default, GREET contains a United States map that has a feature for every state. The program will attempt to match the scenario names with the name or abbreviations included in the GIS shapefile for each feature. Clicking on the "Manage Maps" button will open the Map Manager, which lets the user select which map layers to display and assign each scenario to a specific feature (state, region, or nation) of the map.

In the included default US map, the GIS shapefile attribute field "NAME" contains the full spelled out name of each state, while the attribute field "STUSPS" contains the two letter abbreviation for each state. If the scenarios have not yet been assigned to a feature, the GREET program will attempt to automatically assign them based on whether NAME or STUSPS is selected. The user may provide their own shapefiles with their own custom attributes and scenario names, allowing them greater control or flexibility in aligning scenarios and mapping locations.

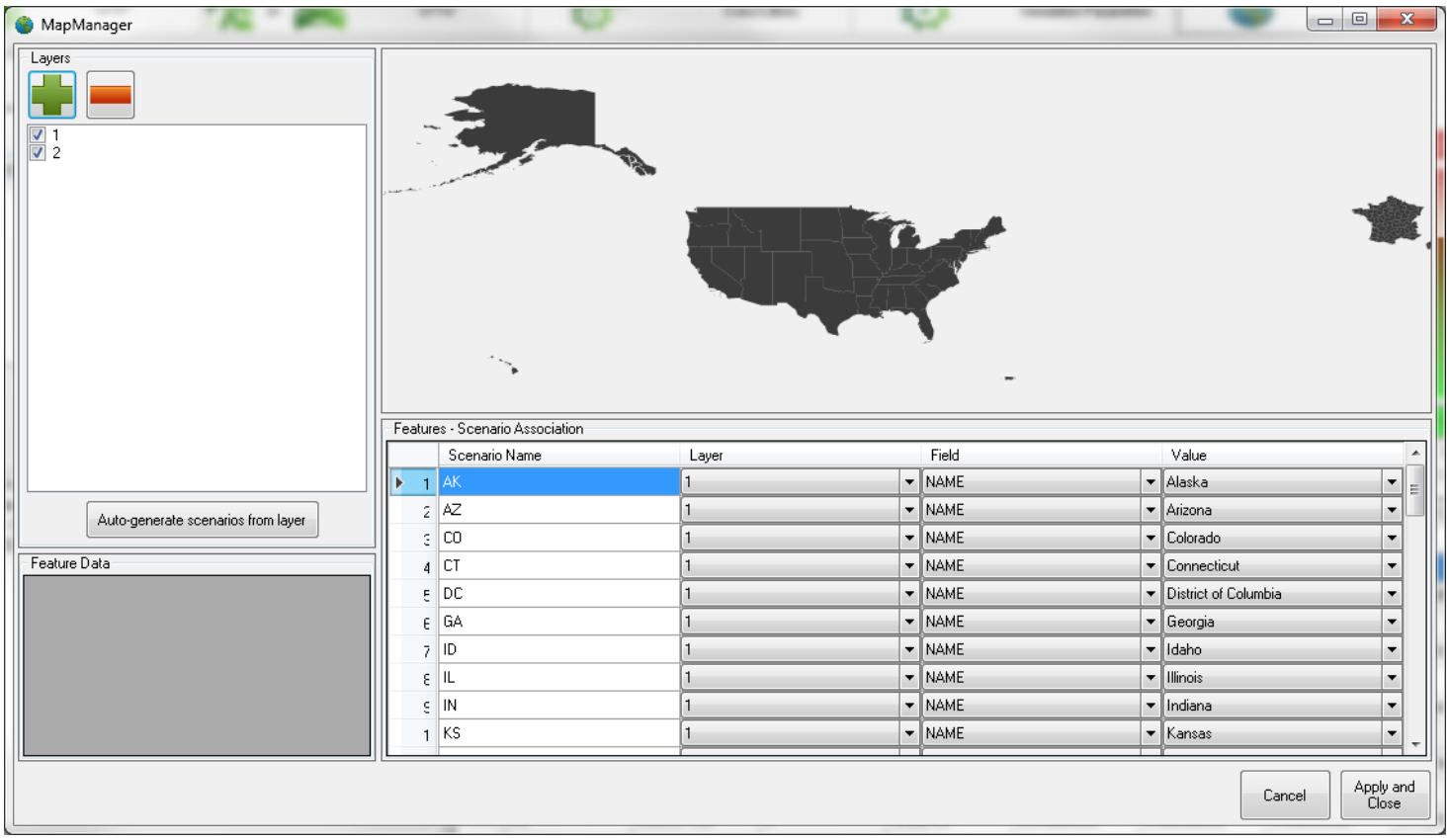


Figure 88: The Map Manager window that assigns scenarios to map regions

The rest of the left panel of the mapping tool is very similar to the Scenario Manager. Each scenario and its modified parameter inputs are listed, as well as the recorded pathways, mixes, and vehicle results.

Mapping and Graphing Results

GREET will generate bar charts, maps, or tables to compare scenario results. Click on a pathway, mix, or vehicle in the "Recorded entities" list to select it as the result to display. On the right display panel, a dropdown menu lets the user select which pollutant (CO2, CH4, etc) or resource (Total Energy, Fossil Fuel, Water, etc) to represent. Three radio buttons will select between displaying a barchart, map, or table.

When the map option is selected, the user can select the color range to represent values between the minimum and maximum results. By default, the two colored boxes at the top and bottom of the color bar can be clicked on to choose a color for each. A third color can be added as a mid-range value by selecting the "3 colors" checkbox and choosing a color. Finally, the map can be panned around with the mouse and zoomed in and out with the scrollwheel.

Scenario Output Quantities

By default, GREET provides energy and emission intensities for its pathway results, generally per energy (e.g. MJ), mass (e.g. kg), or distance (e.g. mi or km). When comparing different regions like states, it is often useful to consider the absolute quantity of emissions or energy use, rather than just the intensity. For instance, one may need to compare the total annual emissions produced by vehicle fleets in different regions or states with unique fuel inputs and measured vehicle miles traveld (VMT). The GREET mapping tool allows for this type of comparison by providing the option to define quantities of each output for each scenario.

The "Activities and Quantities" button will open a new window that lists each scenario or location and includes an entry text box for each result output. Once a scenario is selected, the output quantity for each result should be entered in each text box. This process is repeated for each scenario. The available units for each quantity is autopopulated by GREET, and only needs to be selected once for each result. Changing the unit will apply the selection to all scenarios. When "Use Absolute Quantities" is checked on the bar chart, map, or table, these values and units will be used for the graphic.

Finally, user provided quanitity values and unit selections will be saved in the mapping scenario file along with the parameter inputs.

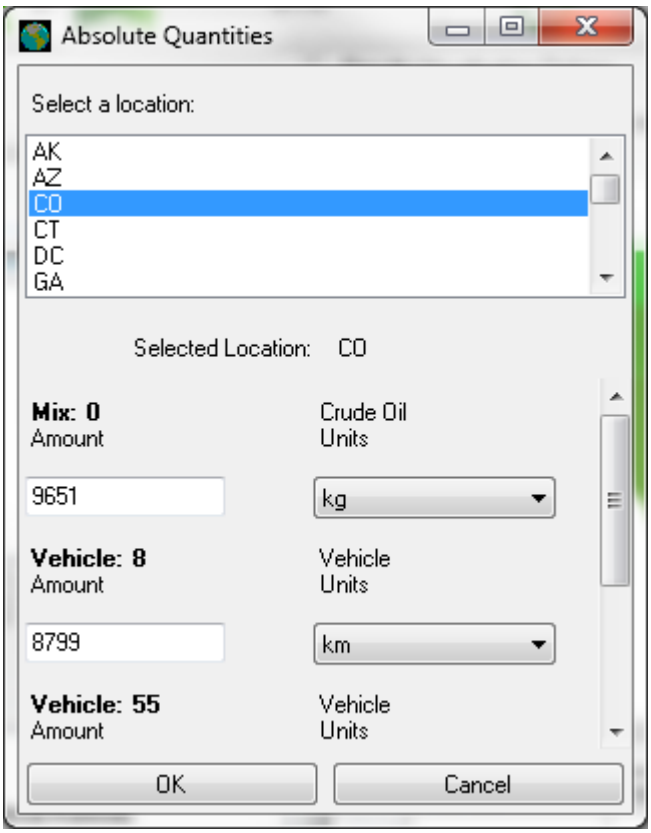


Figure 89: The quantity editor window that sets output quantities for each result in each scenario

4 Auxiliary 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 90.

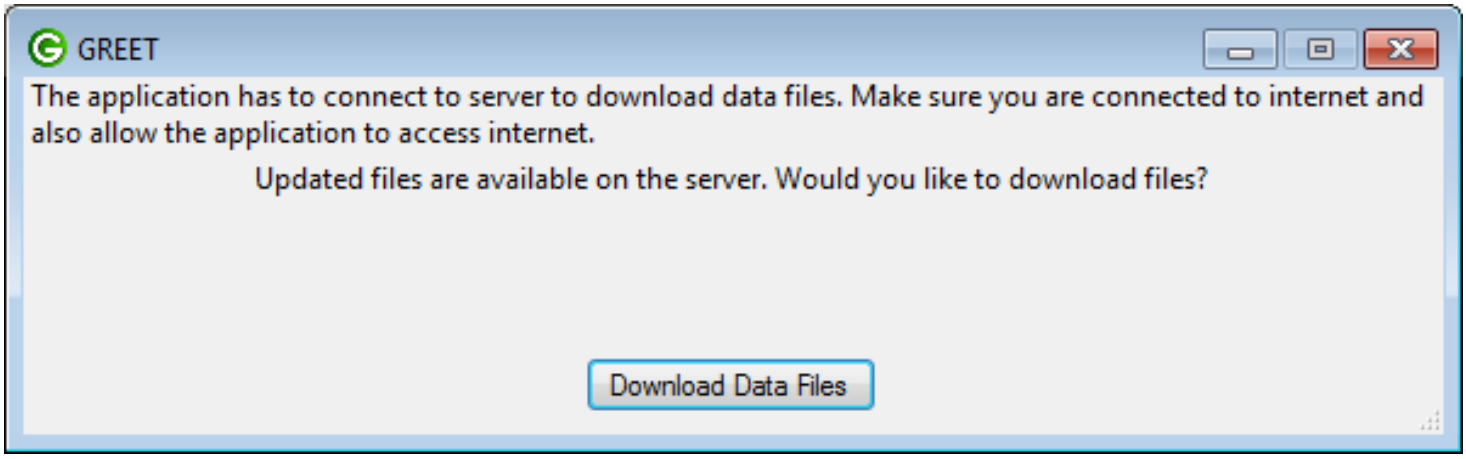


Figure 90: Download Data Files Option

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 91. 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.

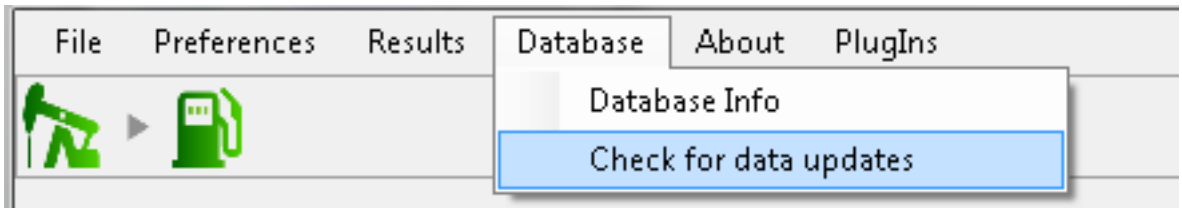


Figure 91: Database Menu

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 92.

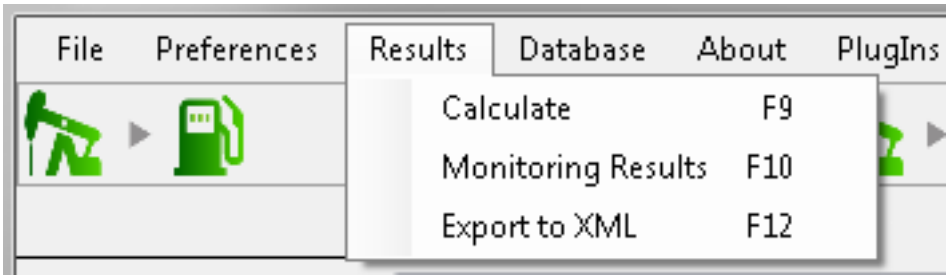


Figure 92: Results Menu

A new window will appear showing you all the monitored results. See Figure 93.

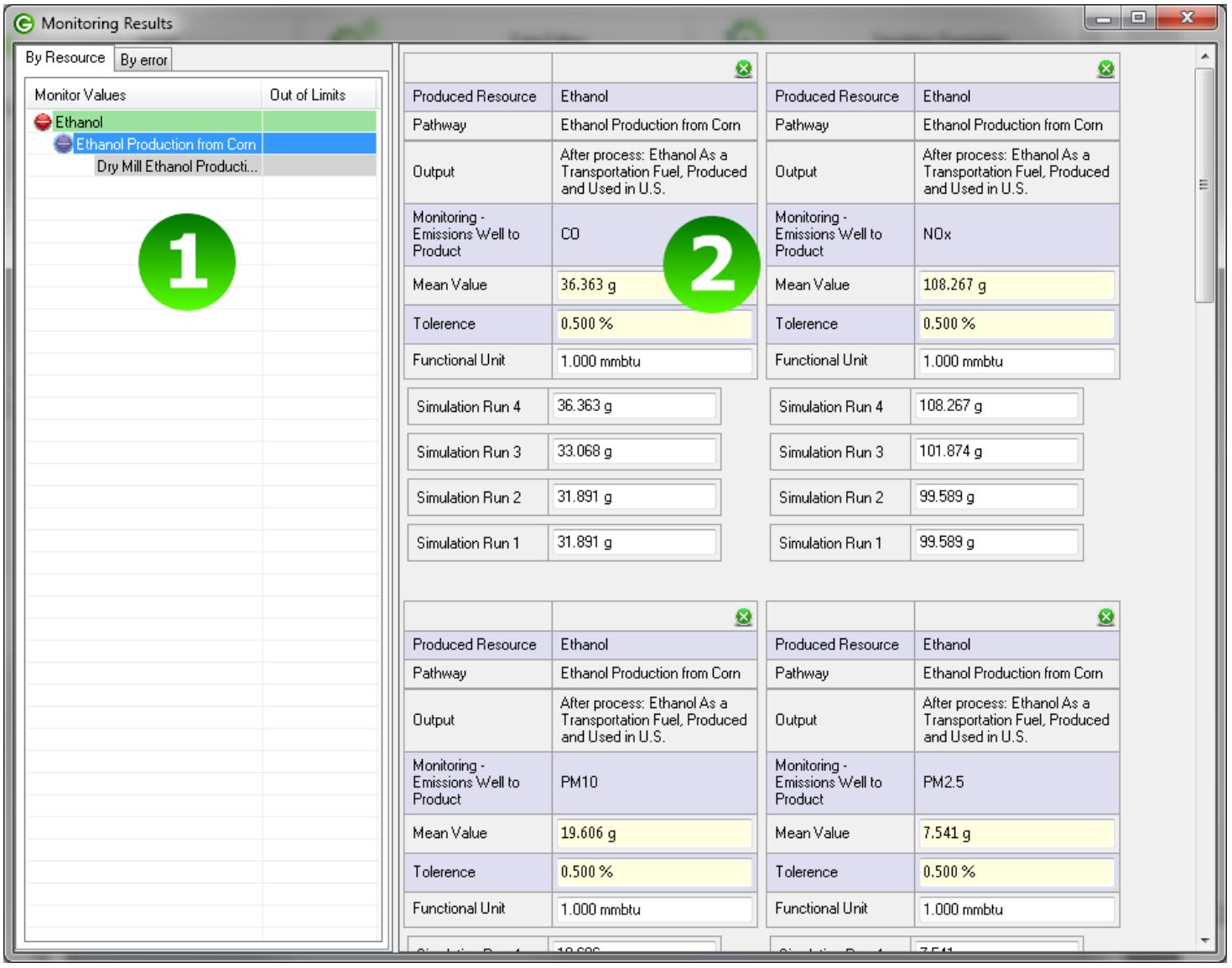


Figure 93: Result Monitored Window

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.

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 3.6.2 of this document.

4.4 Preferences

4.4.1 Units

The units displayed can be changed globally using the Global Parameters. See Figure 94. 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**.

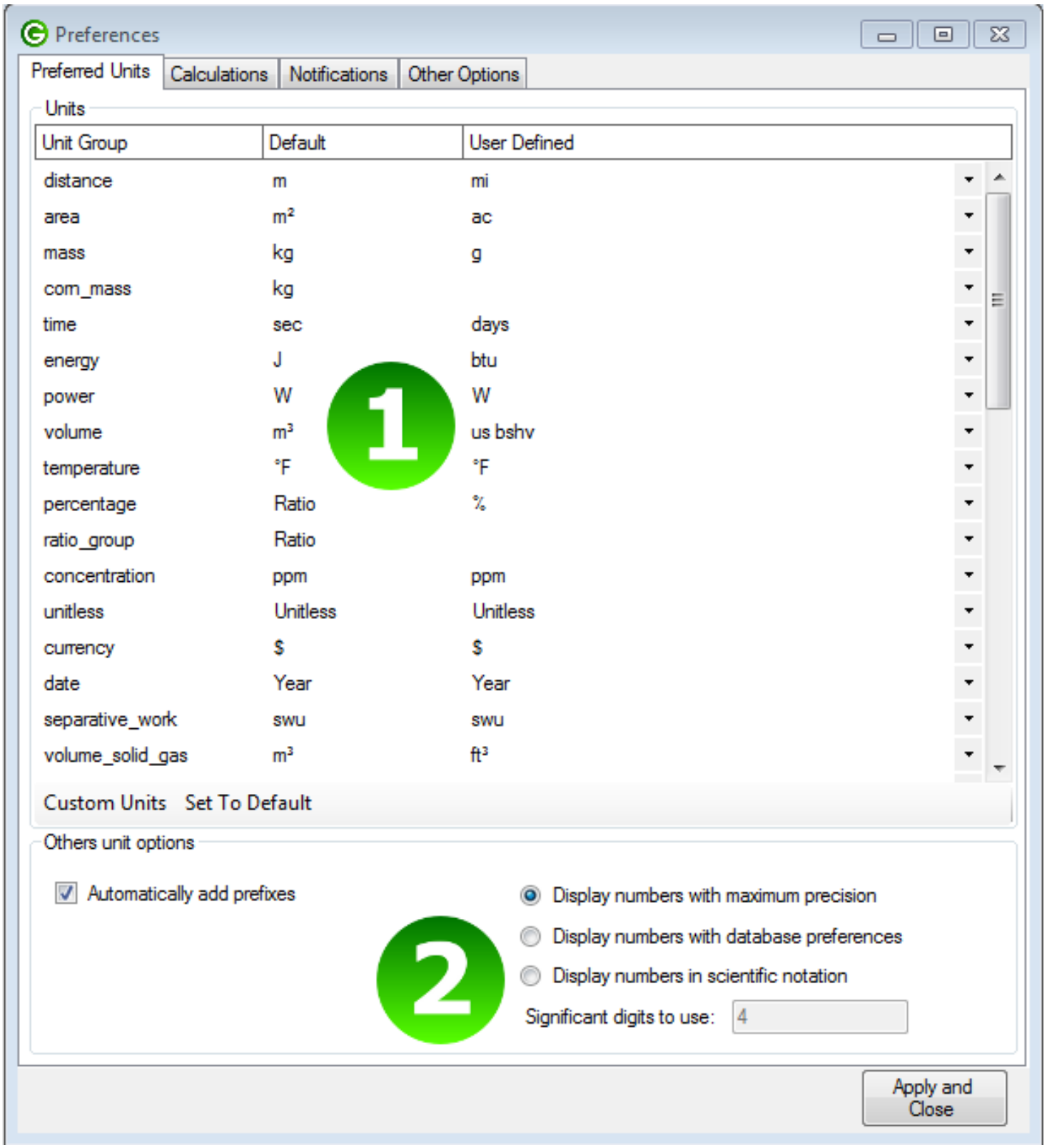


Figure 94: GREET Preferences

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 95. The preferences are contained in the **Running Options** section.

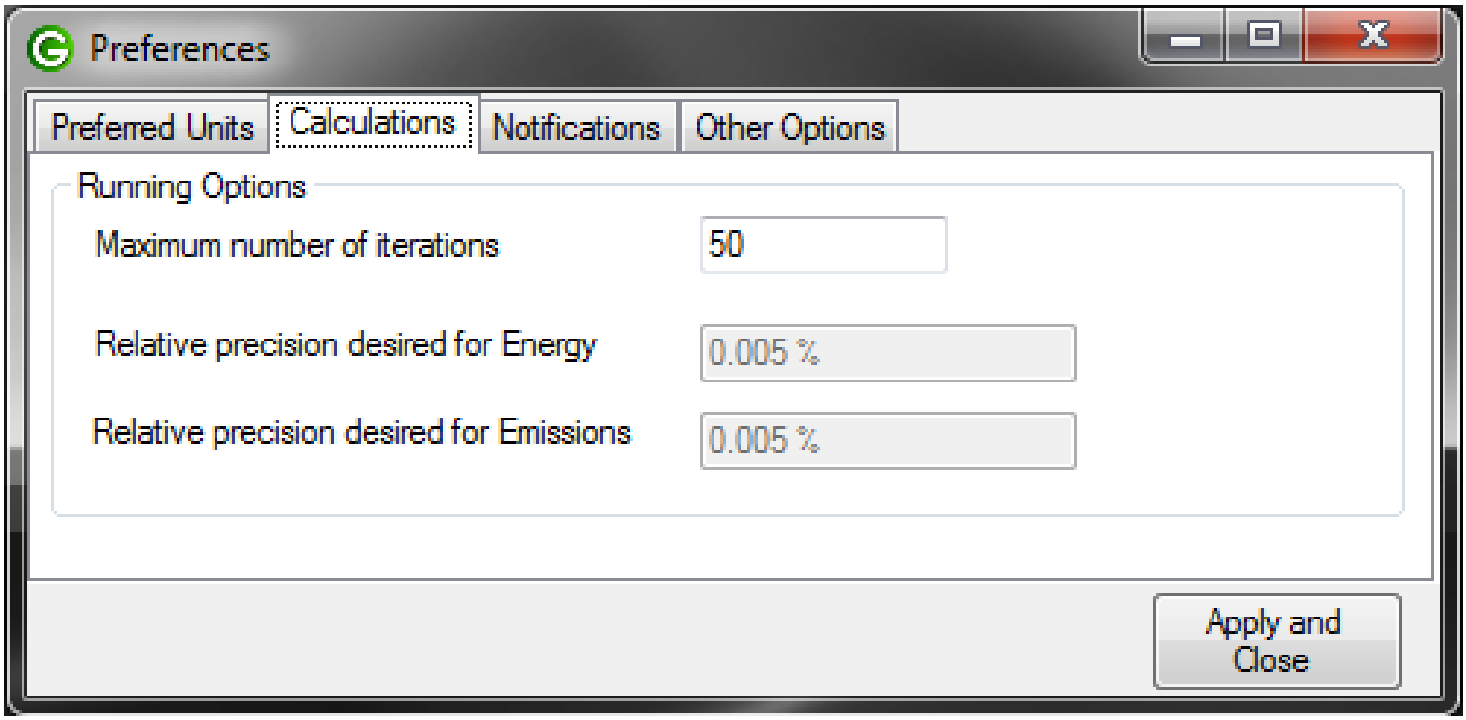


Figure 95: GREET Preferences: Calculations Tab

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 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.

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 96.

