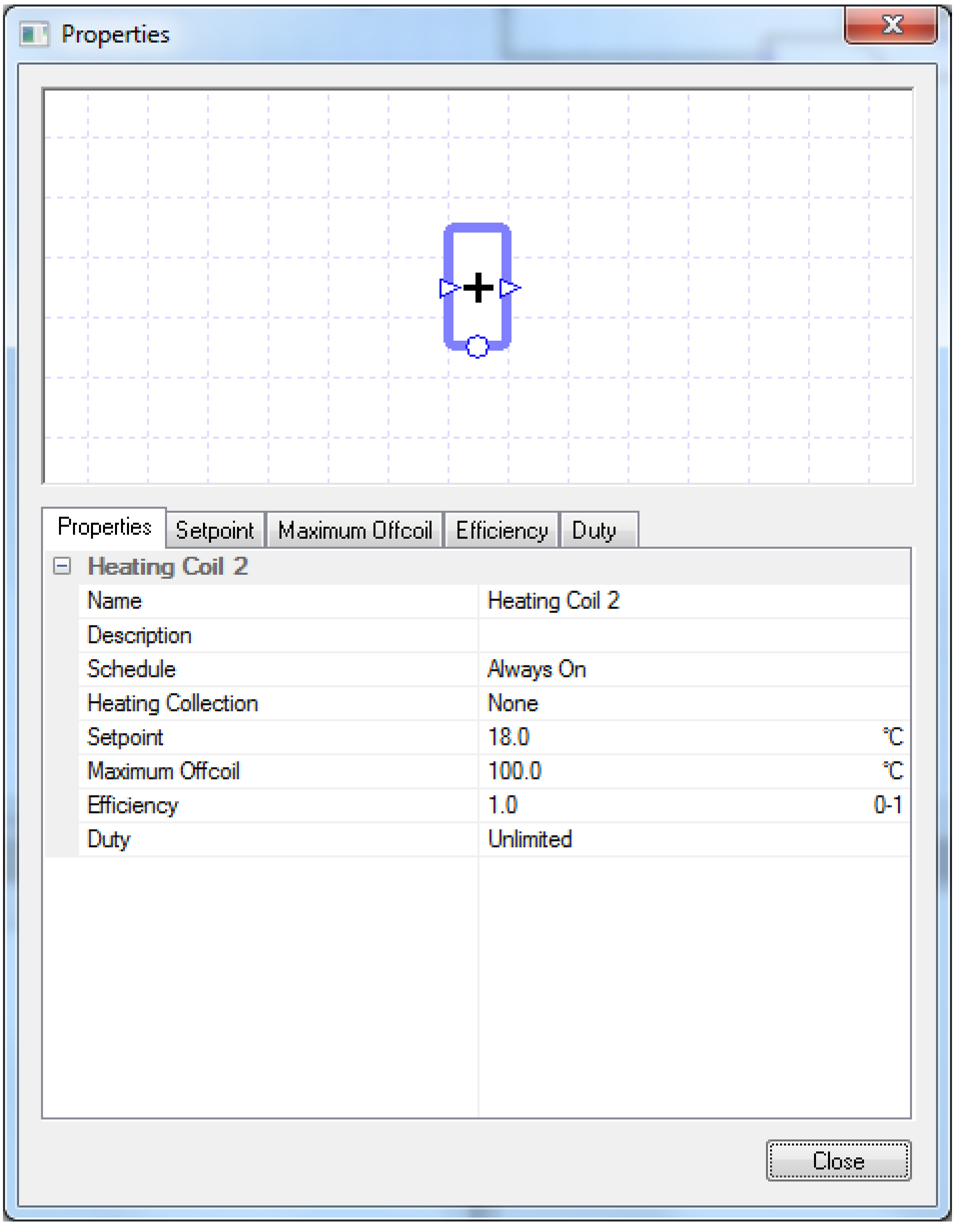
TAS Systems Component Guide

V 1.2

1 Contents

2 System Components

2.1 Heating Coil



A heating coil is used to warm up the air that passes through it. You can use a controller with the heating coil to control how the coil behaves. For a heating coil, the controller controls the coil by informing it of the amount of power the coil should provide to heat up the air. The controller does this by sending a signal, between zero and one, to the coil dictating the proportion of the coil's duty it should use to heat up the air. So, for example, if the coil received a signal of zero the coil would not warm up the air flowing through it. While if it received a signal of 1, the coil will warm up the air flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the air temperature for the Setpoint field is assumed to be directly after the coil.