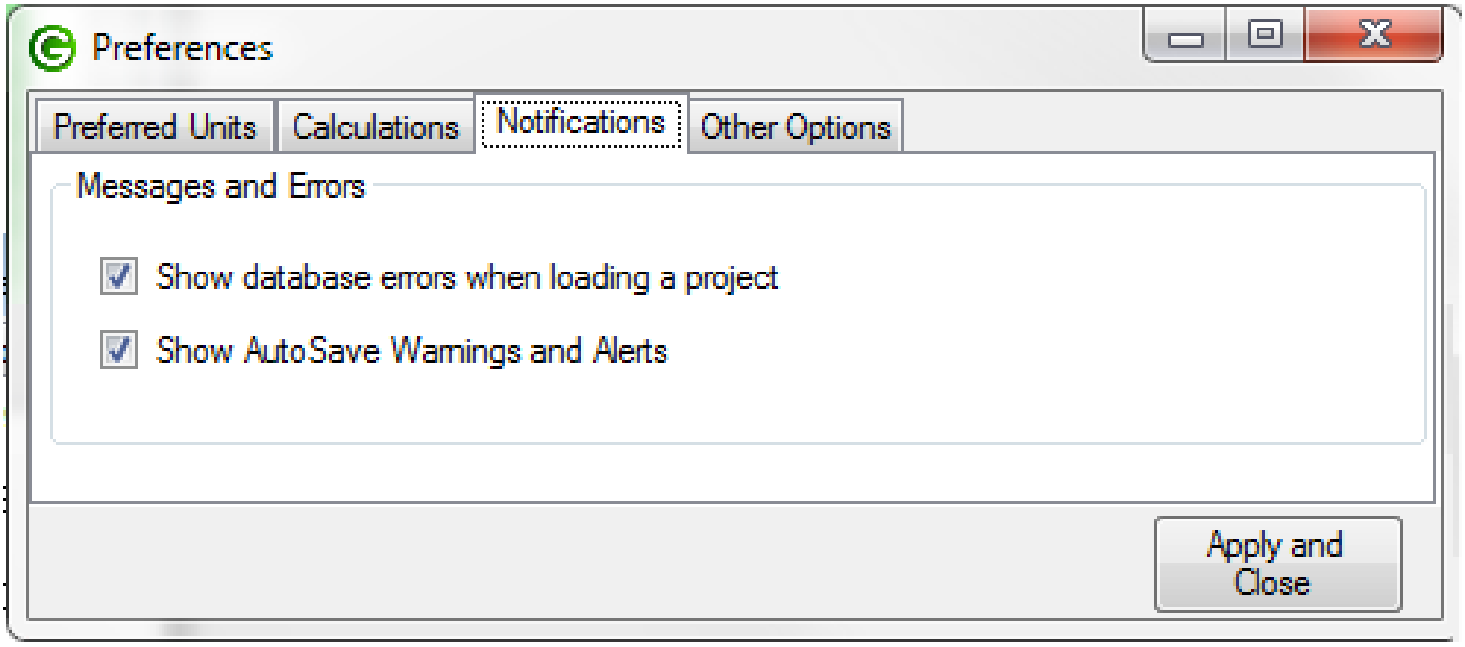


Figure 96: GREET Preferences: Notifications Tab

4.4.4 Other Options

The **Other Options** tab displays two categories of options: **User Interface** features and **Loading/Saving** options. See Figure 97.

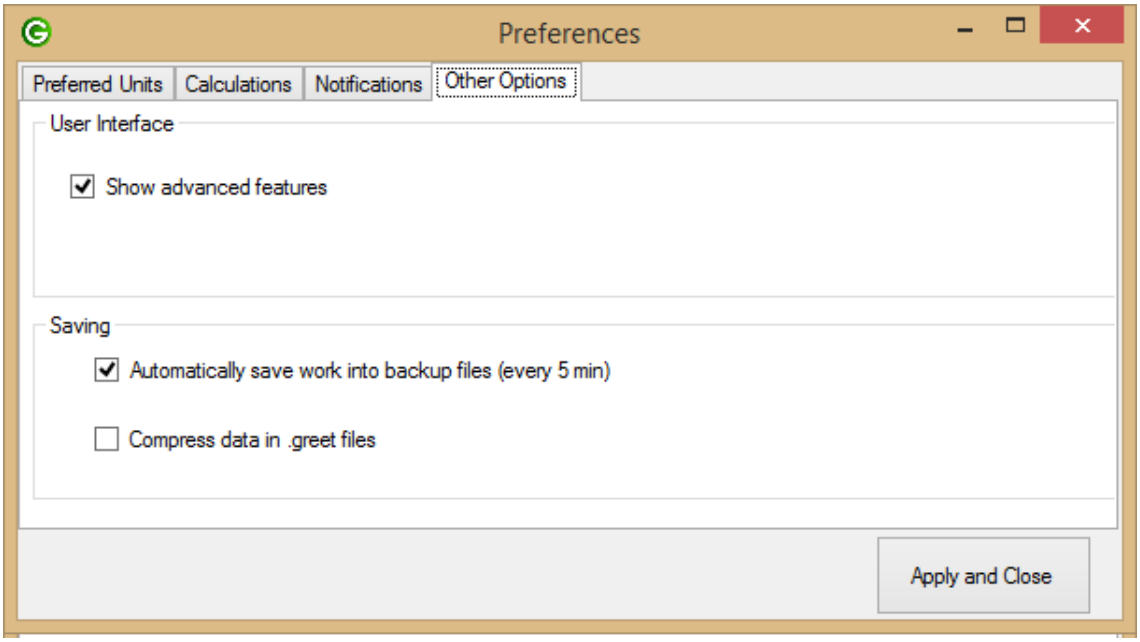


Figure 97: GREET Preferences: Other Options Tab

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.

Default Zoom: For more information about advances features see next chapter.

The Default Zoom allows you to specify whether the pathway should fit to the screen when it is opened or if a specific percentage should be used. Only values in the drop-down list can be selected as the zoom percentage.

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.

4.5 Advanced Features

When the "Advanced Features" option is checked in the General Settings under the Other Options tab, some extra information and features will be available though the user interface.

4.5.1 ID displayed

The ID of Pathways, Processes, Mixes and other entities will be displayed in the editor. For example when you open the process editor, you'll see the ID of that process in the database alongside with it's name.

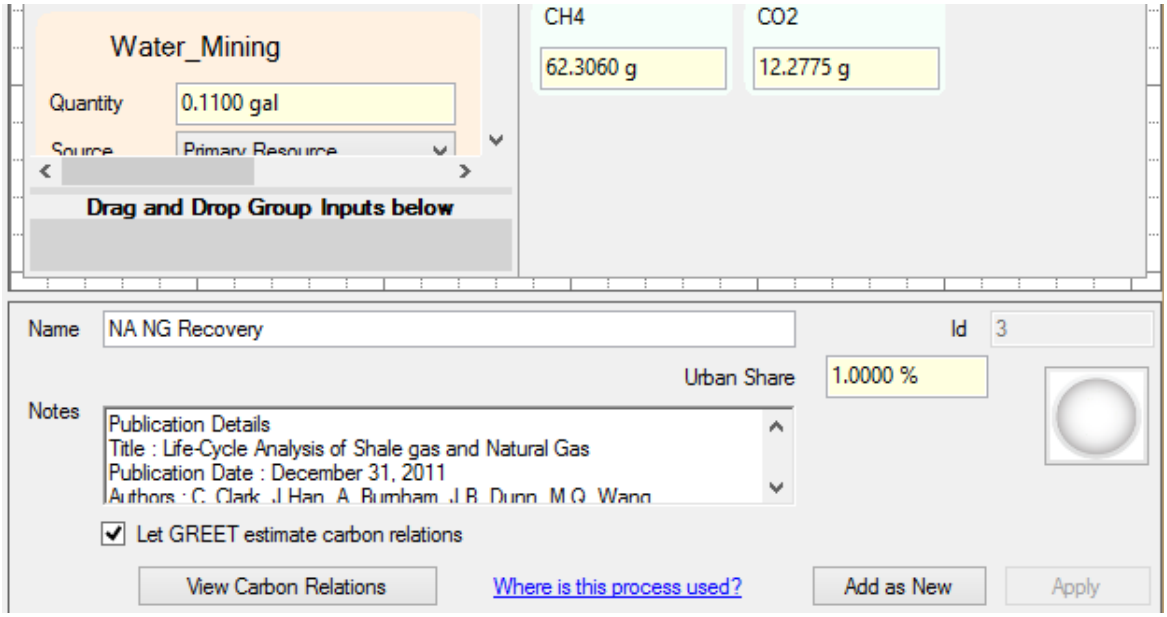


Figure 98: The ID of the process being edited is displayed to the right of it's name

When looking for entities in the data editor, the selection window will also allow you to select entities by name or by ID.

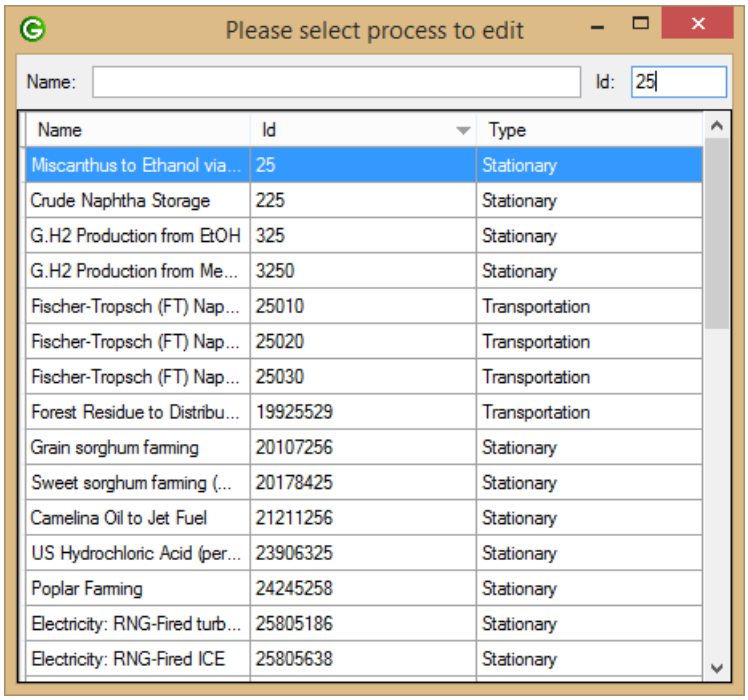


Figure 99: A process can be selected by looking it up by ID when entering an ID number to the right of the name search box

4.5.2 Special features

In the Data Editors pane, a new menu is visible when the Advanced features option is checked. That menu offers you access to the Database Cleaner, Statistics about the database and a tool to check for consistency in the database.

Clean Database This tool allows you to remove from the database extra entities that are never referenced or used anywhere. For example a process was created a long time ago, but is never used anywhere and is now obsolete: It can be discarded in the database.

Another utility of that tool is that it can help you to speed up the calculations. For example, if stochastic simulations are performed it can be very slow to run the simulations on all pathways. Furthermore if the interest on the study is limited to a few pathways only, the other hundreds that are irrelevant to that study can be discarded. Thus calculations are going to run on a fewer number of pathways and results will be obtained faster.

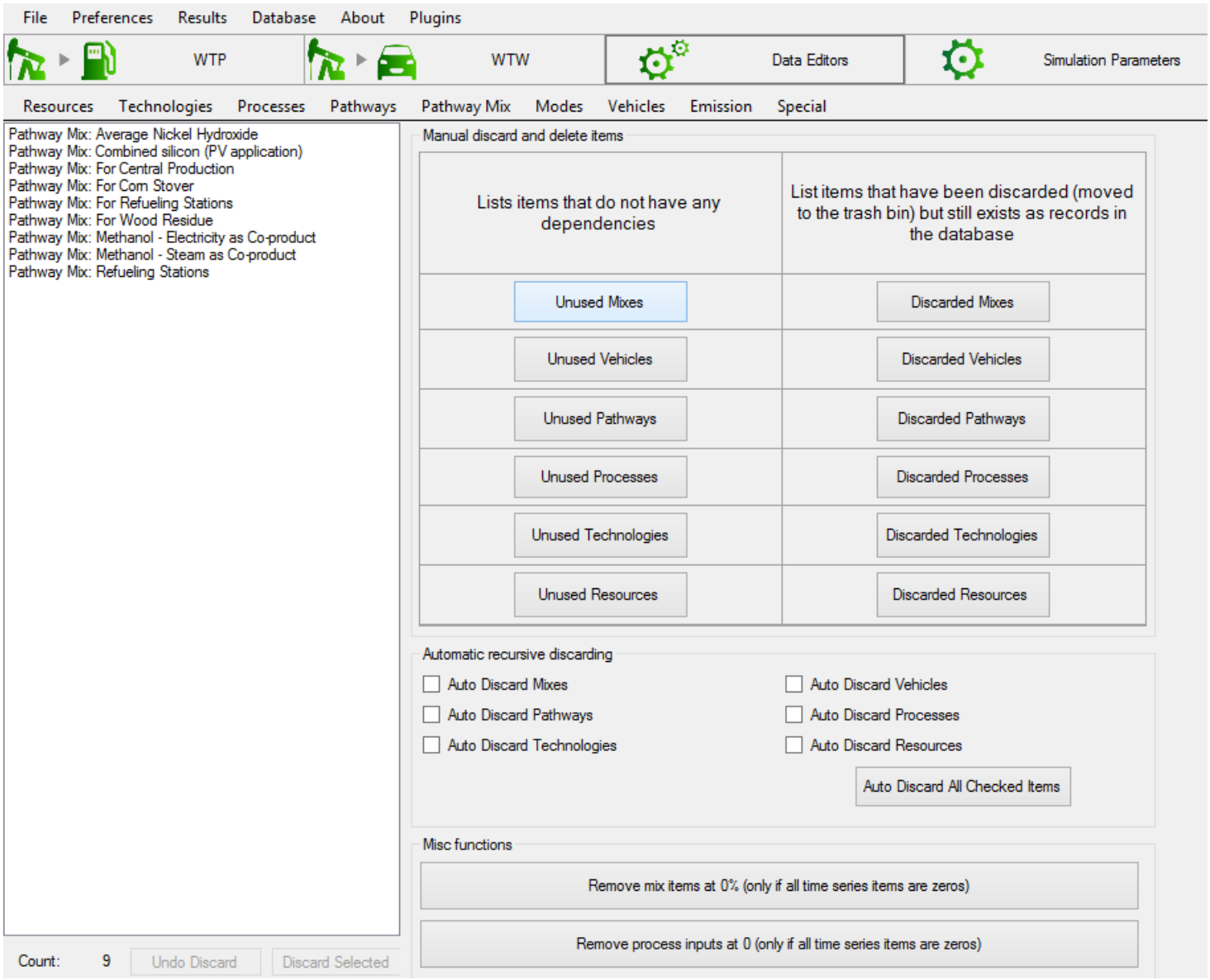


Figure 100: The database cleaning tool offer multiple options for discarding unused entities.

Note: The items that are discarded are "flagged" as discarded, but are still in the database and can be recovered. Note 2: Items can be then deleted and removed completely from the database, but we do not suggest you to do that because the database merging will have trouble discerning between which processes you deleted and which were added in the update.

Stats Clicking the menu "Special -¿ Stats" will show you some information about the database. More importantly it will show you who was the last person making changes to the database and which entities where modified. This could help

keeping track on the evolution of the database content.

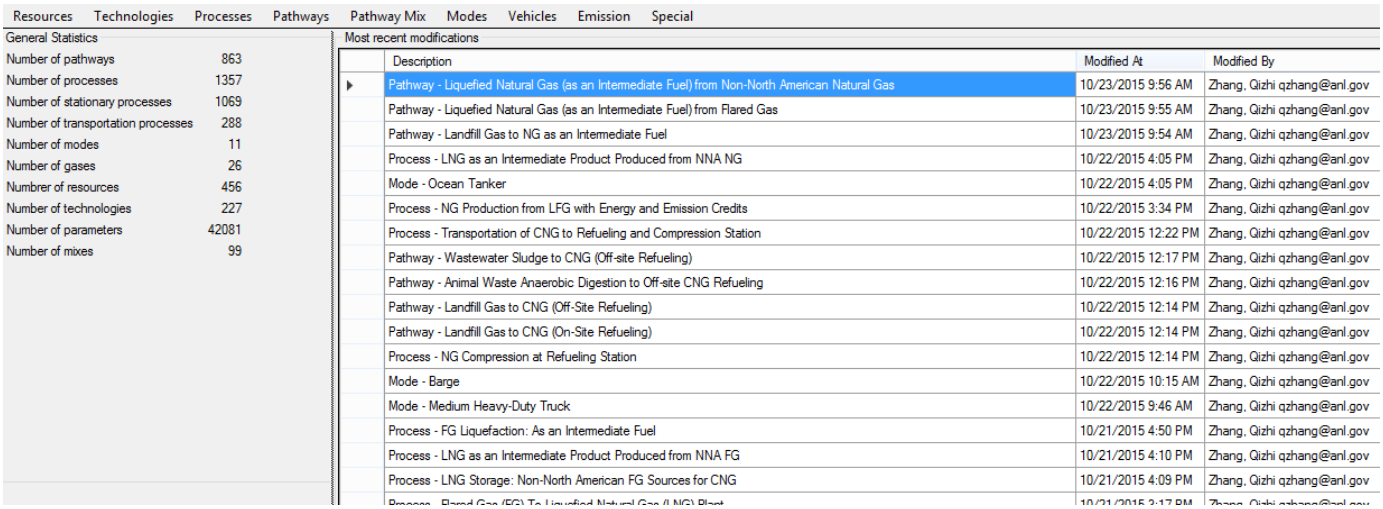


Figure 101: Statistics for the database and history of changes.

Note: The history of changes only stores time and user, it does not store what actually has been changed. If you wish to track the changes in the data file, we suggest to use Git or SVN and track changes in the file Default.greet

4.6 Import from CSV

Starting GREET 2016 it is now possible to import many resources and flows at once using CSV files. This is very handy when you need to import large amounts of data or build complex pathways from unit processes flow data.

4.6.1 Import Resources from CSV

In order to import resources from CSV, prepare a file that contains one resource per row. The header for the first line must be:

"NAME,AltNames,Notes,Families,Density,Density U,LHV,LHV U,HHV,HHV U,MarketVal,MarketVal U,Cratio(Mass),Sratio(Mass),IsPrimary,MemberGroups,OptionalID"

Then each row represent a resource that needs to be inserted in the database.

When resources are created, they need to have at minimum a unique name. The other properties will be initialized to zero or null. The ID of the resource will be automatically picked but it can also be chosen by the user in the last column (In case of conflicts a new ID will be generated).

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q
1	NAME	AltName	Notes	Families	Density	Density	LHV	LHV U	HHV	HHV U	MarketV	MarketV	Cratio(M	Sratio(M	IsPrimar	Member	OptionalI
2	Resource A	resA;aaa															
3	Resource B		some notes														
4	Resource C			cees													
5	Resource D				1	kg/m^3											
6	Resource D						1	MJ/ft^3									
7	Resource E								1000	J/ton							
8	Resource F										1	$/kg					
9	Resource G												0.1				
10	Resource H													0.2			
11	Resource J														TRUE		
12	Resource K															Renewable;Bioma	
13	Resource L															5;6	
14	Resource M																1111

Figure 102: Representation of the CSV file loaded in GREET

The data represented above can be saved in the .CSV format. The separator must be a comma ',' the values must use the dot '.' for decimal point. When multiple values can be defined in a cell, for example alternative names or memberships a semi-column ';' character must be used to separate these values in the cell.

In GREET.net, from the Data Editors tab, click on the Resources sub-menu and then select

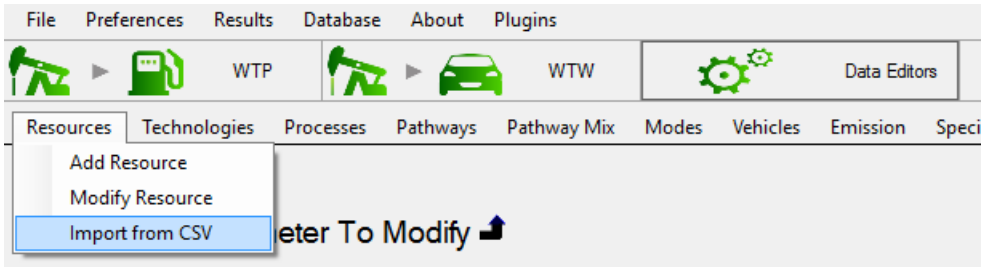


Figure 103: Menu to access the import from CSV feature

Select the CSV file to be loaded:

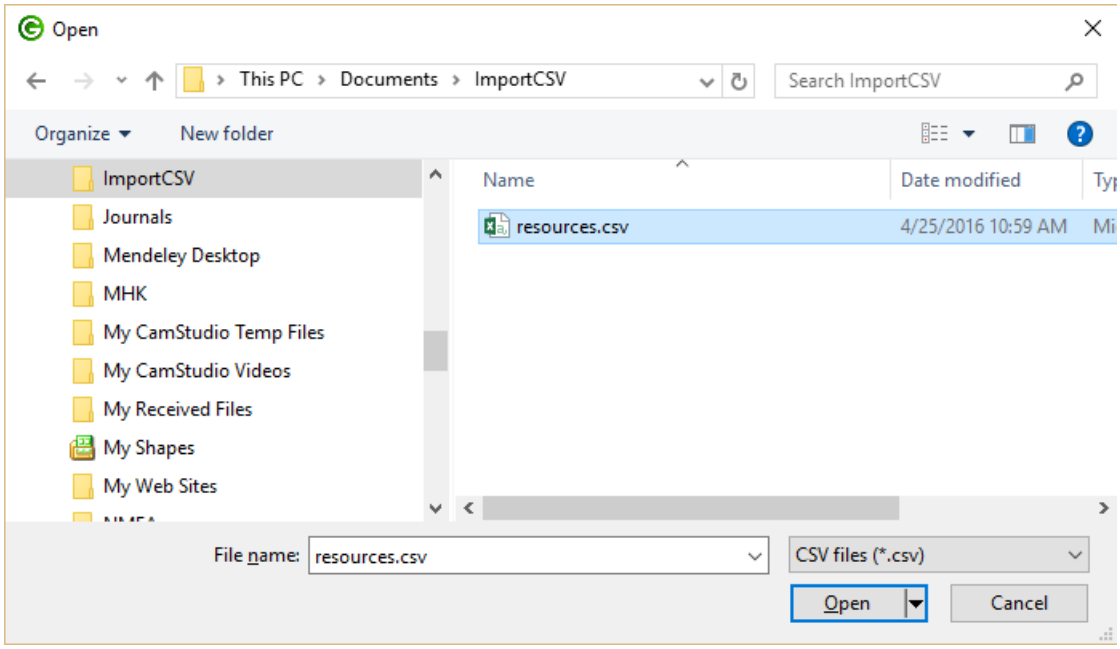


Figure 104: Choosing the CSV file to be imported in GREET

All the resources defined in the CSV file should have been imported in the database. They can now be used in the construction of pathways and processes.

4.6.2 Import Technologies from CSV

In order to import technologies from CSV, prepare a file that contains one list of emission factors per row for a year for a technology. The header must start with the following string: "Name,OptionalId,Notes,Resource,Year,Unit"

Then it must be followed by comma separated gases names. See the example below:

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	Name	Optional	Notes	Resource	Year	Unit	VOC	CO	NOx	PM10	PM2.5	SOx	CH4	N2O	CO2
2	Tractor - 2 Wheel		Diesel Fo	29	1990	g/mmBt	87.7	213.3	2154.2	52.9	47.6	auto	1.2	2	auto
3	Tractor - 2 Wheel		Diesel Fo	29	2000	g/mmBt	86.8	211.2	2132.7	52.4	47.1	auto	1.2	2	auto
4	Tractor - 2 Wheel		Diesel Fo	29	2010	g/mmBt	86	209.1	2111.3	51.8	46.7	auto	1.2	2	auto
5	Tractor - 2 Wheel		Diesel Fo	29	2015	g/mmBt	85.1	207	2090.2	51.3	46.2	auto	1.2	1.9	auto
6	Tractor - 2 Wheel		Diesel Fo	29	1990	g/mmBt	87.7	213.3	2154.2	52.9	47.6	auto	1.2	2	auto
7	Tractor - 2 Wheel		Diesel Fo	29	2000	g/mmBt	86.8	211.2	2132.7	52.4	47.1	auto	1.2	2	auto
8	Tractor - 2 Wheel		Diesel Fo	29	2010	g/mmBt	86	209.1	2111.3	51.8	46.7	auto	1.2	2	auto
9	Tractor - 2 Wheel		Diesel Fo	29	2015	g/mmBt	85.1	207	2090.2	51.3	46.2	auto	1.2	1.9	auto
10	Tractor - 4 Wheel		Diesel Fo	29	1990	g/mmBt	95.2	235.6	2317	57.6	56.6	auto	2.2	2	auto
11	Tractor - 4 Wheel		Diesel Fo	29	2000	g/mmBt	94.2	233.2	2293.8	57	56	auto	2.2	2	auto
12	Tractor - 4 Wheel		Diesel Fo	29	2010	g/mmBt	93.3	230.9	2270.9	56.5	55.5	auto	2.2	2	auto
13	Tractor - 4 Wheel		Diesel Fo	29	2015	g/mmBt	92.4	228.6	2248.2	55.9	54.9	auto	2.1	2	auto

Figure 105: Choosing the CSV file to be imported in GREET

The data represented above can be saved in the .CSV format. The separator must be a comma ',' the values must use the dot '.' for decimal point.

In GREET.net, from the Data Editors tab, click on the Technologies sub-menu and then select

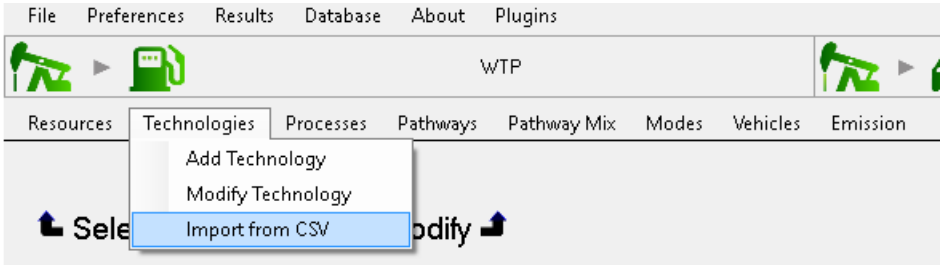


Figure 106: Menu to access the import from CSV feature

Select the CSV file to be loaded:

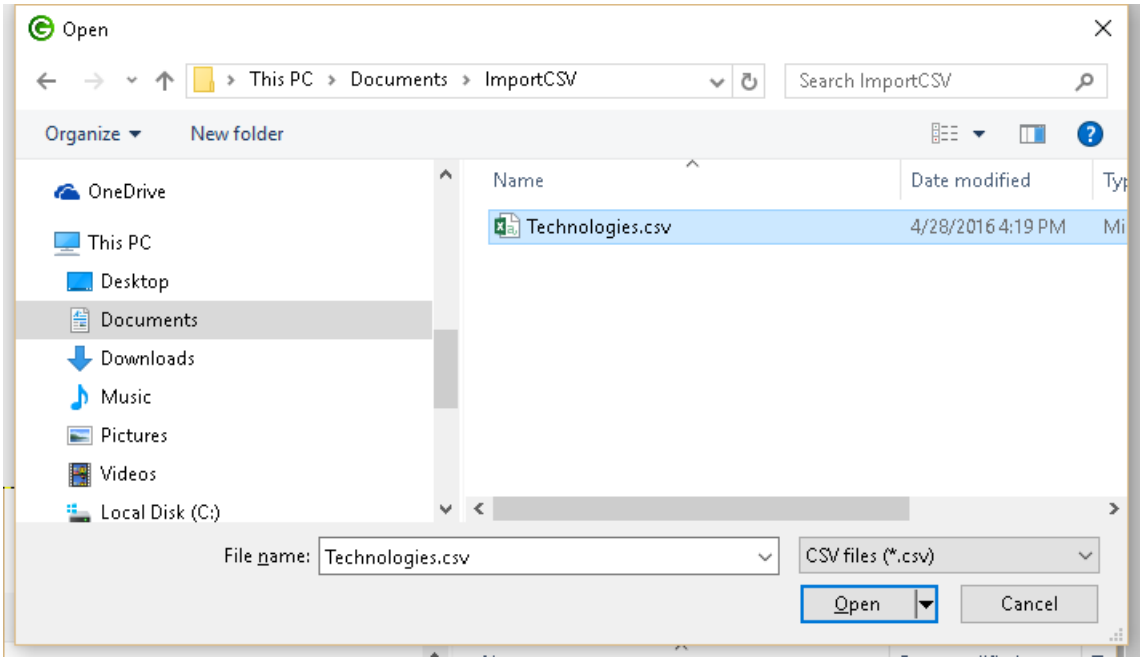


Figure 107: Choosing the CSV file to be imported in GREET

All the technologies defined in the CSV file should have been imported in the database. They can now be used in the construction of processes.

4.6.3 Import Processes from CSV

In order to import processes from CSV, prepare a file that contains the flows for all the unit processes you want to create. The header for the first line must be:

"UnitProcess,Pathway,Stream,Quantiy,Unit,CoProdType,AllocationType,DisplacedPorM, DisplacedPorMId,DisplacementPorMShare,TechId,TechShare"

Then each row represent a flow for a unit process. Negative flows are inputs, positive flows are outputs. In order to create a unit process, you can add as many inputs and outputs flows. They can be resources/products or emissions. If multiple products are produced by the unit processes, the user can choose which allocation/displacement method should be used for these products.

It is also possible to automatically assemble the unit processes in a pathway. The software will match the inputs and outputs by resource name and quantities. It will automatically connect resources which have the same name and quantities. For others it will create "pools" of material that can be re-used to connect to other processes.

If the "Pathway" column is left blank, the processes will be added as individual unit processes in the database but no pathway will be created

Finally technologies can be added to the inputs by specifying the technology ID and share desired in the "TechId" and "TechShare" columns.

	A	B	C	D	E	F	G	H	I	J	K	L
1	UnitProce	Pathway	Stream	Quantiy	Unit	CoProdTy	Allocation	Displaced	Displaced	Displacem	TechId	TechShare
2	ProcA	Path 1	a	-1.2	J							
3	ProcA	Path 1	e	-0.1	J						100	1
4	ProcA	Path 1	b	1	J							
5	ProcA	Path 1	CO2	0.1	kg							
6	ProcB	Path 1	b	-1	J							
7	ProcB	Path 1	g	0.3	J	displacement		pathway	1	1		
8	ProcB	Path 1	d	1.5	J							
9	ProcC	Path 1	e	-0.1	J							
10	ProcC	Path 1	a	-1.5	J							
11	ProcC	Path 1	c	1	J							
12	ProcD	Path 1	c	-3	J							
13	ProcD	Path 1	f	1.5	J							
14	ProcD	Path 1	e	1	J	allocation	energy					
15	ProcD	Path 1	Water	-2	kg							
16	ProcE	Path 2	a	-1.1	kg							
17	ProcE	Path 2	d	1	kg							
18	ProcF	Path 2	b	-2.1	kg							
19	ProcF	Path 2	d	2	kg							
20	ProcG	Path 2	c	-3.1	kg							
21	ProcG	Path 2	d	3	kg							
22	ProcH	Path 2	d	-1.4	kg							
23	ProcH	Path 2	e	1	kg							
24	ProcI		a	1	m^3							
25	ProcI		e	1	m^3							

Figure 108: Representation of the CSV file loaded in GREET

The data represented above can be saved in the .CSV format. The separator must be a comma ',' the values must use the dot '.' for decimal point. When multiple values can be defined in a cell, for example displaced products, a semi-column ';' character must be used to separate these values in the cell.

In GREET.net, from the Data Editors tab, click on the Resources sub-menu and then select

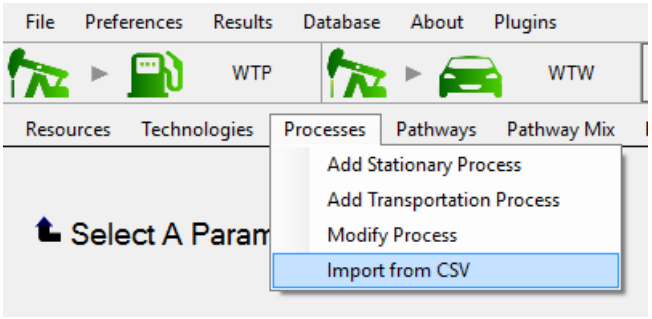


Figure 109: Menu to access the import from CSV feature

Select the CSV file to be loaded:

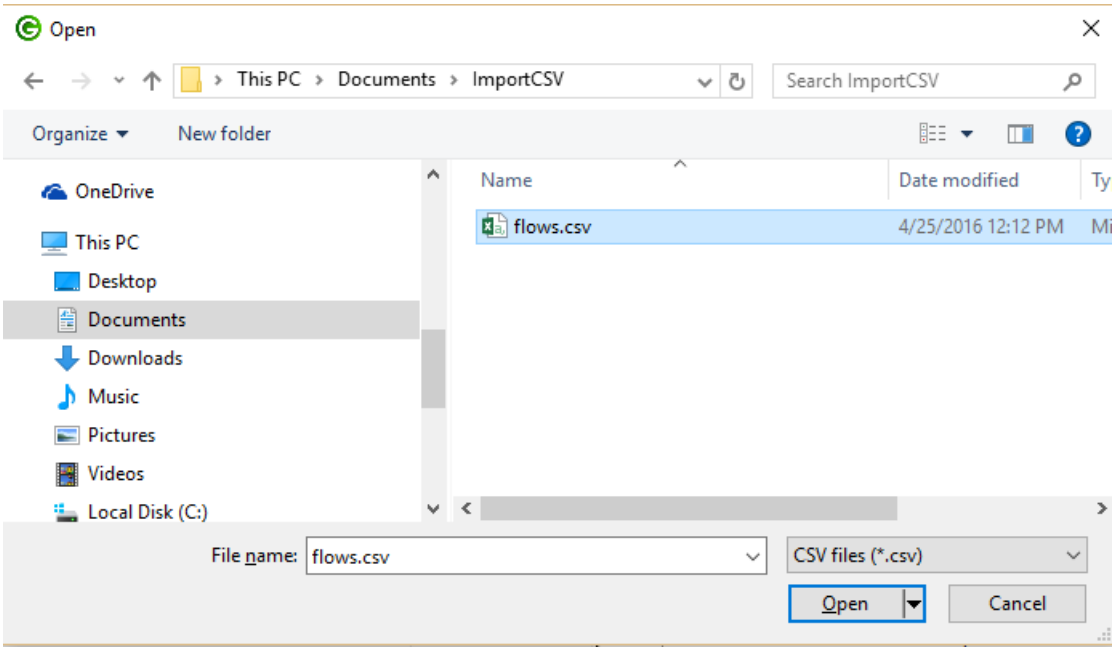


Figure 110: Choosing the CSV file to be imported in GREET

All the unit processes and flows defined in the CSV file should have been imported in the database. They can now be used in the editors or visualized in the WTP results panel.

5 Plugins

GREET is delivered by default with external modules that communicate with the main software via the GREET API. These modules were added in order to provide specific features to GREET. Developers are encouraged to develop their modules for GREET and send requests to greet@anl.gov for API modifications or chat on freenode.net in the room #greetlca. Popular modules can then be advertised on the main website along the default ones developed by Argonne.

5.1 Jet Fuel WTWa

5.1.1 Introduction

Jet fuel WTWa (Well-to-Wake) module was added to include analyses of aviation fuels and aircraft. The WTP (well-to-pump) pathways for aviation fuels (such as Conventional Jet Fuel and Fischer-Tropsch Jet Fuel) have already been implemented in the GREET main model. The Jet Fuel WTWa module allows us to calculate WTWa results for each aircraft type and fuel type combination. Users can also add new aircraft types.

The results are presented in terms of energy and emissions. The low heating value of a fuel is used for the calculations. Energy use is specified in terms of energy intensity, which is the amount of energy consumed per each great-circle unit of distance traveled per kg of freight transported, the default units are

$$\frac{J}{kg \times km}$$

The energy results are divided into several categories. The first category is aircraft operation (PTW) energy use, also called Payload Fuel Energy Intensity (PFEI). The second category is full life cycle energy results (WTWa), that include aircraft operation and energy associated with fuel production. The energy use is broken down by type in the WTWa cycle; energy types include petroleum, fossil, coal, and natural gas.

Currently there are six classes of passenger aircraft (single aisle, small twin aisle, large twin aisle, large quad, regional jet, and business jet), and four classes of freight aircraft (single aisle, small twin aisle, large twin aisle, and large quad). A user can add new types of aircrafts. Each aircraft class was characterized by its average payload, average trip great-circle distance, total flight payload fuel energy consumption, emissions during cruise, and fuel consumption and emissions during a landing and takeoff (LTO) cycle. The LTO and cruise fuel use and emissions for alternative jet fuels were normalized relative

to the baseline petroleum jet fuel's energy use and emissions. We distribute the LTO energy use and emissions over the entire flight by spreading their numerical values over the flight payload and great-circle distance. For mode details on data sources of the input parameters, see [2].

Emissions results are given for each of the individual pollutants, plus GHG and TotalCO2. GHG emissions calculation combines carbon dioxide, methane, and nitrous oxide with their global warming potentials (1, 25, and 298, respectively, based on a 100-year time window). The TotalCO2 emissions is the total amount of CO2 including the equivalents from VOC and CO.

5.1.2 Using the Module

To open the module click on the Jet Fuel WTWa item on the top menu bar

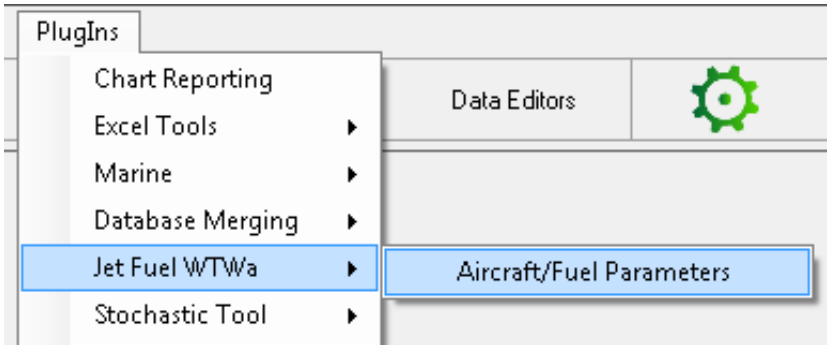


Figure 111: Main menu for the Jet Fuel WTWa module

Once you click **"Aircraft/Fuel Parameters"** you will see the main module window.

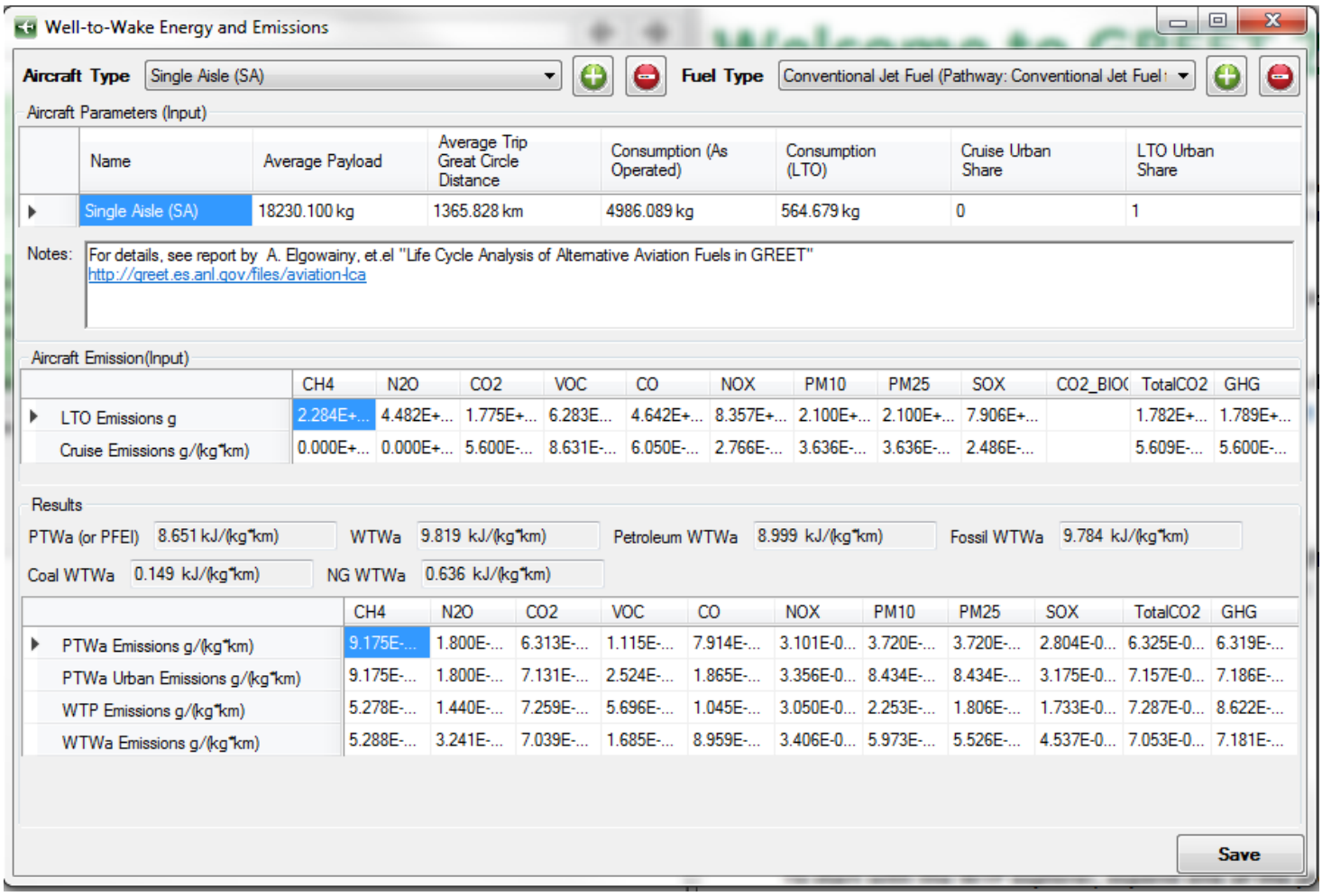


Figure 112: Jet Fuel WTWa module

Exploring results for different Aircraft and Fuel configuration By default the data includes data for some aircraft such as the average payload, distance, energy consumption. It also includes a selection of typical fuels that are used in these aircraft.

Selecting a different aircraft and/or fuel will update the results part. The upstream for the fuels are selected from the GREET pathways and added to the energy and emissions associated with the operation of the aircraft itself.

Editing aircraft parameters When an aircraft type is selected, parameters can be modified in the two tables marked as **"(Input)"**. They represents the typical emissions for landing and takeoff as well as emissions in cruise mode. Physical characteristics of the plane and parameters for the trip can be changed in the **"Aircraft Parameters** table.

For all airplanes, the note indicates the source of the data and/or publication supporting the values used in the simulations.

When parameters are changed, calculations will be executed and the results table will be updated automatically as soon as the an "Enter" is pressed.

In order to save your changes to the database file (.greet) extension, click the **"Save"** button on the bottom right below the results table.

Adding a new Aircraft In order to add a new aircraft, click the + button on the right of the Aircraft Type selection.

The new plane added will be selected with its parameters displayed. At this point the user has to fill out all the parameters for that plane. Both tables **Aircraft Parameters(Input)** and **Aircraft Emissions(Input)** needs to be completed before calculations can be started.

After adding a new aircraft it is important to click the **"Save"** button on the bottom right below the results table in order to save your changes to the database file (.greet)

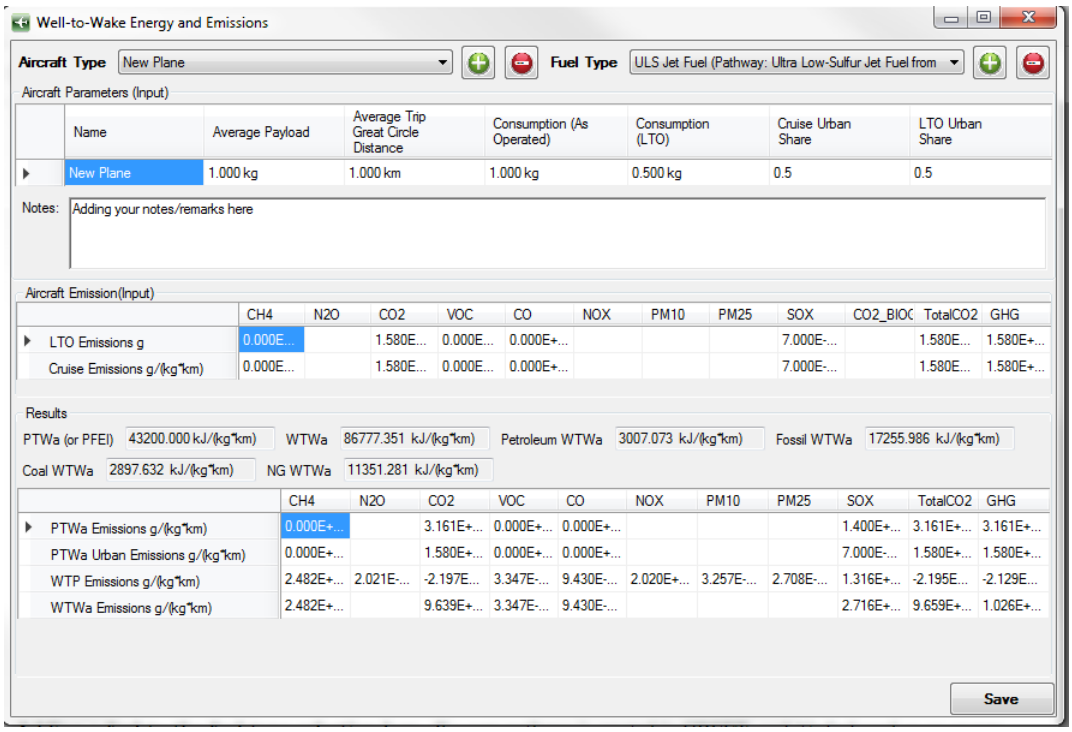


Figure 113: A new plane added to the dataset, all parameters are set to default

Adding a fuel to the fuel type selection box If a new pathway is created in GREET, and its fuel can be used as an energy source for an aircraft, it needs to be added to the list of Jet Fuel available.

By default all the Jet Fuels available in the current database are listed.

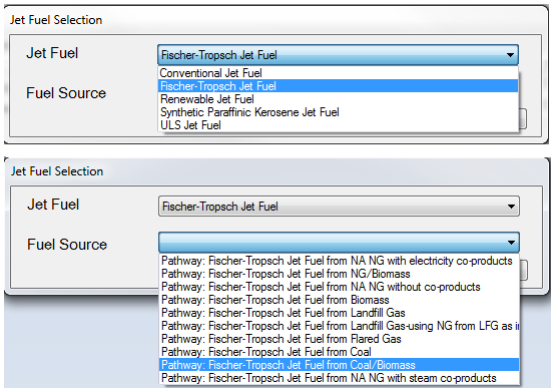


Figure 114: Adding an existing pathway mix to the list of available fuels

The figure above shows the selection of Fischer-Tropsch Jet Fuel produced from Coal/Biomass, as a possible selection for aircraft. This is hypothetical and any type of energy source can be selected as seen in the following figure:

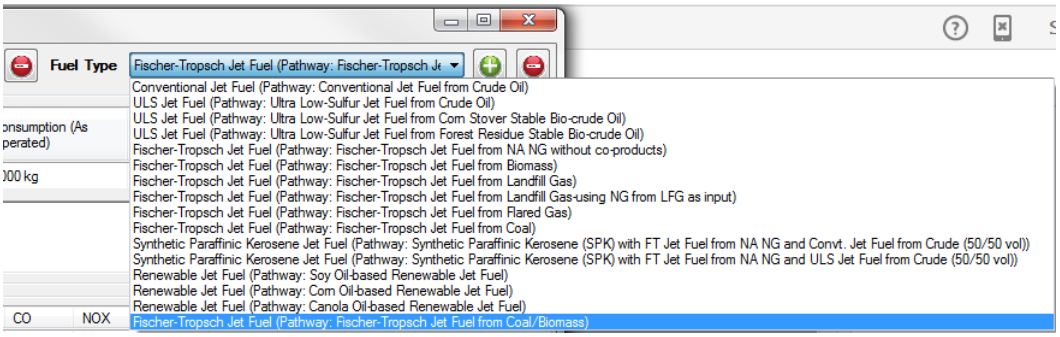


Figure 115: Selecting added mix for the current aircraft

Note: Any type of fuel can be selected with any aircraft type. The Well to Pump results (WTP calculated from GREET) is going to be used in association with the selected aircraft typical energy consumption to calculate the results.

Note2: When changing attributes of an aircraft, check that all of the parameter dimensions are properly defined, otherwise results would not be calculated properly.

5.2 Data Merging

The Data Merge tool allows the merging of data from one XML data file to another XML data file.