Properties:

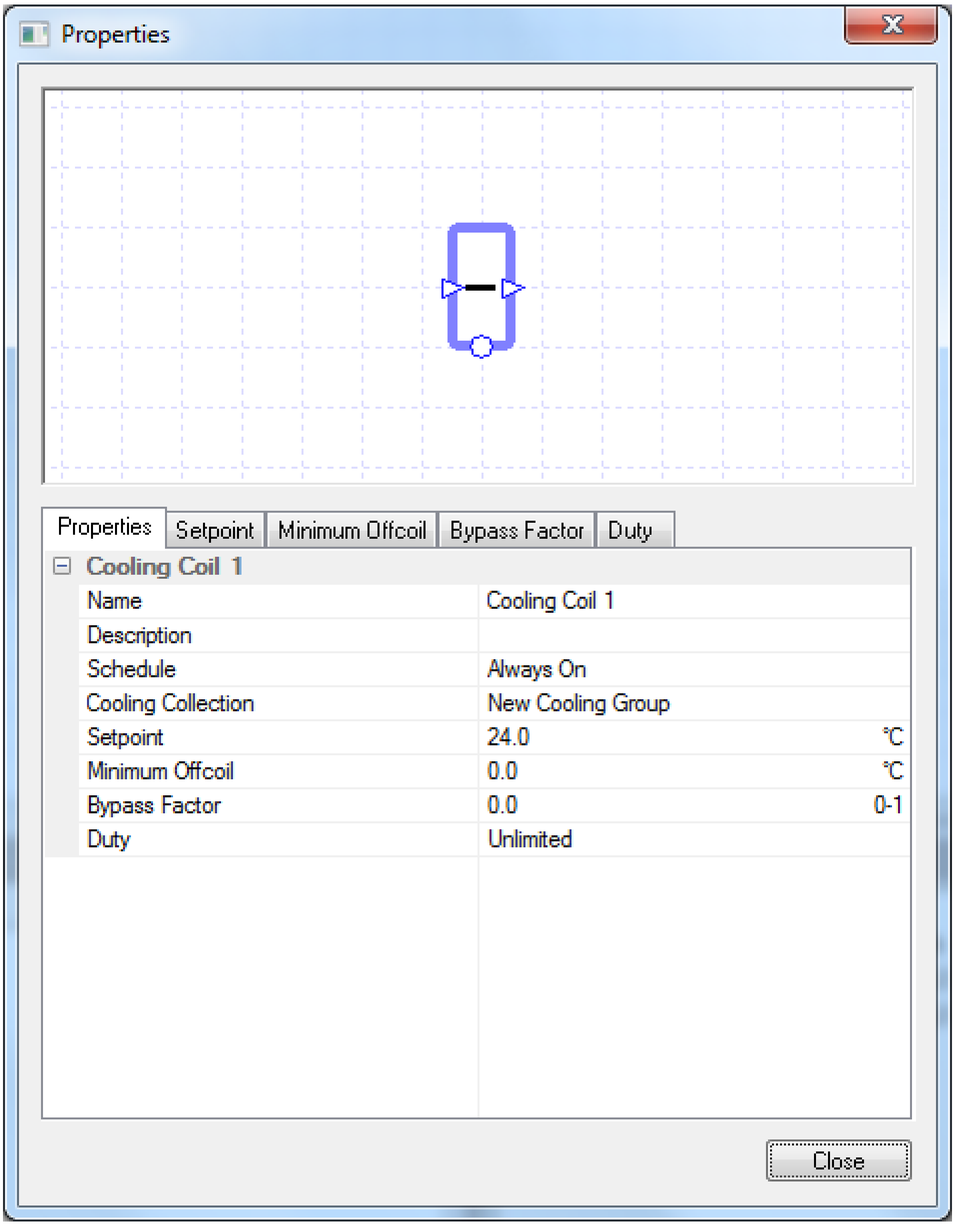
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the heating coil, this will mean that the air will flow through the coil unheated, even if there is a controller sending a non-zero signal to the coil. The default schedule option is always on, meaning that the component will operate 24/7.
- **Heating Collection –** Collections are a way of grouping components that share the same source of energy. Once a component is added to a Heating, Fuel or Electrical collection, any heating load it has will be added to that collection's heating demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Please note you will receive a warning telling you this. With the Heating Collection, you will be able to join: Heating, Fuel and Electrical groups. Your choice of group will depend on how the heat is being generated. For instance if the heat is being provided to the coil from a boiler, you would choose a Heating group but if the coil was generating the heat itself by burning fuel, you would choose a Fuel group.
- **Setpoint –** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the air going through it to reach this setpoint. In the case of the heating coil, it will warm up the air so it reaches the setpoint, but it will not be able to cool down the air to reach the setpoint. To cool the air you would need another component, for instance a cooling coil. Please note that when a controller is used in conjunction with the coil, the Setpoint field will disappear from the properties. This is done because the coil is being controlled by a controller and will heat up the air when the controller sends a signal informing the coil to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Maximum Offcoil –** The Maximum Offcoil field of the heating coil allows the user to set the maximum temperature the coil can heat the air passing through it up to, i.e. the maximum temperature of the air coming off the coil. This