Step 1. Selecting the **"Database Merging"** then **"Open Merging Tool"** from the main menu. See Figure: 116

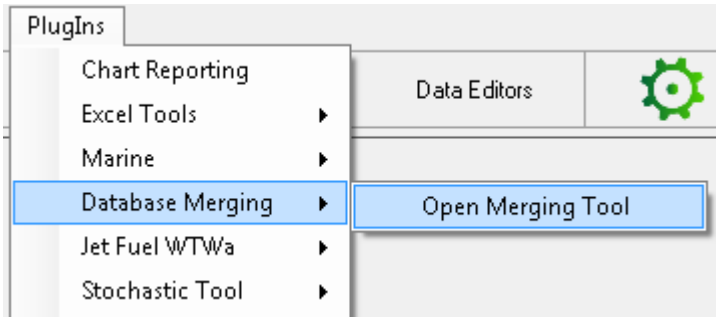


Figure 116: Select a GREET data file to Merging From

Step 2. Select the **Merge From** data file. See Figure: 117

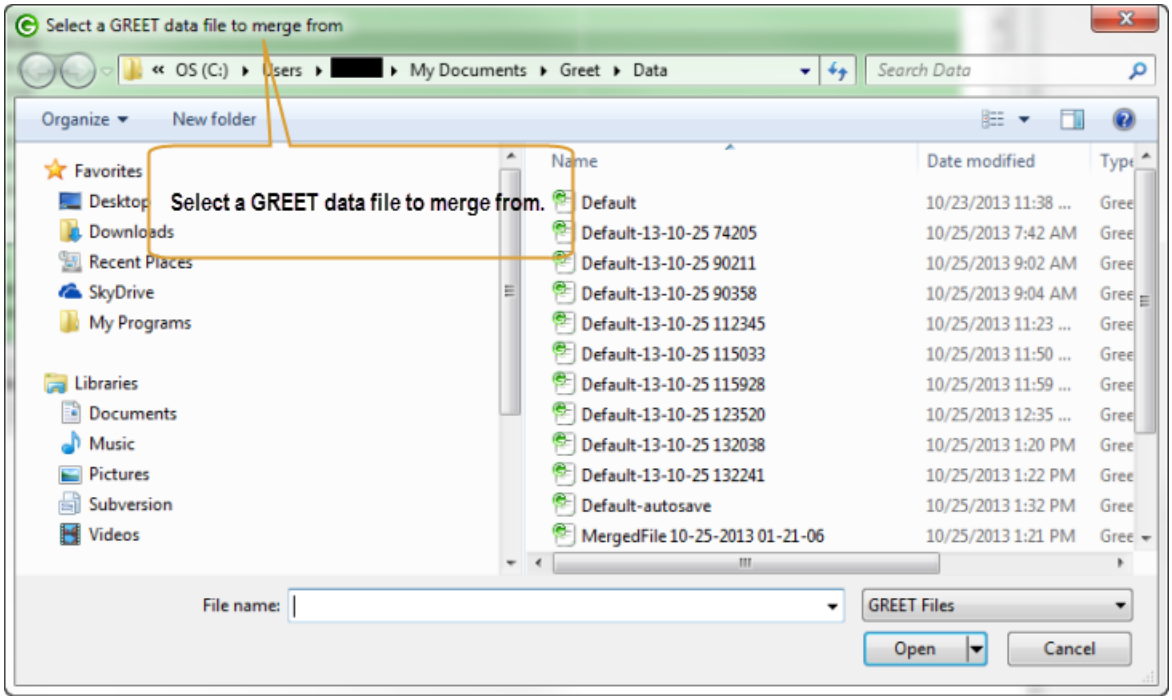


Figure 117: Select the Merge From from the form.

You now have all the data you need to merge data. But, to refine the merge to merge parts of the data you will need a broader understanding of the interface.

Review of "The Merge Data Interface".

The column on the left displays GREETs higher level entity objects such as Resources, Mixes, Pathways, Processes, Simulation Parameters, Images, Technologies, and more. All of these entity objects store data of some type. All entity objects are displayed in check box tree format with the root of each entity objects all having check boxes. Checked entity objects will merge unchecked entity objects will not. Some entity objects allow you drill deeper down their tree, GREET calls these entity objects "Dependencies". The Merge Interface also allows for checking or unchecking of Dependencies for more control on what specific data is merged.

The middle column displays data, in XML format, that comes from the XML data file that you are intending to merge data from. The data displayed in this colume is read only data and informational only. The data shown is the data that is different from the **To XML File** right colume as the highlighting in Figure: 118 shows.

The column on the right displays data, in XML format, that comes from the XML data file that you are intending to merge data to. The data displayed in this colume is read only data and informational only. The data shown is the data that is different from the **From XML File**. middle colume as the highlighting in Figure: 118 shows.

Step 3. Made your refinement in left colume then click the **Merge Selected** button in the lower right corner of the Merge Data form.

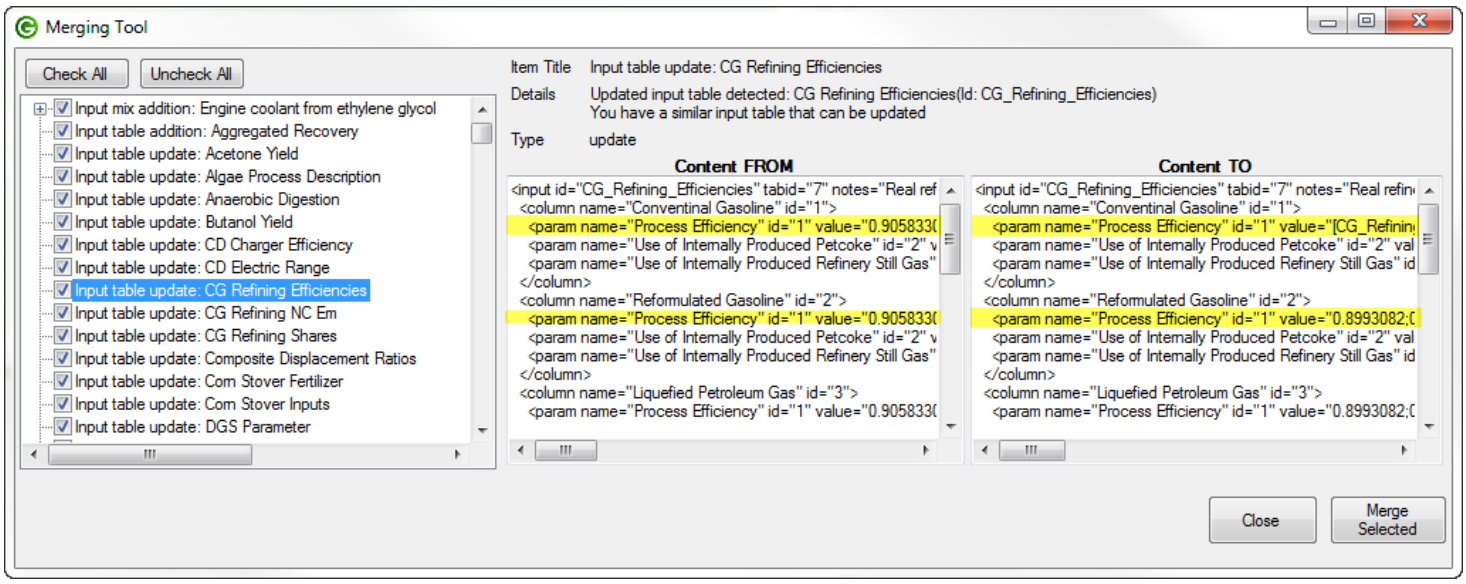


Figure 118: Merge data interface.

Step 4. Next, the Merge Tool will check for XML data file updates posted on GREET online and a form will appear asking you if you would like to merge that data prior to merging data from your "Content FROM" data file. Select "Yes" or "No".

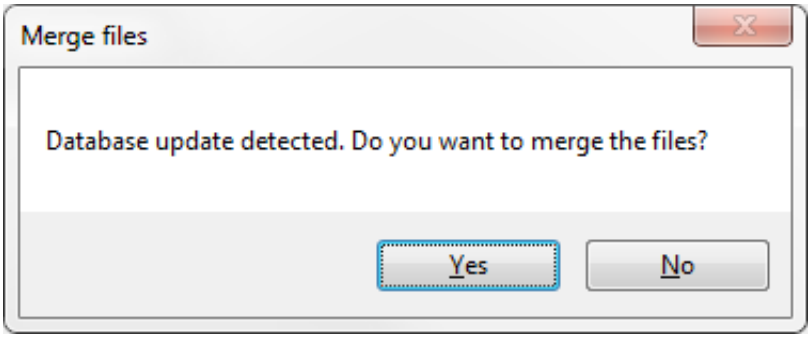


Figure 119: Merge data from GREET online.

5.2.1 A Merge Where No Differences Remain

Figure: 120 shows a merge with all three columns completely blank meaning no differences remain. The data in the "Content TO" XML data file is the same as data in the "Content From" data file.

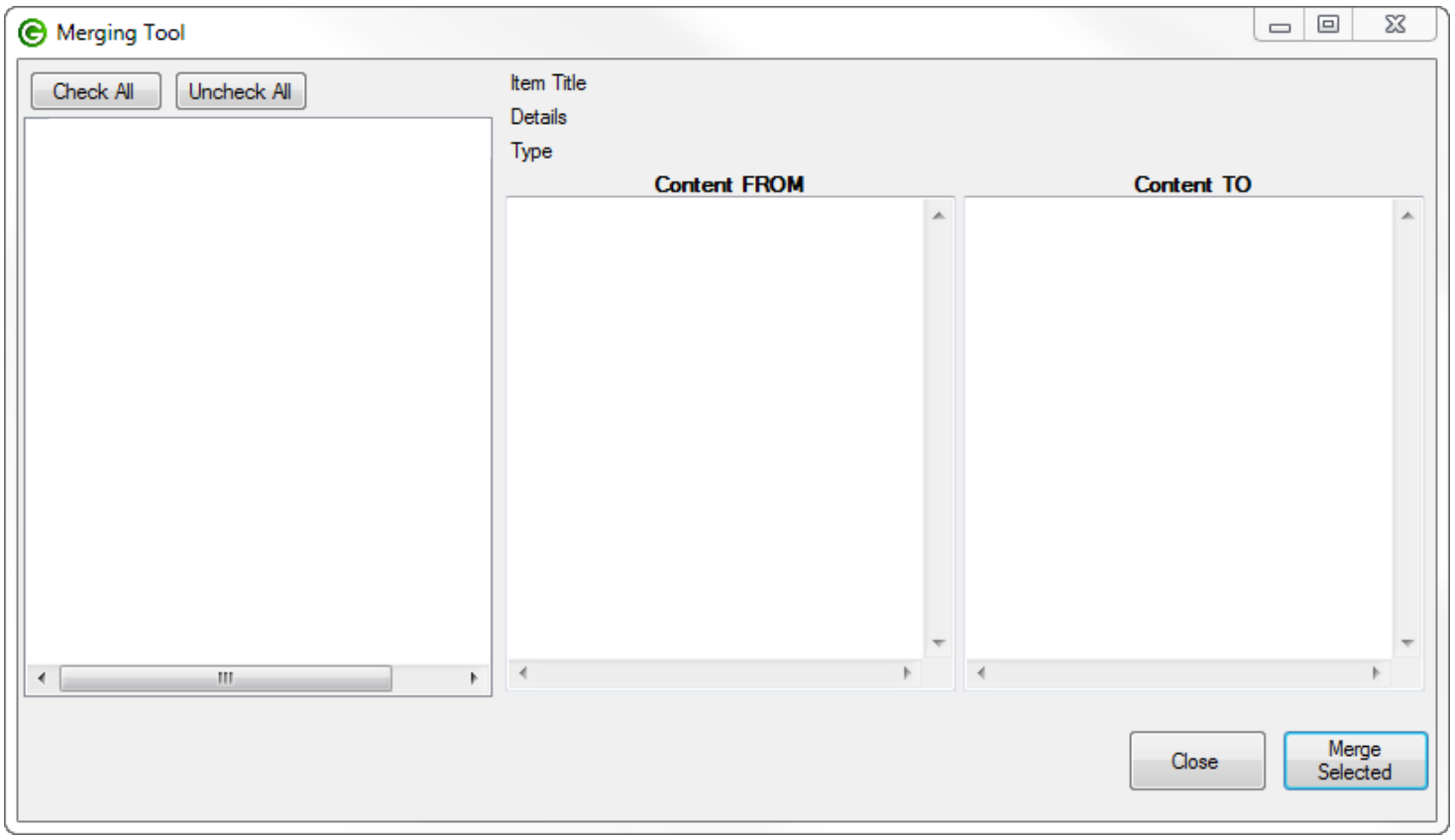


Figure 120: Everything Blank Showing No Differences.

5.2.2 A Merge Where Differences Remain

Figure: 121 shows a merge with differences remaining. What took place with this merge was the Merge Tool came across a situation where the Merge Tool couldn't programmatically resolve if an entity object coming from the "Content FROM" data file should overwrite or change the data in the "Content TO" data file risking unintended consequences. In this situation no entity objects in the "Content TO" data file were changed. But instead the entity object coming from the "Content FROM" data file was added, completely intact, to the "Content TO" data file as a new entity object.

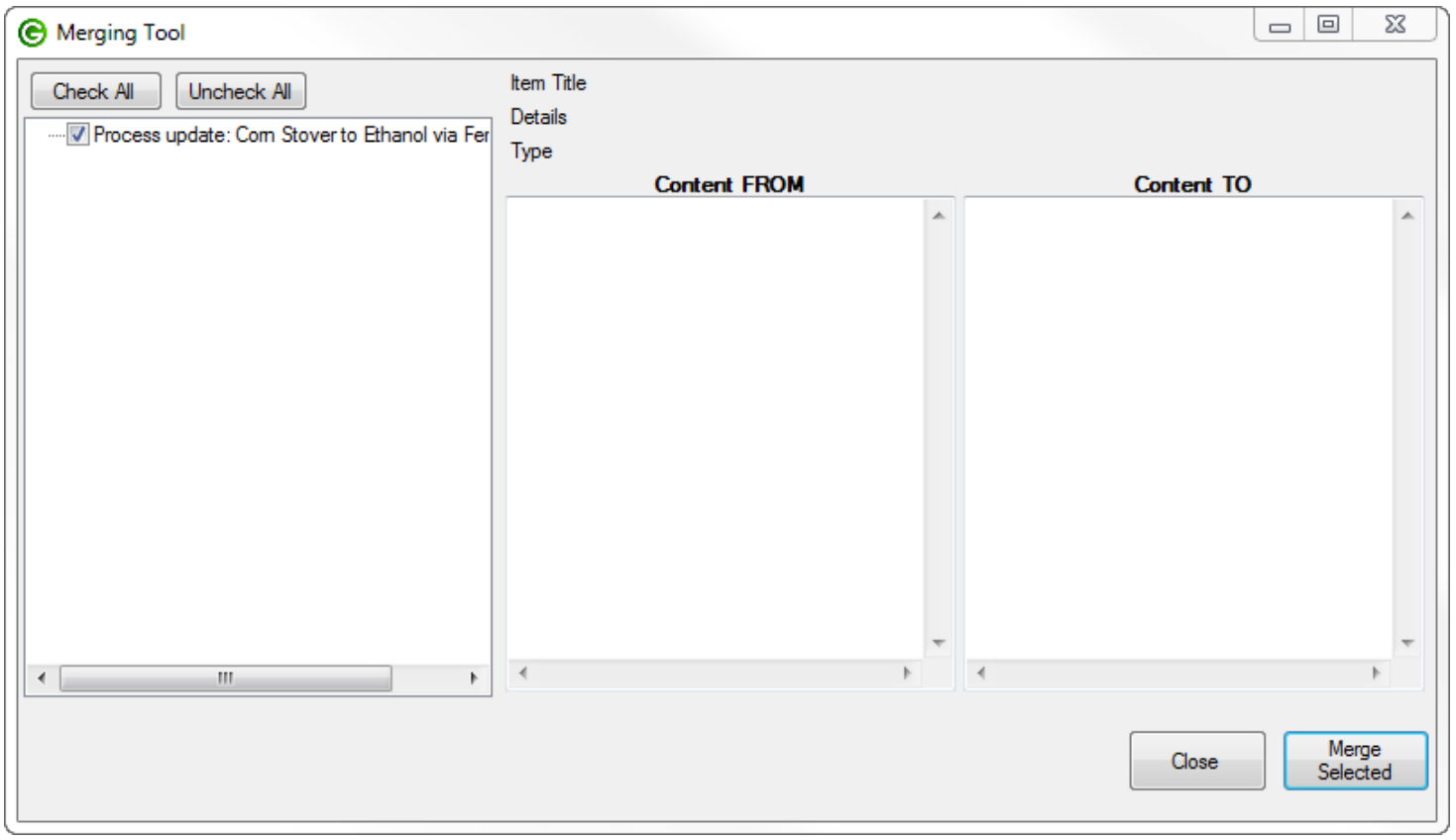


Figure 121: Merging tool main window showing differences between host and update file

5.3 Stochastic Simulations

Stochastic Simulations Tool is a newly added feature of GREET and currently in the early stages of it's development. The **inputs required** for a stochastic simulations are:

- The outputs values in WTP that are available to be Monitored by GREET's Stochastic Simulations Tool. Selecting one or more values to be monitored are covered in **"Selecting a Forecasted Results ("Monitor a Result")"** Section: 5.3.1
- GREET Distributed Parameters, refered to as **"Parameters"**. Any Parameter that is shown in GREET in a yellow box can be distributed. These are usually found in the **"Data Editors"** or **"Simulation Parameter"** (**"Simulation Parameter"** are covered in detail in Section 3.5). One distribution is allowed per parameter. Parameters that have a distribution defined are displayed with a with a blue border around the parameter's displayed value. (Distributions editing are covered in detail in Section 5.3.2)

Results of a Stochastic Simulation run are found by clicking GREET's main menu item **"Stochastic Tool"** then **"Explore Sampled Parameters and Monitoring Results"**.

NOTE: The Stochastic Simulations Tool currently doesn't have an all inclusive place (its own form) for flagging input parameters for inclusion or setting up the details of a distribution. Stochastic Simulations inputs (Parameters, Distributions, and Values being monitored) are flag for inclusion into a Stochastic Simulations from the places they currently reside in GREET. The main menu item "Stochastic Tool" and it's sub-menus will display form that only deal with the running, viewing, and the editing of existing parameters distributions. See Figure: 122.

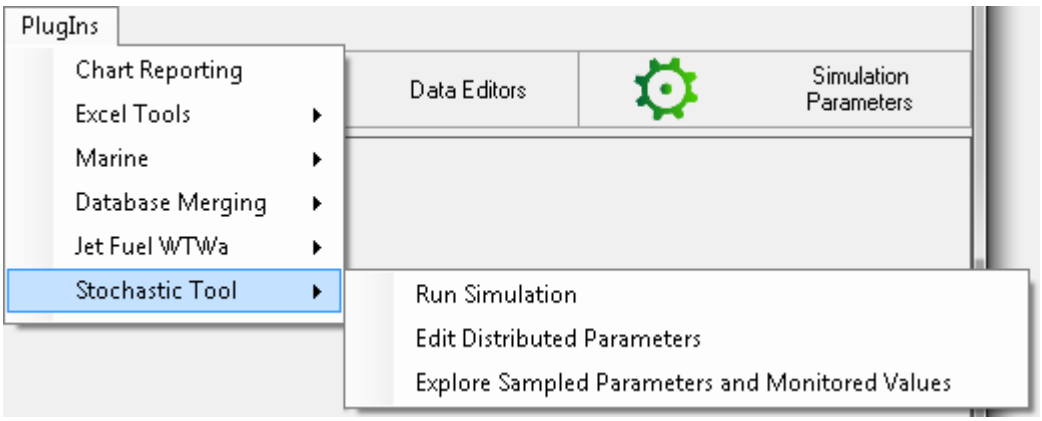


Figure 122: Main Menu for the Stochastic Simulations Module

5.3.1 Selecting a Forecasted Results ("Monitor a Result")

GREET Emission and/or Resource values that are available to be selected to be monitored by the Stochastic Tool are currently only located in "WTP". Click the **"WTP"** button just under GREET's main menu. Select a **"Product"**, then a **"Mix"** or **"Pathway"**, then the **"Produced Resource"** if more than one exits. Lastly, select and right click on an Emission or Resource **"Name"**, not the value, then select **"Monitor this value"** from the menu. See Figure: 123.

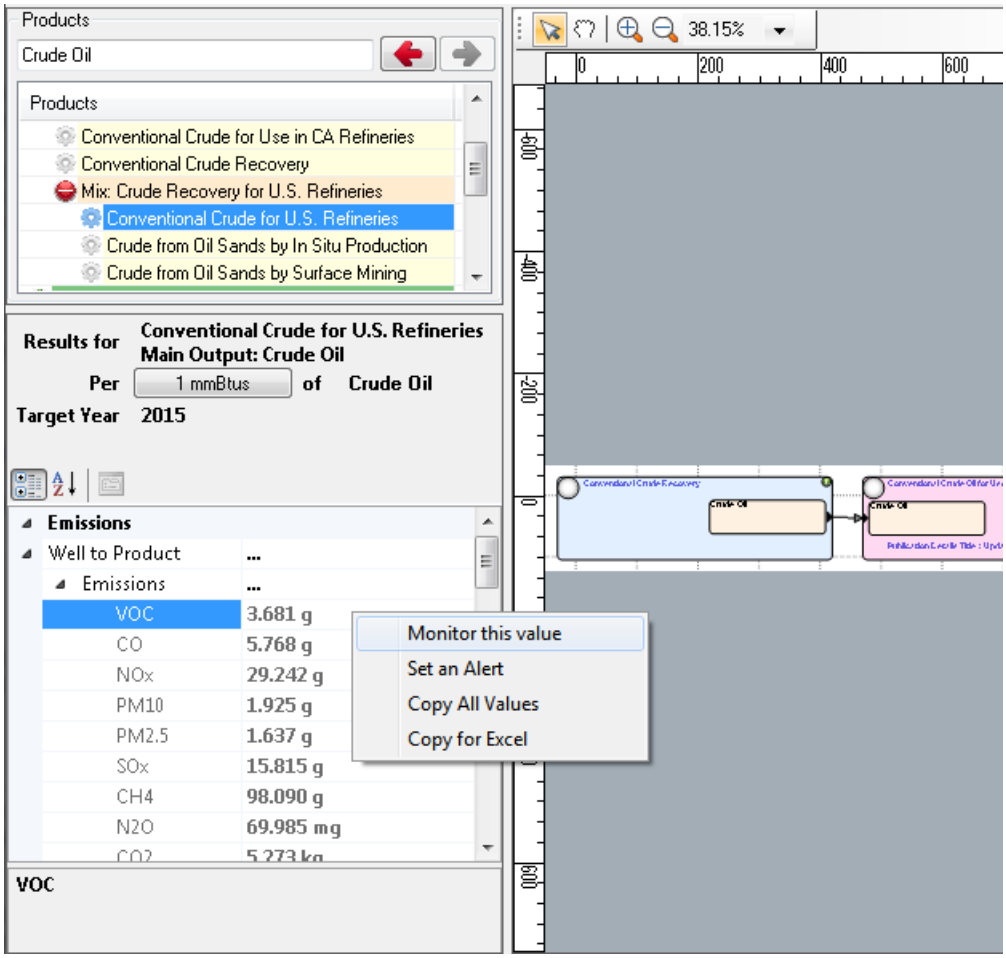


Figure 123: Select "Monitor this value"

5.3.2 Define and Edit Distributions

The GREET default data contains parameters that have default distributions defined for parameters. Most are specified now for stationary process efficiency (Crude oil recovery, natural gas recovery and processing) and resource physical parameters

(Low and high heating values, density, carbon and sulfur ratios).

As stated above, parameters that have a distribution defined are displayed with a with a blue border around the parameter's displayed value. Shown in Figure: 124.

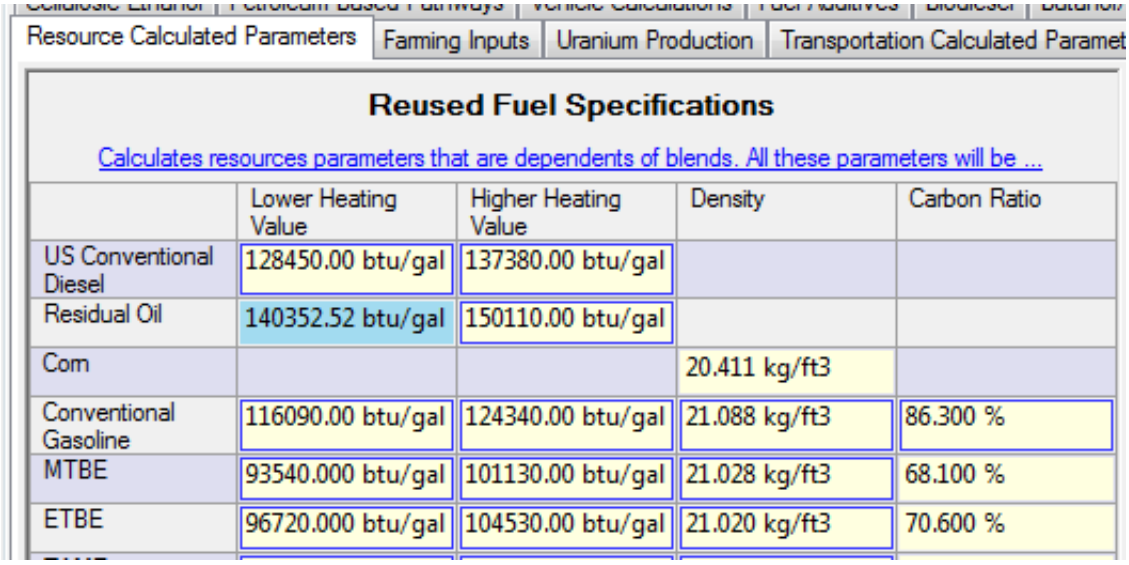


Figure 124: Parameters Showing a Blue Border Indicating that Distributions are Defined

To add a distribution for a parameter, right click on the value, then click on the "Add distribution" menu item. Shown in Figure: 125.

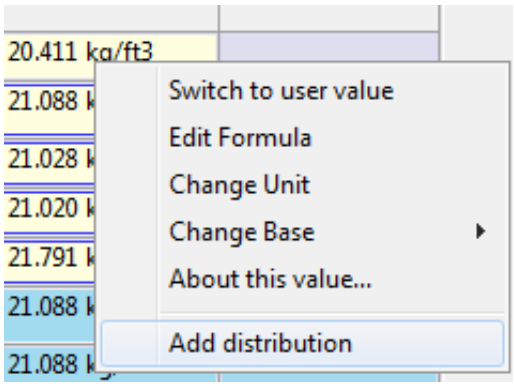


Figure 125: Menu After Right-Click on a Parameter

The distribution editor will appear. Shown in Figure: 126.

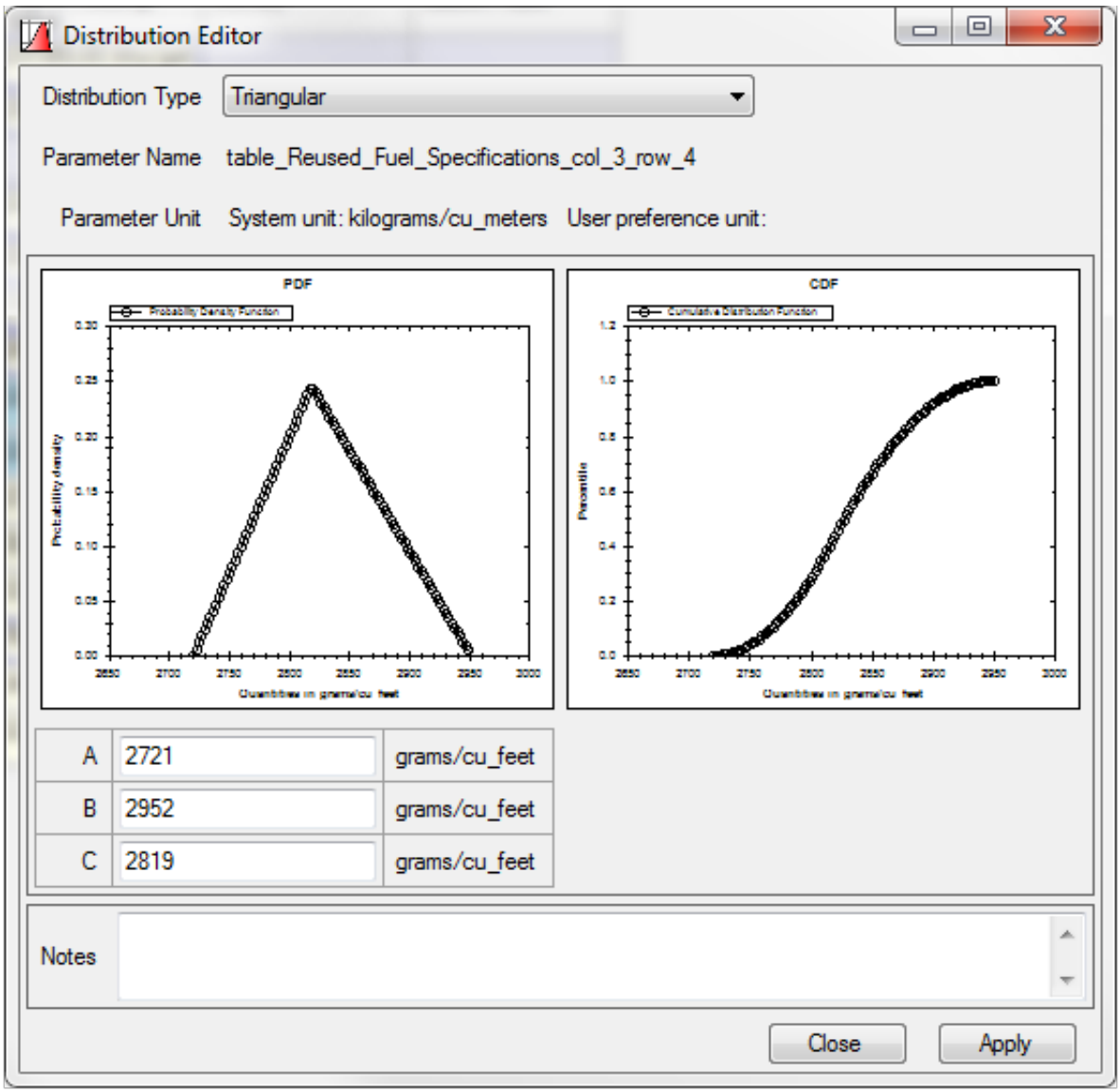


Figure 126: Distribution Editor Showing for a Density Parameter

In sampling the parameter shown above, a triangular distribution is used. For this particular type of distribution, units of the parameters A, B and C are easily known and displayed in preferred units (See unit preferences 4.4.1)

The availability to define a distribution for a parameter using a different unit of measure than GREET's default system unit of measure may not be currently available. If you attempt the change always check the PDF and CDF that the distribution is defined correctly.

The notes field may be used to indicate which publication or person supports the numbers (data) used; documenting the source of the data used.

When parameters are changed, the PDF and CDF are updated, the plots using the current values are in black, and the plots representing the previous values are in red. See Figuire 127.

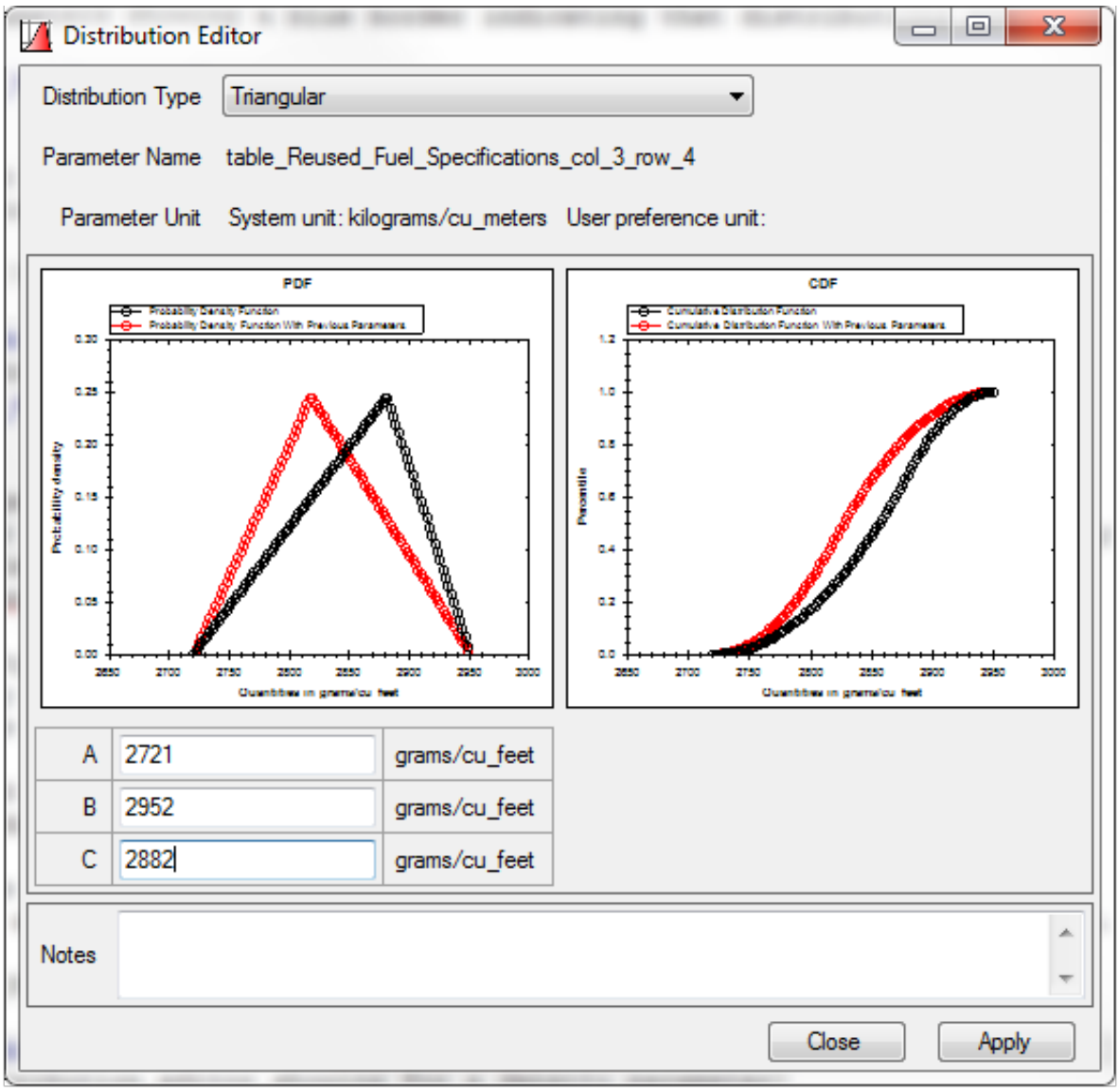


Figure 127: Distribution Editor Showing Colored Plots to See the Influence of Distribution Definition Changes

Some definitions can be entered using P-values. This options is not yet available for all types of distributions. The P-values as well as the probabilities can be changed. In the example below the user defined the distribution by knowing the type (Weibull) as well as P10 and P90 values for that parameter. You'll see that the scale and shape are not offering the option to be defined in preferred unit. Therefore for a Weibull distribution, if one desires to enter this values, they must be appropriately calculated for the system unit (in the case shown below: Joules). See Figuire 128.

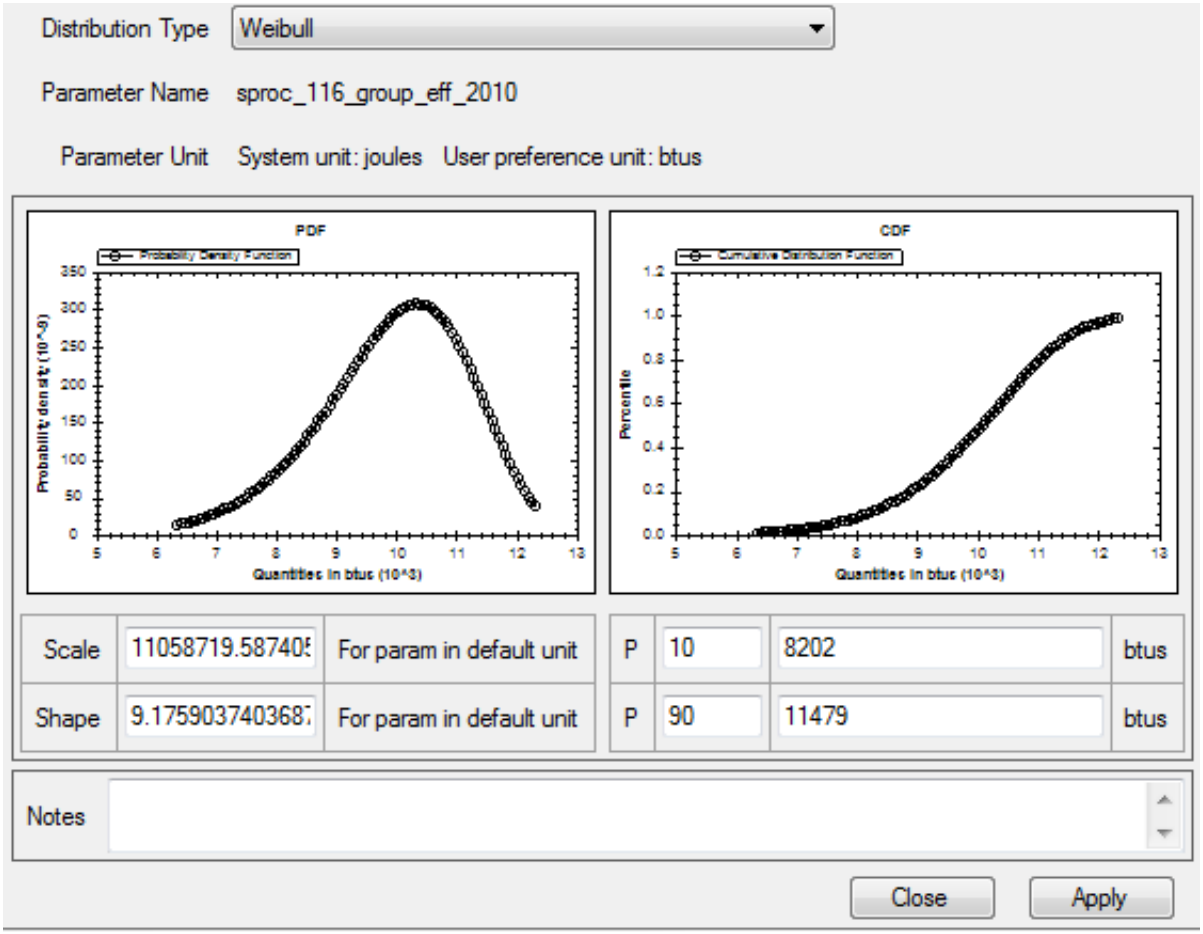


Figure 128: Distribution defined using P-Values

5.3.3 Define and Edit Correlations

Starting GREET 2016, the user can define correlation between distributed parameters. This is important for example in the case where a heating value and density are sampled and rank correlations needs to be respected, for example the higher the heating value, the higher the density.

The correlation can be set from the "Edit Distributions" windows of the stochastic plugin:

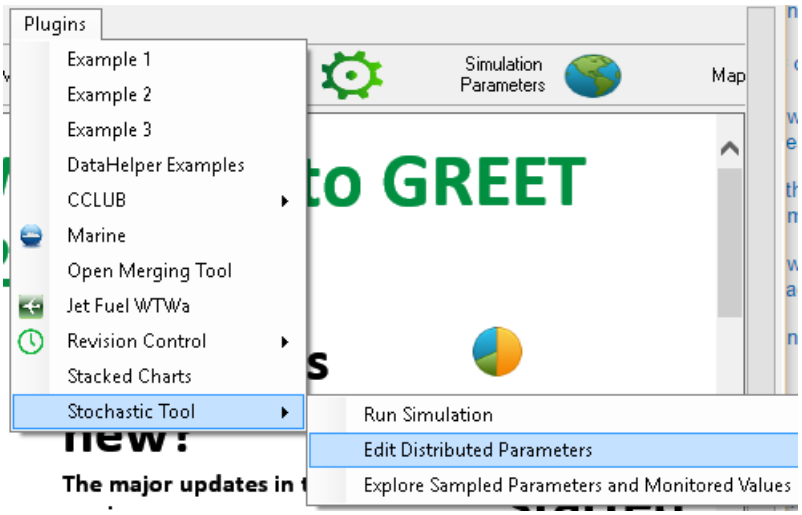


Figure 129: Accessing the distribution editor from the main menu

Then after navigating to the desired distributed parameter, one can click the button "Correlations..."

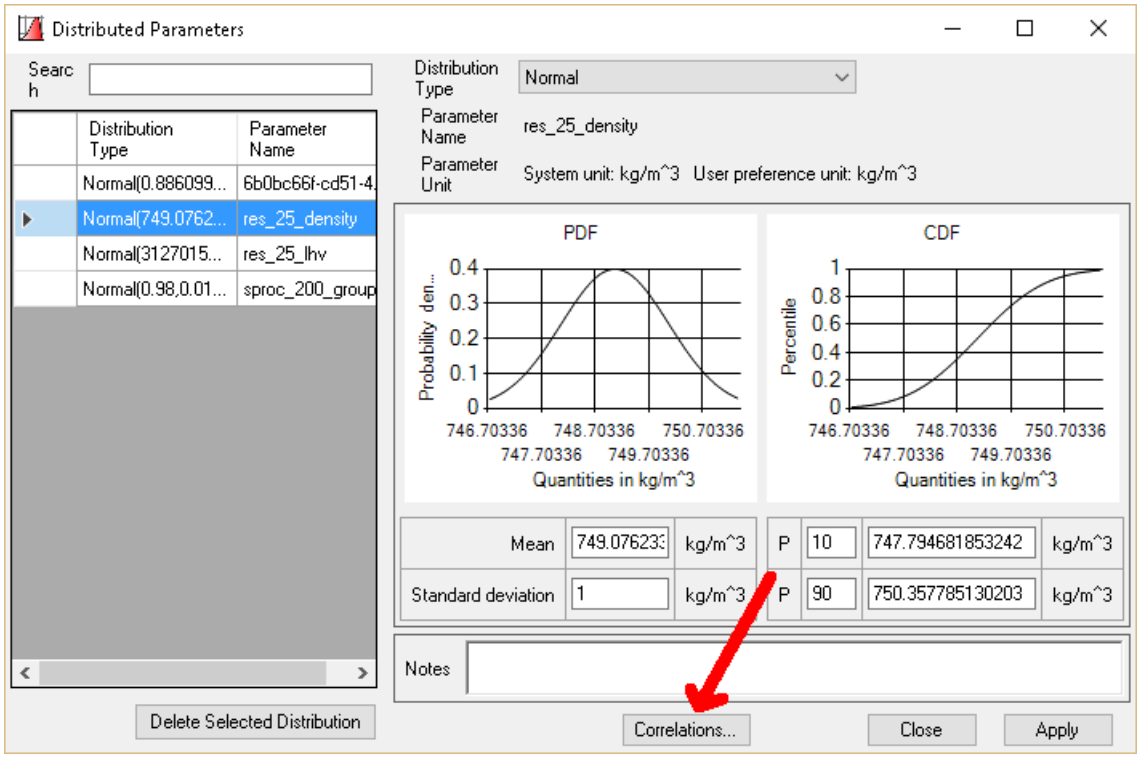


Figure 130: Correlation editor button

From the distribution editor, one can set correlation between parameters. These correlation factors will be then set as targets to a large correlation matrix containing all distributed parameters.

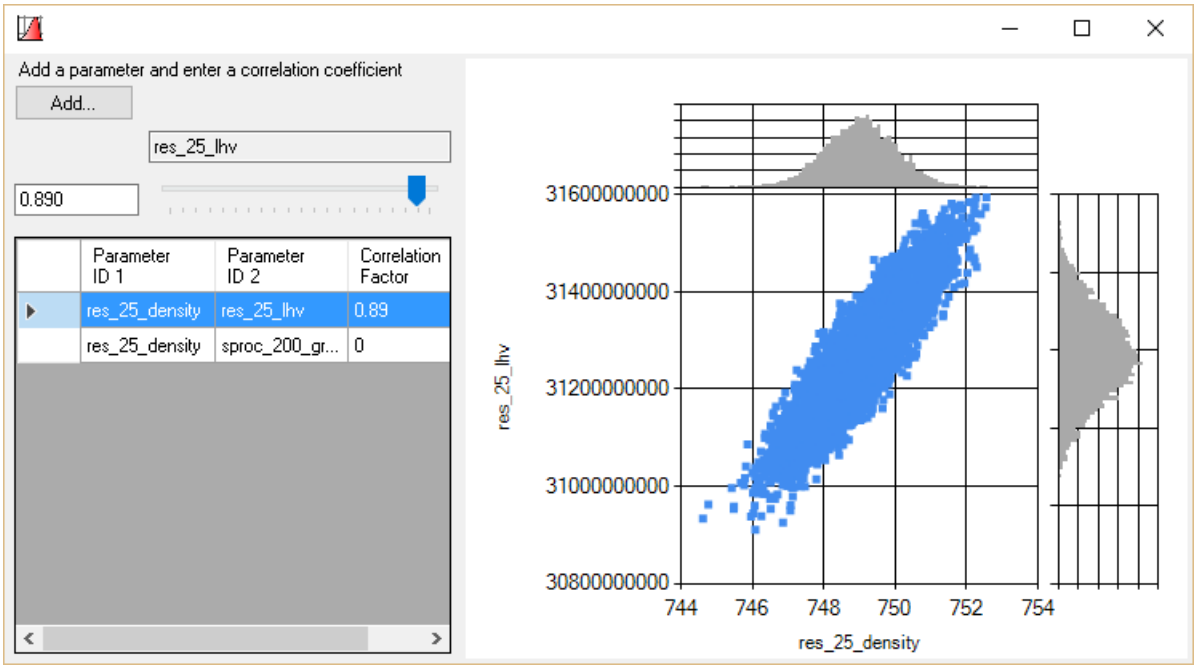


Figure 131: Correlation editor

In order to set a new correlation between variables, one must click the "Add..." button, then select a second parameter by checking the check-box to the left of that parameter.

Once the parameter correlation is added, it is set to zero by default. The user can then choose to change that value by entering a new value in the box on the left, or by sliding the cursor. Changes will be directly observed to the right.

The graph on the right represents the two dimensions being distributed/sampled and the user can visually see how do they correlate. On each axis a histogram also symbolize the probability density bins for the samples.

Once the desired correlation is set, simply close the window.

Correlations can be deleted by right clicking a correlation row, then selecting "Delete correlation"

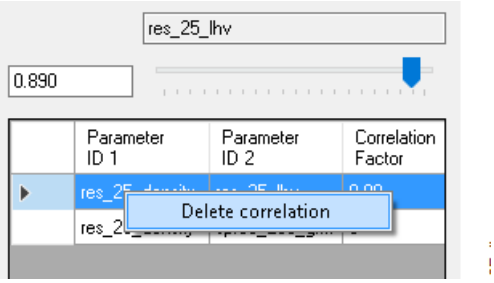


Figure 132: Right click to delete a correlation

Note: Deleting a correlation is different than setting it to zero. By default no correlation are set, but for example if you have 3 parameters, and introduce one correlation between two of them. It is then possible that "unwanted" correlation appears between the two others. In order to force a correlation to be zero between two parameters it is preferable to set the value to zero.

Non positive matrix error If the user sets target correlation coefficient that are not attainable an error message will suggest to resolve this matter with three different options. The safest option is to use the previous coefficient that worked. An optimal method is to use the nearest positive definite matrix coefficients. This will change all targets correlations coefficients to the closest that are possible to obtain using the current implementation.

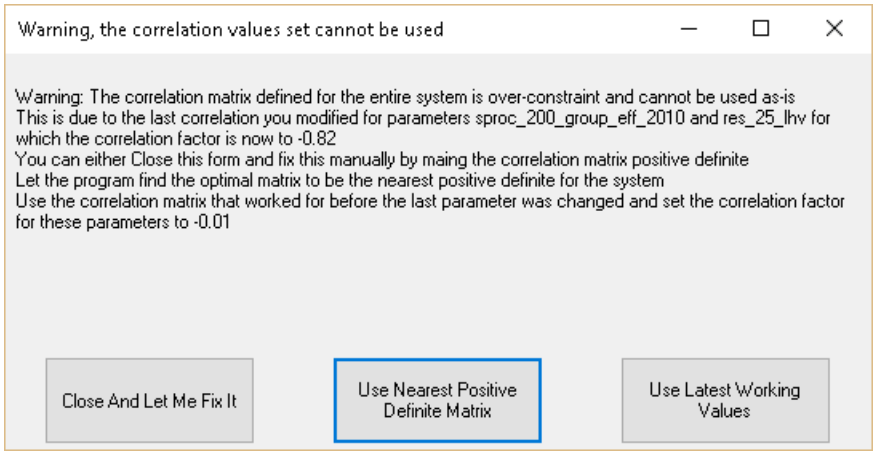


Figure 133: Right click to delete a correlation

In order to understand more about this, please read the two publication referenced in the next paragraph.