field comes in useful when using a controller to control the coil. As the coil will heat up the air according to the signal from the controller, it is possible that the coil will heat up the air to an undesirable temperature. Setting a maximum temperature the air can come off the coil at can stop this from happening and can also be used to control the maximum temperature the supply air can be provided at. Please note that setting the maximum offcoil temperature too low may cause unmet hours and other issues with your results, as the coil won't be able to warm up the air to a temperature required to condition the zone appropriately. Also, while there is no need to change this field from the default value when no controller is connected to the coil, if the maximum offcoil temperature is lower than the setpoint temperature the coil will only heat up the air passing through it to the offcoil temperature. Modifiers can be added to the maximum offcoil temperature using the Maximum Offcoil tab.
- **Efficiency** – The Efficiency field of a heating component only appears when a user chooses: an electrical group, a fuel group or the "None" option in the Heating Collection field; as choosing one of these options allows the user to model a component using its energy source directly at the component to produce heat. The Efficiency field allows the user to enter how efficient this process is, as a factor. This field will not have any effect on the air-side results, apart from the consumption results for this component now appearing in the results section of this component. Please note that while this field appears when the "None" option is chosen in the Heating Collection field, any loads from the component will be discarded as the Heating Collection field is set to "None". The user will be able to add a modifier to this field by going to the Efficiency tab, which also appears when the appropriate options are chosen in the Heating Collection Field.