How this works? As the user can choose any type of distribution we needed an technique to set correlations regardless of the type of distribution chosen. This is why we're using a rank-correlation approach described in Iman, Ronald L., and William-Jay Conover. "A distribution-free approach to inducing rank correlation among input variables." Communications in Statistics-Simulation and Computation 11.3 (1982): 311-334.

Their variation reduction technique is also implemented so that the samples correlations are very close to the user set targets correlations.

This technique assumes that the target correlation matrix is positive definite as we need the Cholesky decomposition. In case the user provides correlation coefficients that do not define a positive definite matrix, we offer 3 choices: The user can choose to correct the coefficients manually, revert to the latest coefficient that worked, or find the nearest positive definite matrix. This problem is defined in Higham, Nicholas J. "Computing the nearest correlation matrix—a problem from finance." IMA journal of Numerical Analysis 22.3 (2002): 329-343.

Using these two references publications implementations of the techniques were implemented in that plugin.

5.3.4 Running Simulations

To run a simulations, click the main menu item **"Stochastic Tool"**, then click **"Run Simulation"**.

A form will appear showing you options for running a simulation:

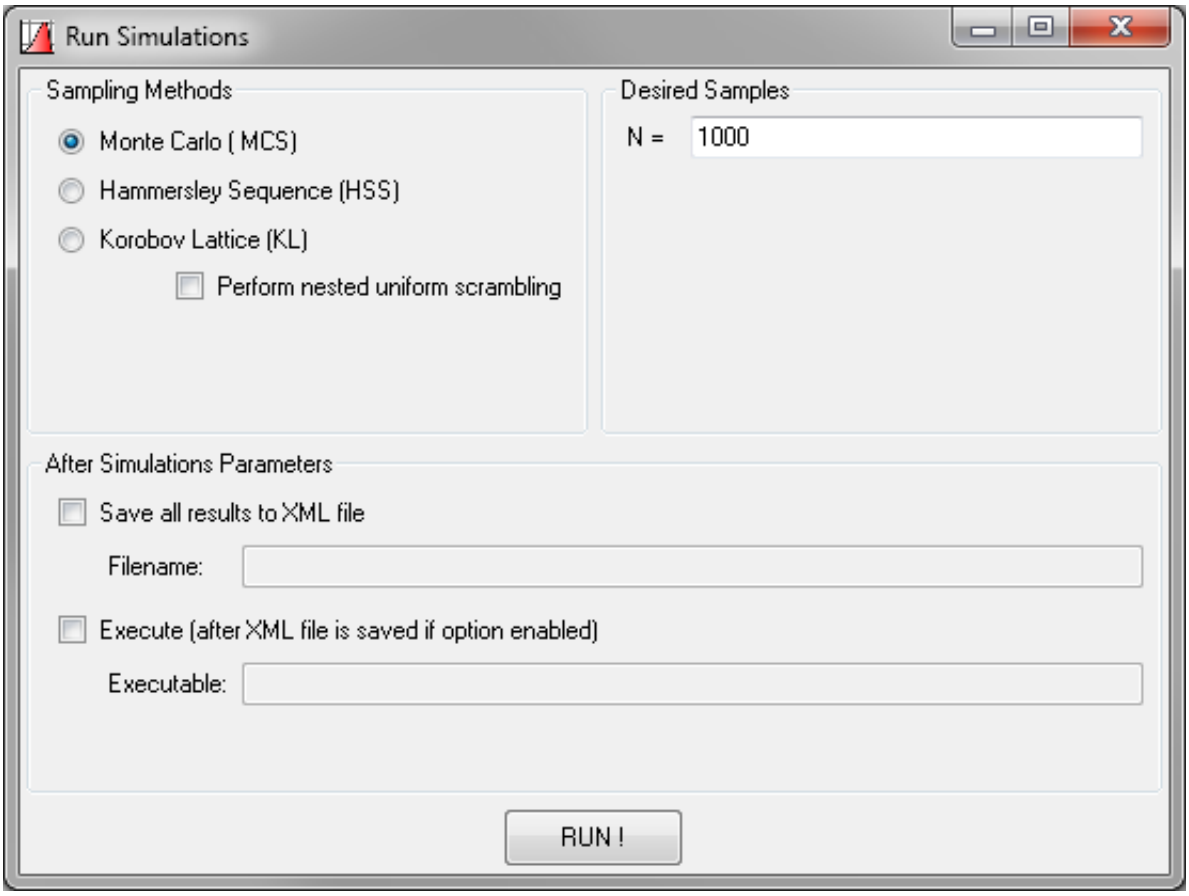


Figure 134: Options before running simulations

Different sampling methods are available, the differences between the available sampling methods are detailed in the next section. But to give a quick overview, the MCS technique is based on a pseudorandom number generator in picking individual samples. However the other methods HSS and KL are based on quasirandom sequences, they offer the advantages of filling up more uniformly the n-space than pseudorandom points.

The number of desired samples depends on your data and sampling method. One has to ensure that there are enough samples to ensure accurate results, but not waste computing time by asking for too many. A good starting point could be 4000 samples using Monte Carlo sampling, or 1000 samples using HHS or KL sequences.

This form also provides options such as running events once the simulation completes. You have the ability to run a batch script, export all the samples and results to a file, or run another software.

Simulation can take hours to complete depending on your computers, memory, processor, and other factors of your computer. The good news is simulations are programmed to run on a separate thread from all other applications and processes current running on your computer. This allows you to keep working on other tasks while the simulation is running.

Once the configuration is done, hit the **"Run!"** button.

NOTE: There is no **"Random Generator Seed"** available to the user, the seed is picked up based on random event such as the current time of your computer and mouse position.

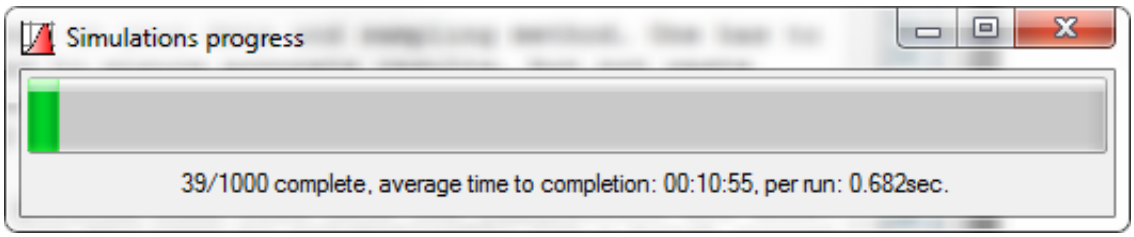


Figure 135: Simulation progress for stochastic simulations

NOTE: **While the simulation is running** don't make any changes to parameters values. Doing so will alter the running of the simulations with unintended results. As the simulation is running parameters value changes are updated and displayed in the "Simulation Parameters" form as they happen as a result of the value changes of the sampling.

5.3.5 Exploring Results

From the main menu item **"Stochastic Tool"**, then click on **"Explore Sampled Parameters and Monitored Values"**

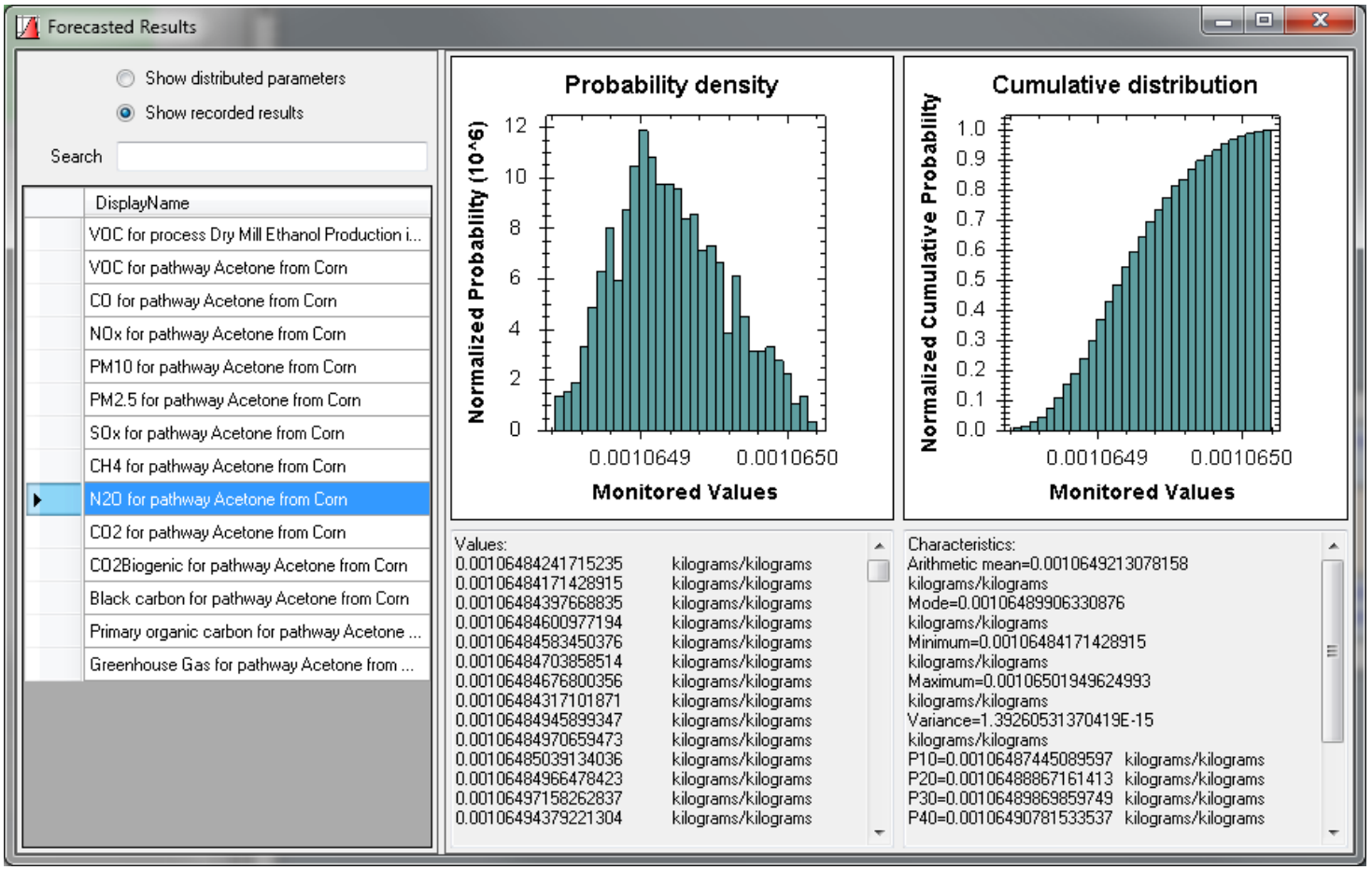


Figure 136: Exploring the Monitored Values or Forecasted Values After 1000 Simulations Using the Monte Carlo Sampling Technique.

This will allow you to go though the monitored values that have been recorded during the simulations as well as looking at the distributed parameters.

Each monitored value or distributed parameter are accessed by ID or name. The search box filters the list of available parameters and forecasts.

The right colume of this list control is histogram representations of the normalized probability density and cumulative distribution. You may want to play with the scales in order to have a nice visualization but the auto-scale feature should work for most cases.

The samples or result values are shown below these charts in order to be re-used somewhere else. You may want for example copy all these results in Excel and perform your own analysis or build your own graphs.

Note that the normalized probability density histogram on the left of the figure looks quite noisy. This is due to the fact that we ran for this example only 1000 simulations using the Monte Carlo sampling technique. Using the Hammerseley sequence in this case would provide better results.

5.3.6 Sampling Methods

In the current version three sampling methods are implemented:

- Korobov Lattice
- Monte Carlo
- Hammersely Sequence

These three samplers are generating values greater or equal to 0 and less than 1, the following figure shows the spaced filled values for first two dimensions using these three techniques. See Figure: 138.

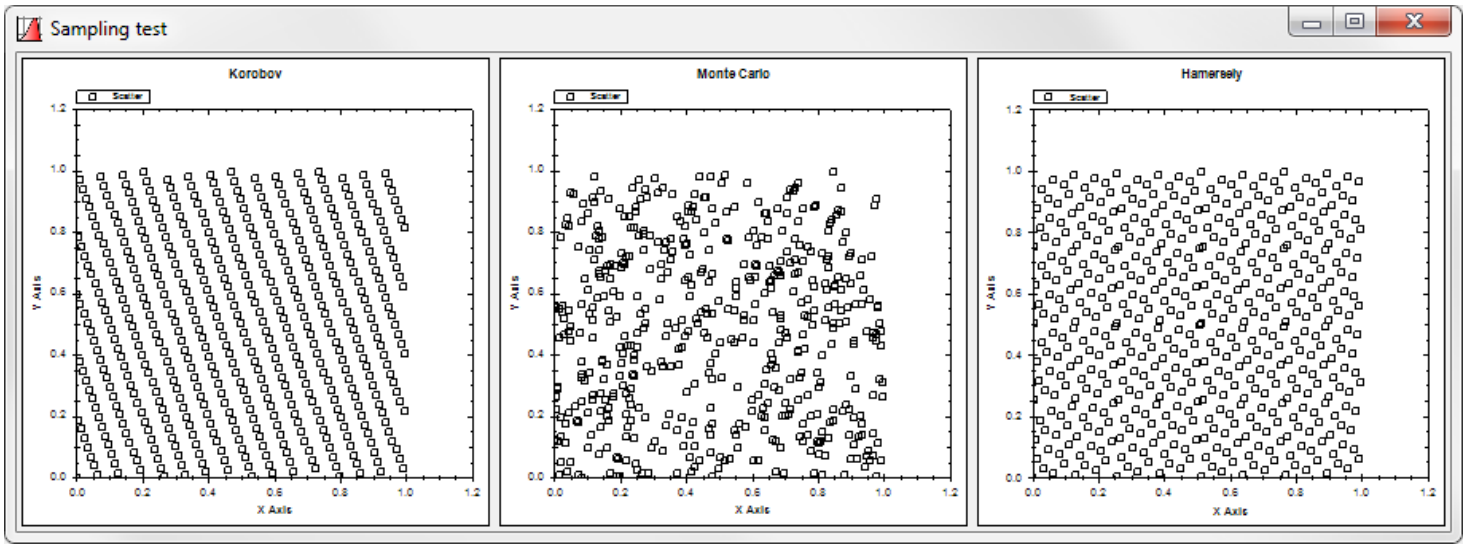


Figure 137: Space filling for the first 2 dimensions with 500 samples

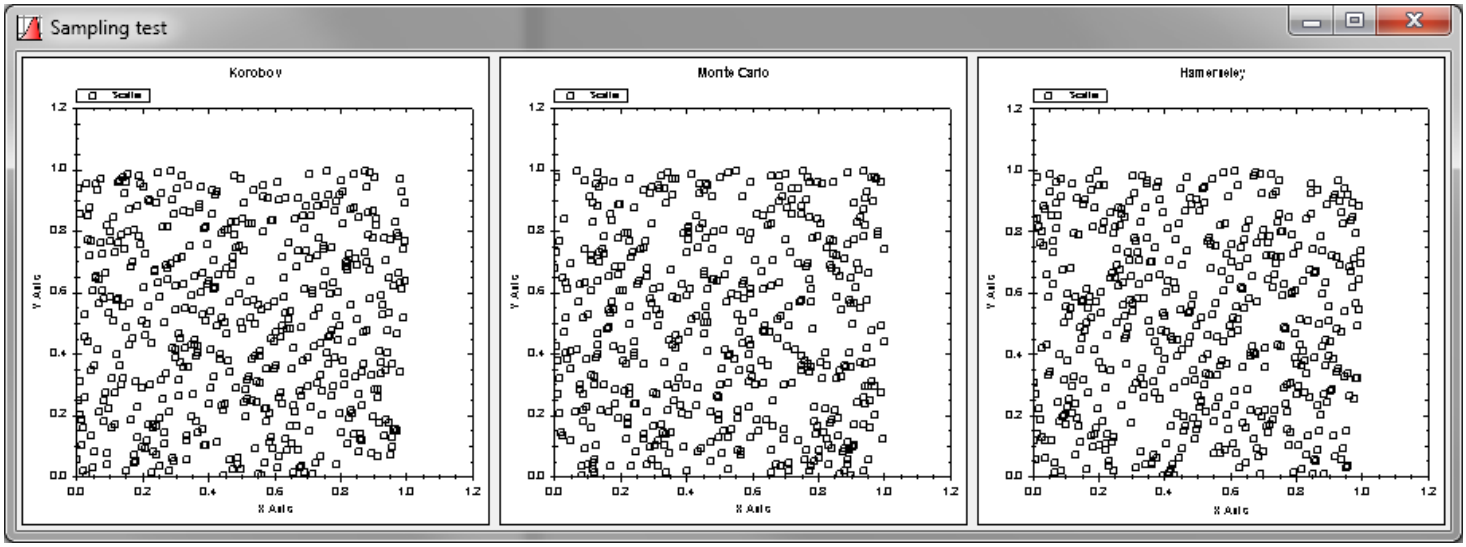


Figure 138: Space filling for the first 2 dimensions with 500 samples and nested uniform scrambling

From the figure above we can clearly see the space filling differences between the three methods. For a small number of samples the Monte Carlo methods trend to some high and low density areas while the two other techniques are more uniform.

The current implementation of the Monte Carlo sampler is based Donald E. Knuth's subtractive random number generator algorithm. This is the default implementation given by the GREET .Net framework.

The implementation of the Hammersely Sequence and Korobov Lattice are based on algorithms that can be seen in the Handbook of Monte Carlo Methods ISBN-10: 0470177934. They both have the advantage of a very smooth space filling, but can act erratically for high dimensions due to the selection of larger prime numbers. The first thousand of prime numbers are hard coded in the software, for larger dimensions a primality test is used to find the next ones.

5.3.7 Distributions Definitions

- Beta

  The beta distribution is defined using two parameters α and β, two shape parameters.

  The PDF is given by:

  $$f(x;\alpha,\beta) = \frac{x^{\alpha-1}(1-x)^{\beta-1}}{B(\alpha,\beta)}$$

  Where $B(\alpha,\beta)$ represents the Beta function

  The CDF is given by:

  $$F(x;\alpha,\beta) = I_x(\alpha,\beta)$$

  Where $I_x(\alpha,\beta)$ represents the regularized incomplete beta function

  If is difficult to compute exact values for the beta function and the regularized incomplete beta function. Therefore numerical approximations are used to compute these functions. The Beta function is calculated from it's relationship with the Gamma function:

  $$B(x,y) = \frac{\Gamma(x)\Gamma(y)}{\Gamma(x+y)}$$

  The Gamma function is computed using an implementation of the Lanczos approximation.

  The quantile function is given by:

  $$Q(p,\alpha,\beta) = I_p^{-1}(\alpha,\beta)$$

  Computation of the inverse incomplete gamma function is done by Halley's root finding method

- Exponential

  The exponential distribution is defined using a single parameter λ the rate or inverse scale.

  The PDF is given by:

  $$f(x;\lambda) = \lambda \exp^{-\lambda x}$$

  The CDF is given by:

  $$F(x;\lambda) = 1 - \exp^{-\lambda x}$$

  And the quantile function is given by:

  $$Q(p,\lambda) = -\frac{1}{\lambda} Log(p)$$

- Frechet

  The Frechet distribution is a specific case of the GEV when $\xi > 0$. Therefore the distribution is defined using three parameters: α the shape, s the scale and m the location. This distribution is a special case of the Generalized Extreme Value described below.

  The PDF is given by:

  $$f(x;\alpha,s,m) = \frac{\alpha}{s}\left(\frac{x-m}{s}\right)^{-1-\alpha} \exp\left(-\left(\frac{x-m}{s}\right)^{-\alpha}\right)$$

  The CDF is given by:

  $$F(x;\lambda) = \exp\left(-\left(\frac{x-m}{s}\right)^{-\alpha}\right)$$

  And the quantile function:

  $$Q(p,\alpha,s,m) = m + s \cdot -Log(p)^{-\frac{1}{\alpha}}$$

- Gamma

  The gamma distribution can be defined in multiple ways, all equivalent, using different sets of parameters. Internally the library uses the α shape, β rate definition. From the user interface the scale: θ can also be used with:

  $$\theta = \frac{1}{\beta}$$

  The PDF is given by:

  $$f(x; \alpha, \beta) = \frac{\beta^\alpha x^{\alpha-1} e^{-\beta x}}{\Gamma(\alpha)}$$

  Where $\Gamma(a)$ represents the Gamma function. The Gamma function is computed using an implementation of the Lanczos approximation.

  The CDF is given by:

  $$F(x; \alpha, \beta) = \frac{\gamma(\alpha, \beta x)}{\Gamma(\alpha)}$$

  Where $\gamma(\alpha, \beta x)$ is the lower incomplete gamma function. In this module this function is not directly calculated. We are using an algorithms that calculates the regularized gamma function directly.

  The Regularized incomplete lower gamma function is given by

  $$P(s, x) = \frac{\gamma(s, x)}{\Gamma(s)}$$

  which is also the CDF of the Gamma distribution. In order to perform this, the algorithm uses different approximations techniques to calculate this function depending on the values of s and x.

  The quantile function:

  $$Q(p; \alpha, \beta) = \frac{P^{-1}(\alpha, p)}{\beta}$$

  Where P^{-1} represents the inverse regularized incomplete lower gamma function, this function is computed using Halley's root finding method.

- Generalized Extreme Value

  The GEV distribution can be seen as a generalization of the Frechet, Weibull and Gumbel distributions. It is defined using three parameters: μ the location, σ the scale and ξ the shape.

  The shape parameter ξ defines if we are one in the sub families of distributions. For $\xi = 0$ the GEV corresponds to a Gumbel distribution, for $\xi > 0$ the GEV corresponds to a Frechet distribution and for $\xi < 0$ the GEV can be assimilated as a Weibull distribution.

  The PDF is given by:

  $$f(x; \mu, \sigma, \xi) = \frac{t^{\xi+1} e^{-t}}{\sigma}$$

  where

  $$t(x, \mu, \sigma, \xi) = \begin{cases} \left(1 + \xi \frac{x-\mu}{\sigma}\right)^{-1/\xi} & \text{if } \xi \neq 0 \\ e^{\frac{-(x-\mu)}{\sigma}} & \text{if } \xi = 0 \end{cases}$$

  The CDF if given by:

  $$F(x; \mu, \sigma, \xi) = e^{-t(x)}$$

  The quantile function:

  $$Q(p; \mu, \sigma, \xi) = \begin{cases} -\frac{(-Log(p))^{-\xi}(\sigma(-Log(p))^{\xi} - \sigma - \mu\xi(-Log(p))^{\xi})}{\xi} & \text{if } \xi \neq 0 \\ \mu - \sigma \cdot Log(-Log(p)) & \text{if } \xi = 0 \end{cases}$$

- Gumbel

  Gumbel distribution is a specific case of the GEV when $\xi = 0$. Therefore we only define two parameters for the Gumbel definition: μ the location and β the scale.

The PDF is given by:

$$f(x;\mu,\beta) = \frac{e^{-z-e^{-z(x)}}}{\beta}$$

where

$$z(x,\mu,\beta) = \frac{x-\mu}{\beta}$$

The CDF if given by:

$$F(x;\mu,\beta) = e^{-e^{-z}}$$

and the quantile:

$$Q(p;\mu,\beta) = \mu - \beta Log(-Log(p))$$

- Logistic

  The logistic distribution is based on a common location-scale definition. The parameters used are μ the locations and s the scale.

  The PDF is given by:

$$f(x;\mu,s) = \frac{e^{-\frac{x-\mu}{s}}}{s\left(1+e^{-\frac{x-\mu}{s}}\right)^2}$$

  The CDF is given by:

$$F(x;\mu,s) = \frac{1}{1+e^{-\frac{x-\mu}{s}}}$$

  and the quantile:

$$Q(p;\mu,\beta) = \mu + s \cdot Log(\frac{1}{p} - 1)$$

- LogNormal

  The LogNormal distribution is defined using two parameters σ^2 the log-scale and μ the shape.

  The PDF is given by:

$$f(x;\mu,\sigma) = \frac{1}{x\sigma\sqrt{2\pi}}\, e^{-\frac{(\ln x-\mu)^2}{2\sigma^2}}$$

  The CDF is given by:

$$F(x;\mu,\sigma) = \frac{1}{2}\left[1+\operatorname{erf}\left(\frac{\ln x-\mu}{\sigma\sqrt{2}}\right)\right]$$

  For this distribution the quantile function is not used. In that case we are generating LogNormally distributed values using a normally distributed variable: if $Z \sim \mathcal{N}(\mu,\sigma^2)$ then $X = e^{\mu+\sigma Z}$

- Normal

  The normal distribution is defined using two parameters: μ the mean or location and σ the standard deviation.

  The PDF is given by:

$$f(x;\mu,\sigma) = \frac{1}{\sigma\sqrt{2\pi}} e^{-\frac{(x-\mu)^2}{2\sigma^2}}$$

  The CDF is given by:

$$F(x;\mu,\sigma) \;=\; \frac{1}{2}\left[1+\operatorname{erf}\left(\frac{x-\mu}{\sigma^2\sqrt{2}}\right)\right]$$

  and the quantile:

$$Q(p;\mu\sigma) = \mu + \sigma\sqrt{2} \cdot erf^{-1}(2\sigma^2)$$

  The inverse error function erf^{-1} is computed from a polynomial approximation followed by a Newton's roots finding method.

- Pareto

  The pareto distribution is defined by two parameters: β the scale and α the shape

  The PDF is given by:

$$f(x;\alpha,\beta) = \begin{cases} \dfrac{\alpha\beta^{\alpha}}{x^{\alpha+1}} & \text{for } x \geq \beta \\ 0 & \text{for } x < \beta \end{cases}$$

  The CDF is given by:

$$F(x;\alpha,\beta) = \begin{cases} 1 - \left(\frac{\beta}{x}\right)^{\alpha} & \text{for } x \geq \beta, \\ 0 & \text{for } x < \beta. \end{cases}$$

  and the quantile:

$$Q(p;\alpha,\beta) = \beta(1-p)^{-1/\alpha}$$

- Triangular

  The triangular distribution is defined using three parameters: lower limit a, upper limit b and mode c

  The PDF is given by:

$$f(x;a,b,c) = \begin{cases} 0 & \text{for } x < a, \\ \frac{2(x-a)}{(b-a)(c-a)} & \text{for } a \leq x \leq c, \\ \frac{2(b-x)}{(b-a)(b-c)} & \text{for } c < x \leq b, \\ 0 & \text{for } b < x, \end{cases}$$

  The CDF is given by:

$$F(x;a,b,c) = \begin{cases} 0 & \text{for } x < a, \\ \frac{(x-a)^2}{(b-a)(c-a)} & \text{for } a \leq x \leq c, \\ 1 - \frac{(b-x)^2}{(b-a)(b-c)} & \text{for } c < x \leq b, \\ 1 & \text{for } b < x. \end{cases}$$

  and the quantile:

$$Q(p;a,b,c) = \begin{cases} a + \sqrt{p(b-a)(c-a)} & \text{for } 0 < p < \frac{c-a}{b-a} \\ b - \sqrt{(1-p)(b-a)(b-c)} & \text{for } \frac{c-a}{b-a} \leq U < 1 \end{cases}$$

- Uniform

  The uniform distribution only defines two boundaries for sampling: a the lower boundary and b the upper boundary

  The PDF is given by:

$$f(x;a,b) = \begin{cases} \frac{1}{b-a} & \text{for } x \in [a,b] \\ 0 & \text{otherwise} \end{cases}$$

  The CDF is given by:

$$F(x;a,b) = \begin{cases} 0 & \text{for } x < a \\ \frac{x-a}{b-a} & \text{for } x \in [a,b) \\ 1 & \text{for } x \geq b \end{cases}$$

  and the quantile:

$$Q(p;a,b) = a + (b-a) * p$$

- Weibull

  The Frechet distribution is a specific case of the GEV. Therefore the distribution is defined using two parameters: λ the scale and k the shape.

The PDF is given by:

$$f(x;\lambda,k) = \begin{cases} \frac{k}{\lambda}\left(\frac{x}{\lambda}\right)^{k-1} e^{-(x/\lambda)^k} & x \geq 0 \\ 0 & x < 0 \end{cases}$$

The CDF is given by:

$$F(x;\lambda,k) = \begin{cases} 1 - e^{-(x/\lambda)^k} & x \geq 0 \\ 0 & x < 0 \end{cases}$$

and the quantile:

$$Q(x;\lambda,k) = \lambda(-Log(1-p))^{1/k}$$

5.4 CCLUB Tool

The Carbon Calculator for Land Use Change from Biofuels Production (CCLUB) module was released by Argonne as an Excel spreadsheet that functions both as a standalone model and as a component of GREET. CCLUB estimates the direct (domestic) and indirect (international) emissions that occur as a result of land use changes during the production of ethanol. Corn, corn stover, miscanthus and switchgrass feedstocks are supported by CCLUB.

Argonne has developed the CCLUB Tool plugin to integrate the CCLUB spreadsheet with the GREET user interface. A valid CCLUB spreadsheet (version 2014 or later) is required and is included with the GREET software. By default, this spreadsheet is made available in the user's My Documents folder (My Documents\GREET\Data \CCLUB_2014_for_GREET1_2014.xlsm), but any valid CCLUB 2014 spreadsheet on the computer's hard drive may be selected.

Comprehensive instructions on CCLUB use and development are available in CCLUB Technical Documentation at `http://greet.es.anl.gov/`

The CCLUB Tool plugin requires Microsoft Excel 2003 or later to be installed on your machine to run properly.

5.4.1 Opening the CCLUB Tool

There are two ways to open the CCLUB Tool in GREET. If you wish to run CCLUB and copy the results to the clipboard, the tool can be opened from the Plugins ->Excel Tools ->CCLUB Menu.

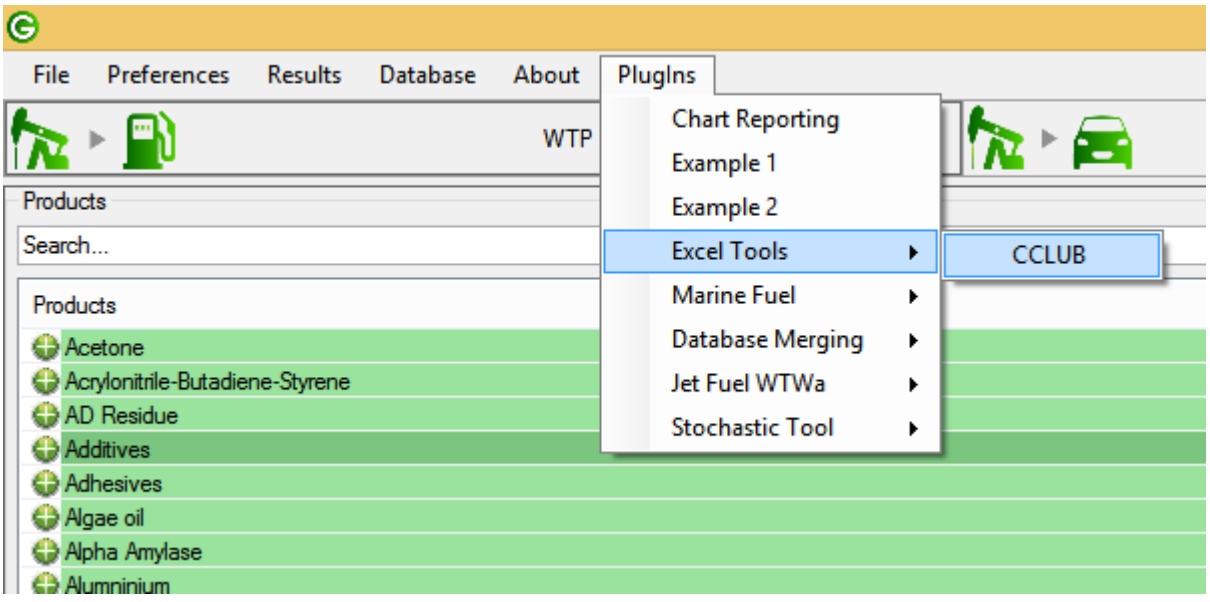


Figure 139: Opening the CCLUB Tool from the menu

You may also associate a session of the CCLUB Tool with a specific CO2 parameter from the process editor in GREET. To do this, open the process to which you'd like to add or modify a land use change CO2 emission from CCLUB. If you already have a land use change CO2 parameter, right click in the CO2 value textbox, then click "Get from CCLUB". If you need to

add a land use change CO2 parameter to the process, you can find CO2 in the "Special Items" menu on the left, then click and drag the CO2 item to the output of your process.

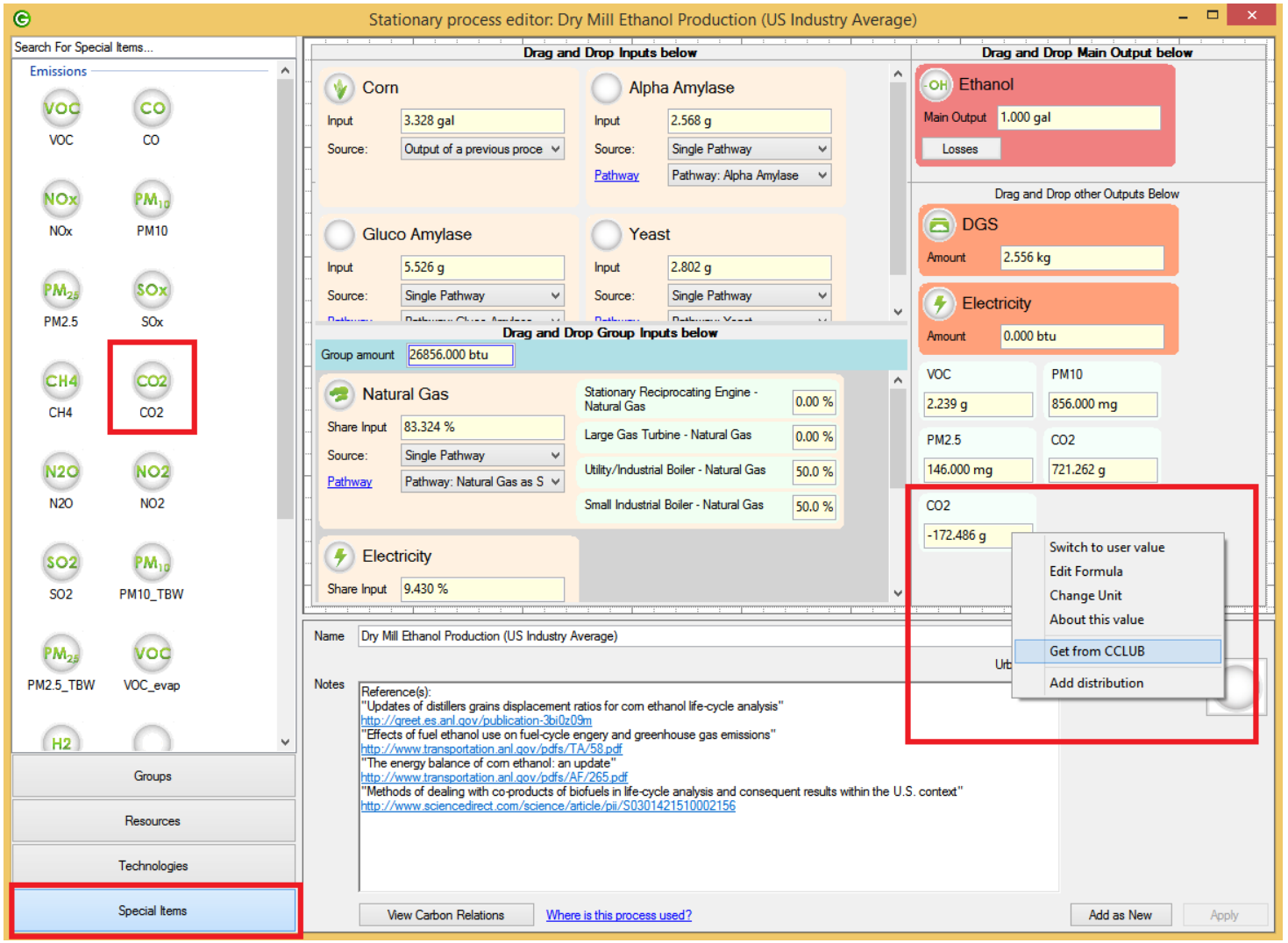


Figure 140: Opening CCLUB from a new or existing CO2 parameter

5.4.2 Using the CCLUB Tool

Once you have opened the CCLUB Tool from the GREET menu or from a CO2 parameter, you may use the default spreadsheet included with GREET and automatically loaded with the tool, or you may click the Browse button to find a different CCLUB spreadsheet that you may have been using. Once a new CCLUB spreadsheet has been chosen, the inputs will be automatically loaded from the spreadsheet and into the CCLUB Tool.

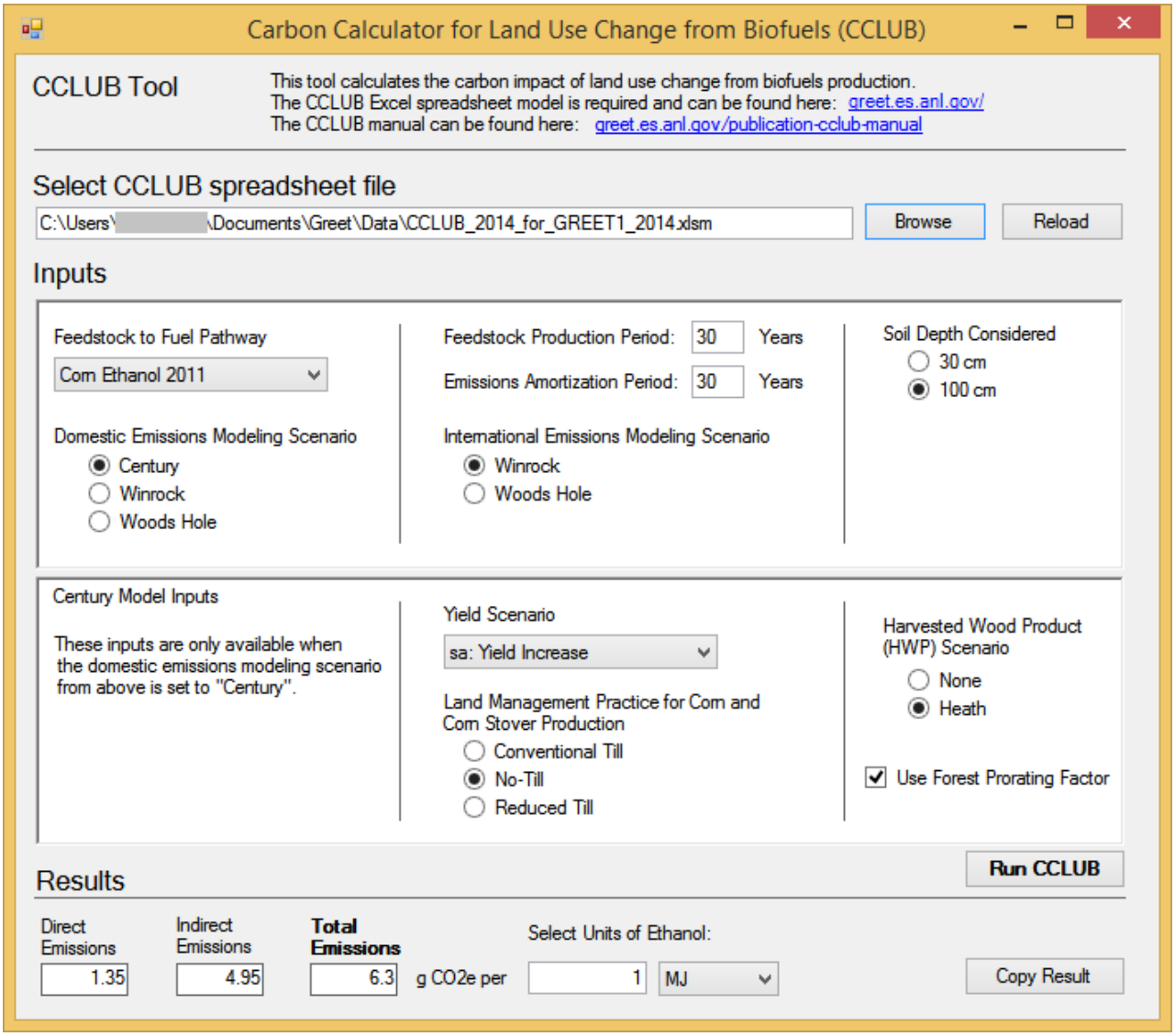


Figure 141: The CCLUB Tool

The inputs shown in the CCLUB Tool are the same as those found in the CCLUB Excel spreadsheet, but with a user friendly and responsive interface. Note that there are two sets of inputs in the CCLUB Tool: General settings in the top input panel and Century Model Inputs in the second input panel. Century Model Inputs are only available when the Domestic Emissions Modeling Scenario input is set to "Century" and are not visible if Winrock or Woods Hole are selected. Additionally, the input "Land Management Practice for Corn and Corn Stover Production" is only available when the of Feedstock to Fuel Pathway is set to Corn or Corn Stover.

You may change any of the inputs to whichever values you'd like to simulate. To send these inputs to CCLUB and calculate the results, click "Run CCLUB". Please note, the CCLUB Tool cannot calculate results if you have the CCLUB spreadsheet open in Excel or another program.

NOTE: When "Run CCLUB" is clicked, the chosen CCLUB spreadsheet will have new inputs written to it and the file will be saved. To ensure that your work is not overwritten, you may create a copy or use a new CCLUB file.

After the CCLUB Tool has opened the CCLUB spreadsheet and run calculations in the background, the new results are displayed near the bottom of the window. Direct (domestic), indirect (international) and total grams of CO2 are displayed in terms of the chosen amount and unit (Megajoules, gallons, btus, and mmbtus). The results will be converted immediately when you enter a new amount or chose any of the four units.

5.4.3 Exporting Results

Once CCLUB results are generated, you may manually copy the results by right clicking each text box, or you can copy the total result to the clip board by clicking the "Copy Result" button.

If you have opened CCLUB in association with a specific CO2 parameter in a GREET process (see *Opening the CCLUB Tool* above), you can simply click "Send to GREET" and the Total Emissions result will automatically be sent back to the process editor in GREET. This option is only available if you have opened the tool in association with a specific CO2 parameter.

Another window will pop up to confirm the units in which you would like to send the results back to GREET. Please verify that these units match with your process output (such as "1 mmbtu" or "1 gal") so that you are saving a correct result.

It is recommended that you add a note to your CO2 parameter in order for yourself and others to understand where this number came from. In the process editor, right click on the parameter and click "About this value", then enter a note describing the CCLUB scenario that you used to calculate this result.

Please note that you must click "Apply" in the GREET process editor in order for your work to be saved.

5.5 Soil Carbon Emission Calculator

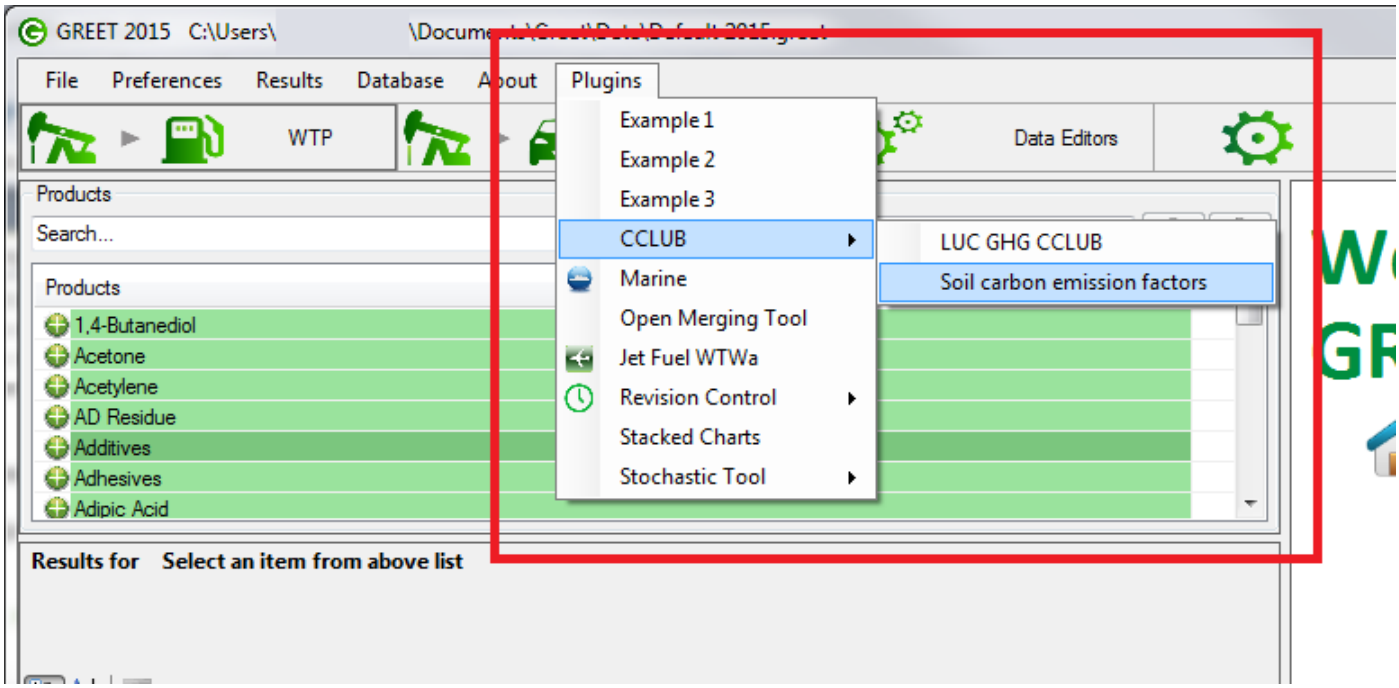
The Soil Carbon Emission Calculator provides soil carbon stock changes that are a result of land use change. This module is based off of Argonne's Carbon Calculator for Land Use Change from Biofuels Production (CCLUB). Whereas the CCLUB spreadsheet and GREET module (see previous section above) will calculate carbon emissions for a variety of biofuel production pathways and scenarios, this Soil Carbon Emissions Calculator allows the user to look more specifically into the Century Model results on which CCLUB is based.

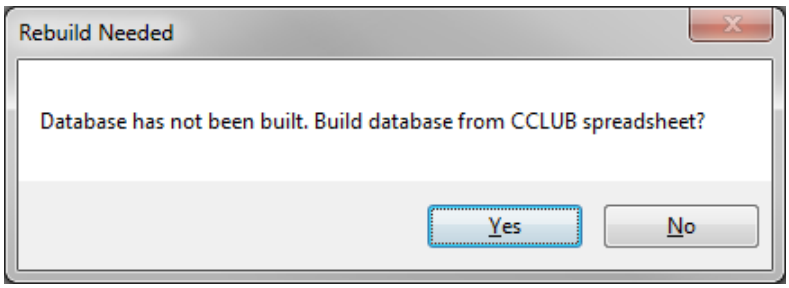
With this tool, the user can select any county (by state and name, or by FIPS code) or Agro-Ecological Zone in the United States and find the change in soil carbon stock (Mg C / hectare-year) for a number of land use or land management change scenarios.

Argonne has developed this plugin to integrate the CCLUB spreadsheet with the GREET user interface. A valid CCLUB spreadsheet (version 2015 or later) is required and is included with the GREET software. By default, this spreadsheet is made available in the user's My Documents folder (My Documents \GREET\Data\CCLUB 2014 for GREET1 2014.xlsm), but any valid CCLUB 2014 spreadsheet on the computer's hard drive may be selected. Comprehensive instructions on CCLUB use and development are available in CCLUB Technical Documentation at http://greet.es.anl.gov/

5.5.1 Opening the Soil Carbon Emission Calculator

To access this tool, simply go to the Plugins menu in GREET, click on the CCLUB sub-menu, and then click on "Soil carbon emission factors". A new window will open.





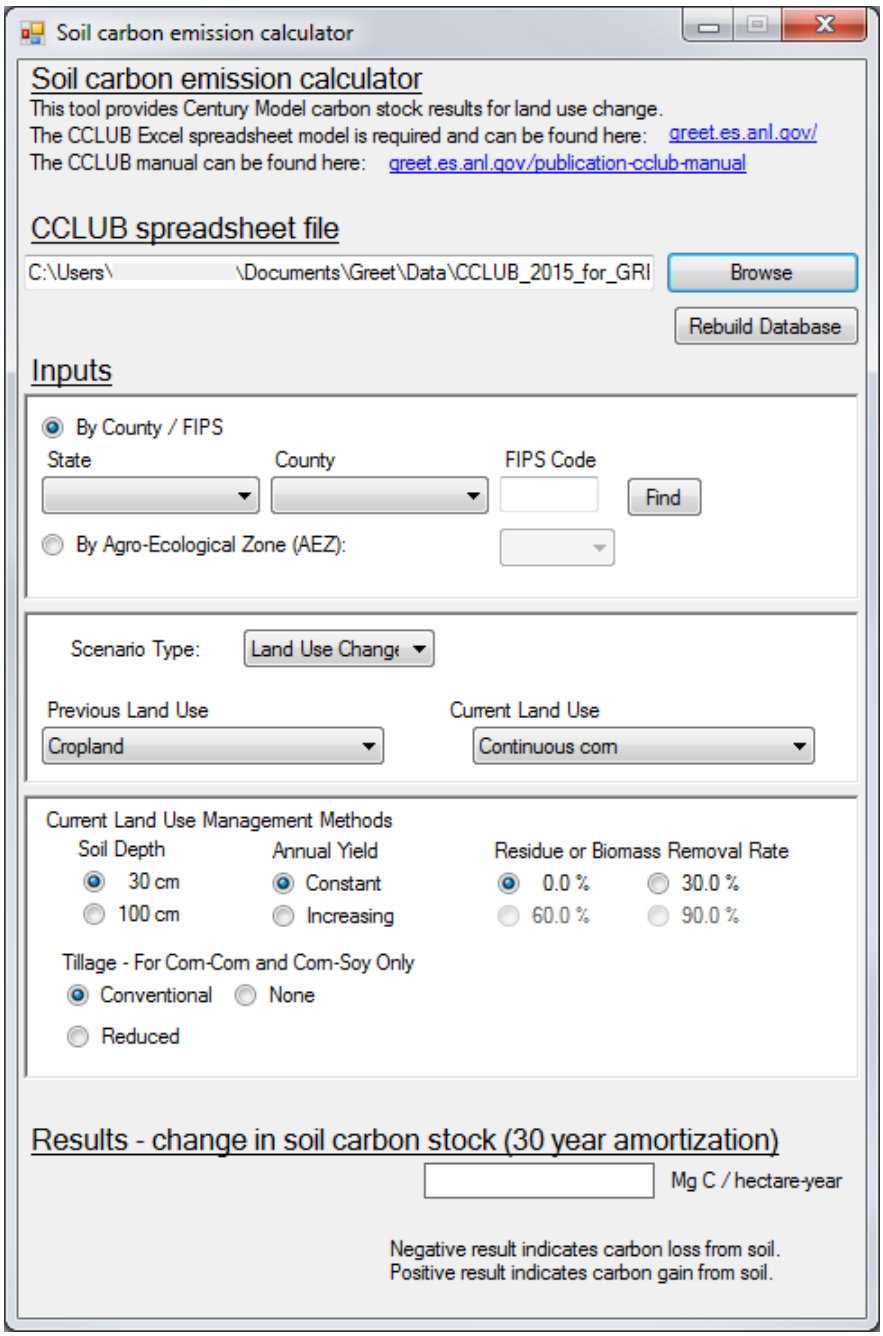
The first time you open this tool, or if the tool cannot find a soil carbon database, it will first need to build the database. This requires a CCLUB 2015 spreadsheet file. If you do not have a CCLUB 2015 file in the default Greet\Data folder on your computer, you may browse to your own version elsewhere. Otherwise, GREET will automatically copy a default CCLUB 2015 spreadsheet to your Greet\Data folder.

If you click "Yes" to build the database, the program will need a few minutes to process the data. Note that the status message near the top of the window will indicate the program's process. During this time, the program is building a database of every possible land use or land management scenario in every single county or AEZ in the United States.

After building the database once, the program will not need to do this again, and will be ready immediately upon opening each time. However, if the database file is missing or corrupted, the program can easily rebuild another by clicking on the "Rebuild Database" button.

5.5.2 Using the plugin: Selecting scenario location and parameters

Once a database has been built or detected, the Soil Carbon Emission Calculator is ready to use. There are three main panels that contain inputs for: location; scenario type; and specific land management assumptions.



Step 1: Select a location First, select a location by picking a state in the dropdown list. Once a state is selected, that state's counties will be listed in the County dropdown list. If you know the FIPS code of the county you'd like to use, simply type the code into the "FIPS Code" box and click Find. If the FIPS code matches the default database, it will automatically select the county you are looking for.

You may also find results for Agro-Ecological Zones (AEZ) instead of by county. To do this, click on the button next to "By Agro-Ecological Zone (AEZ), then select which zone you'd like to use. Note: Numerous counties throughout the United States will display results based on their AEZ zone. This happens when a county does not have results specified by the Century Model. If this is the case, the AEZ dropdown menu will display which zone is being used for results.

Step 2: Select a scenario You may select between Land Use Change scenarios and Land Management Change scenarios. For Land Management Scenarios, the only option for "Current Land Use" is "Corn-soy rotation". For Land Use Change, you can select any previous land use and any current land use.

Step 3: Specify land management assumptions A variety of model assumptions for land practices are available to select. You may select options for soil depth, annual yield growth, tillage practices, residue or biomass removal rates and

carbon inputs. Note that some inputs are not available or cannot be changed under certain scenarios due to not being applicable or not having model results available.

Results: The results are displayed at the bottom of the window. Each time any input is changed, the value in the display box will automatically change to the correct result. Negative results indicate carbon loss from soil, while positive results indicate carbon gain. The results are reported in mega grams of carbon per hectare per year, or Mg C / hectare-year.

You can right click on the Results display box and copy the result for use elsewhere.

5.6 Marine

Analysis of well-to-haul (WTH) of marine fuels is performed in this plug-in. There are two stages of the life cycle of a marine fuel, namely is well-to-pump (WTP) and pump-to-haul (PTH). Combined together WTP and PTH gives the WTH. The implementation of the WTP (well-to-pump) pathways for such marine fuels as Residual Oil, Biodiesel and Fischer–Tropsch diesel has been previously done as part of the GREET model. The Marine module allows us to calculate combined WTWa results for each vessel type and fuel type combination.

Functional unit for the emissions and energy results can be chose by user, current options are trip, mile, ton mile. The energy results are divided into several categories. The first category is PTH NG energy use. The second category is PTH Petroleum energy use, that include vessel operation and energy associated with fuel production. The energy use is broken down by type in the PTH cycle; energy types include fossil, petroleum, natural gas, and coal.

Currently there are tree classes of vessels (Bulk, Container Large, and Tanker VLCC). Each vessel type is characterized by power rating of its two engines - main and auxiliary. For each engine, fuel consumption and emission factors are defined. For mode details on data sources of the input parameters, see [3].

5.6.1 Using the Module

To open the module control click on PlugIns on the top menu bar and then Marine and you will see the main module window.

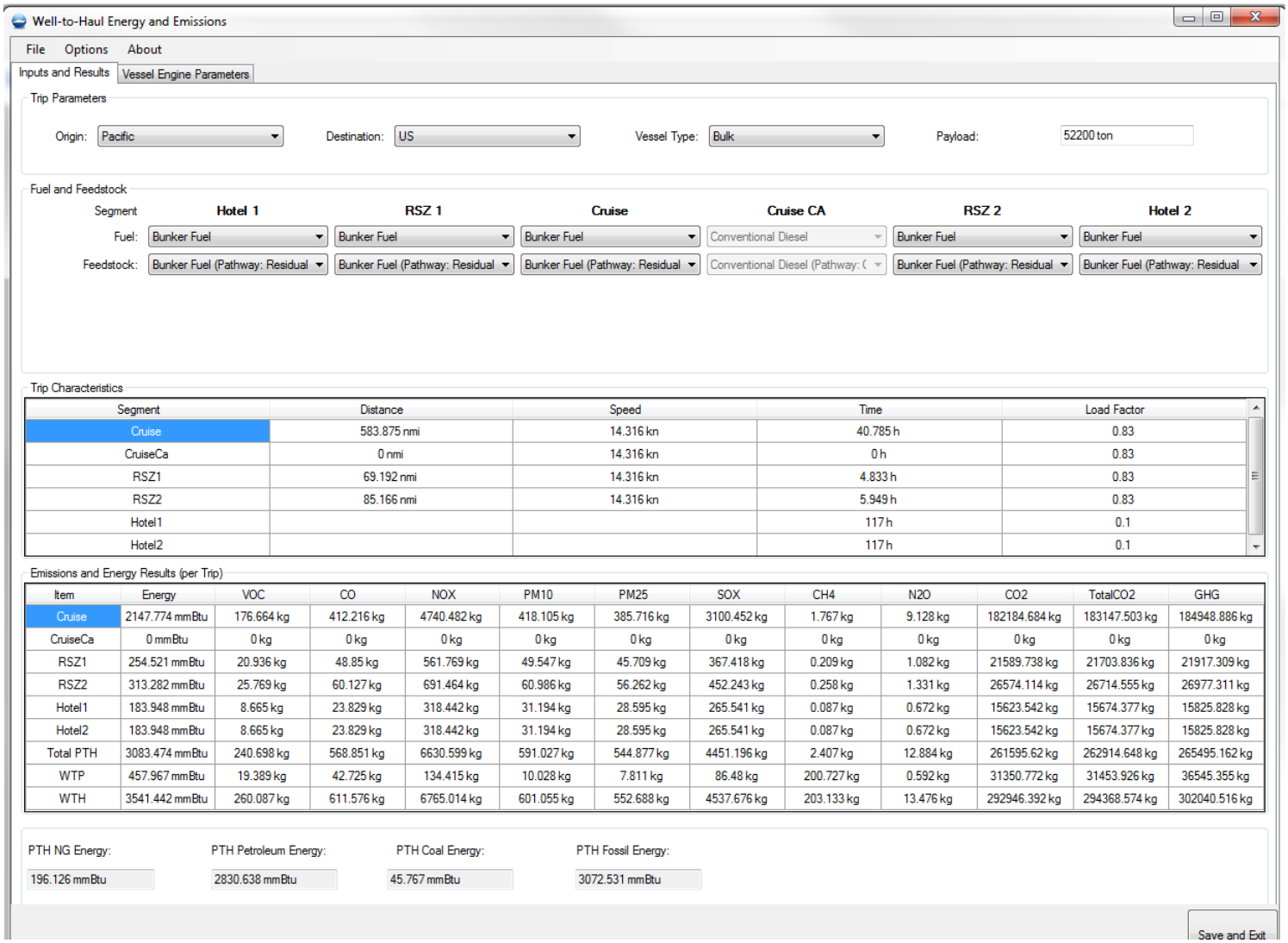


Figure 142: Marine Plug-in

Exploring results for different vessel and route configurations For a given route and vessel type the trip characteristics, fuel and feedstock are defined for each segment of the trip. When choosing another origin, destination or vessel type, the trip characteristics, fuel and feedstock selection and results will be loaded.

For any segments but hotelling a user can change distance, speed and load factor. The time is calculated based on speed and distance. Hotelling segments are defined by time, thus speed and distance cells always stay blank.

Any change in trip characteristics or fuel and feedstock selection will update the emissions and energy results part of the control. User can change functional unit by going Options → Functional Unit. The energy and emissions units can also be changed in the corresponding menus in Options.

To modify vessel engine parameters, use second tab of the control, namely "Vessel Engine Parameters". Figure 143 shows the tab.

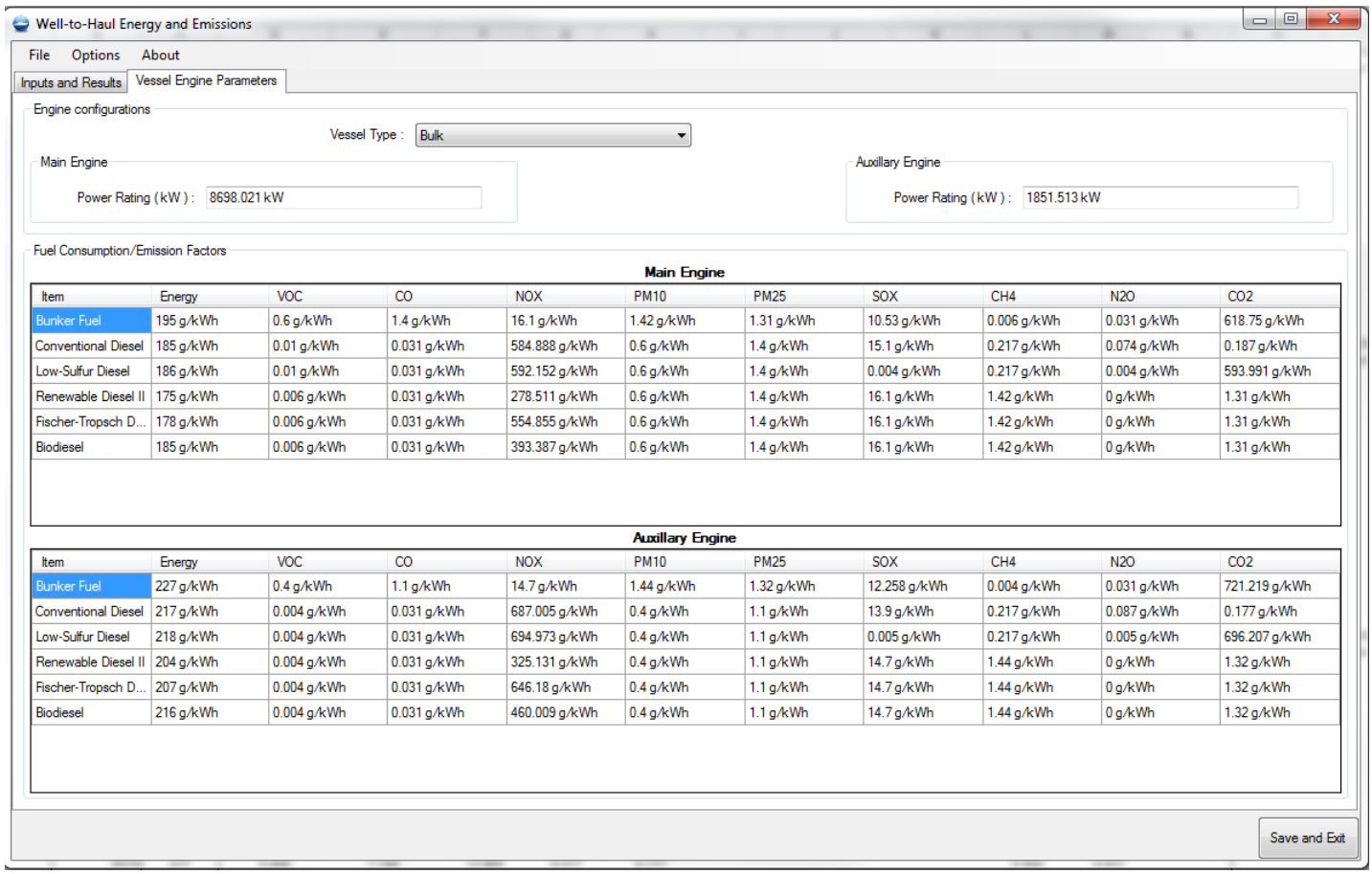


Figure 143: Vessel Engine Parameters Tab

The SOX and CO2 emissions are calculated as a result of carbon and sulfur balancing. Furthermore, the biogenic carbon is subtracted from the CO2 mass. Thus, user will see lower CO2 numbers for those fuels that contain biogenic carbon, such as Biodiesel. Changing any parameters on this tab will automatically trigger calculations and update the results on the Input and Results tab.

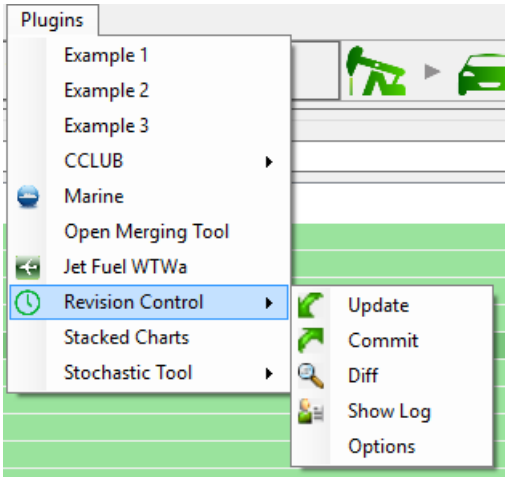
5.7 Git and SVN

The database for GREET can be saved in an XML format or as a compressed binary file. We developed a plugin that allows you to synchronize the data files with a Git or SVN repository.

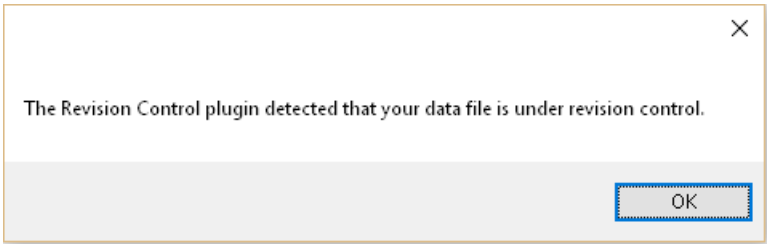
In order to do so, not much is needed. For SVN the file must be placed under revision control and the parent directory must contain a .svn folder. For GIT the file must be placed under revision control and the parent folder must contain a .git folder.

If these conditions are met, the revision control will detect that the files are under revision control and offer you to update/pull when necessary or commit/push when you are saving changes to the files. These features are automatic when loading/saving the database.

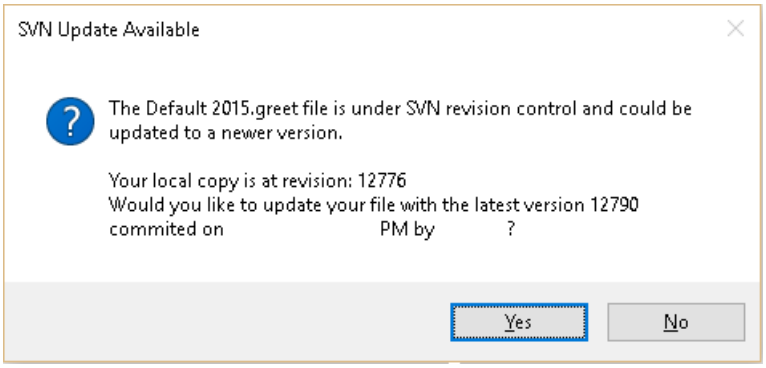
Moreover, features such as showing log, diff of the file and options are available though the plugin's menu:



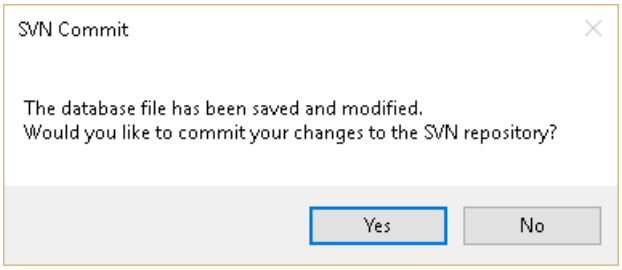
By default when you load a file in GREET.net it should detect weather the file is under revision control or not. If it is, it will display you a warning message on the first time only.

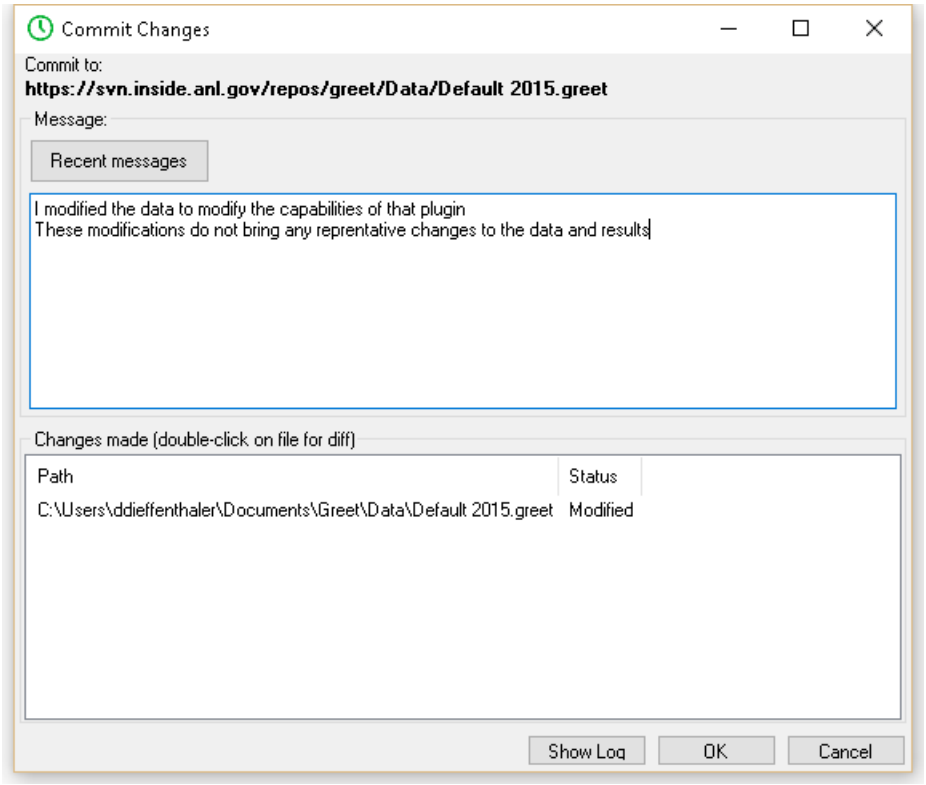


If an update is available on the remote SVN or GIT repository, a warning message will let you know that this data file can be updated. At this point you can choose to update the file by clicking "Yes" or ignore the update.



When saving your file (File-¿Save or Ctrl+S) the revision control plugin will prompt you to decide weather or not you want to commit and associate a commit message with it. For SVN the commit action will actually send your changes to the server. If you're using GIT the plugin will actually do a commit followed directly by a push to the server.





6 Troubleshooting

6.1 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.

6.2 Reporting an Issue

With GREET, you will most of the time see the Error Report window appear, if you have an error. See Figure 144. 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.

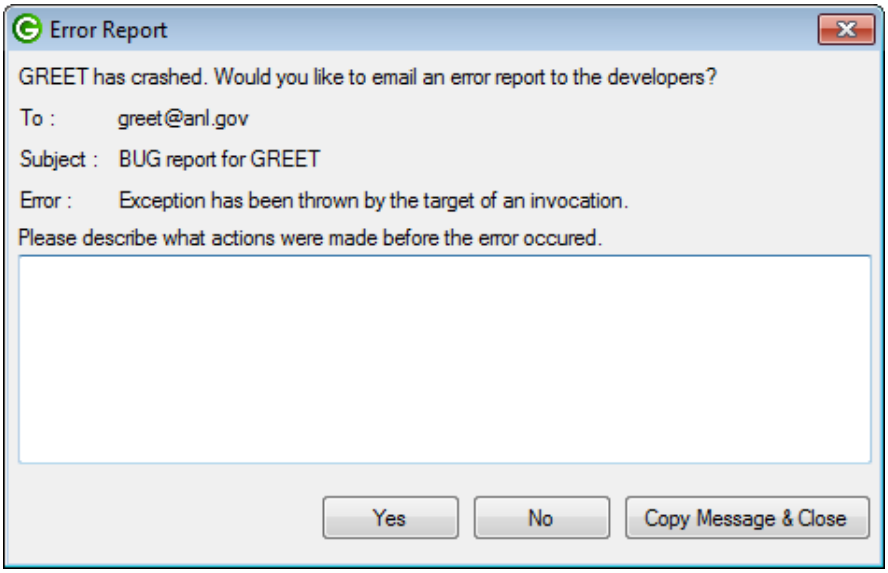


Figure 144: Error Reporting Form

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.

[2] *Life Cycle Analysis of Alternative Aviation Fuels in GREET*, Argonne National Laboratory Technical Report, 2012. Available at http://greet.es.anl.gov/publication-aviation-lca

[3] *Life Cycle Analysis of Conventional and Alternative Marine Fuels in GREETl*, Argonne National Laboratory Technical Report 2013. Available at https://greet.es.anl.gov/files/marine-fuels-13