- **Duty** - The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand. For the heating coil this would mean it wouldn't be able to heat the air to the setpoint, it would fall short. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. There are 3 options for setting the duty:
  - Unlimited – Unlimited means the component can meet any demand. Please note that this option cannot be used when a controller is attached to the component.
  - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
  - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

2.2 Cooling Coil



A cooling coil is used to cool the air that passes through it. You can use a controller with the cooling coil to control how the coil behaves. For a cooling coil, the controller controls the coil by informing it of the amount of power the coil should provide to cool down the air. The controller does this by sending a signal, between zero and one, to the coil dictating the proportion of the coil's duty it should use to cool down the air. So, for example, if the coil received a signal of zero the coil would not cool down the air flowing through it. While if it received a signal of 1 the coil will cool down the air flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the air temperature for the Setpoint field is assumed to be directly after the coil.

Properties:

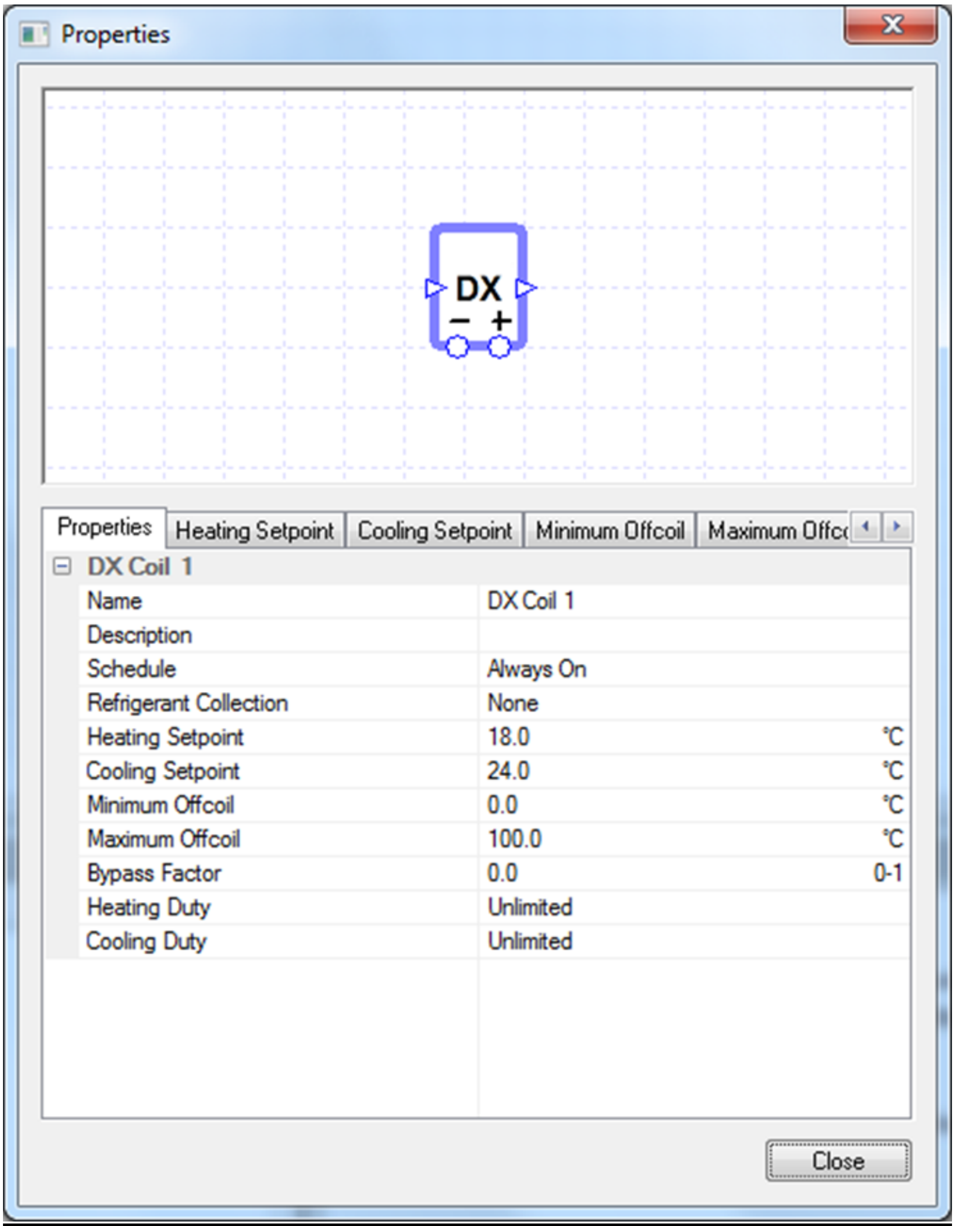
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the cooling coil, this will mean that the air will flow through the coil uncooled, even if there is a controller sending a non-zero signal to the coil. The default schedule option is always on, meaning that the component will operate 24/7.
- **Cooling Collection –** Collections are a way of grouping components that share the same source of energy. Once a component is added to a Cooling collection, any cooling load it has will be added to that collection's cooling demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results; please note you will receive a warning telling you this. Cooling coils can only join Cooling Groups.
- **Setpoint –** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the air going through it to reach the setpoint. In the case of the cooling coil, it will cool the air so it reaches the setpoint, but it will not be able to warm the air to reach this setpoint. To warm the air you would need another component, for instance a heating coil. Please note that when a controller is used in conjunction with the coil, the Setpoint field will disappear from the properties. This is done because the coil is being controlled by a controller and will cool down the air when the controller sends a signal informing the coil to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Minimum Offcoil –** The Minimum Offcoil field of the cooling coil allows the user to set the minimum temperature the coil can cool the air passing through it down to, i.e. the minimum temperature of the air coming off the coil. This field comes in useful when using a controller to control the coil. As the coil will cool down the air according to the signal from the controller, it is possible that the coil will cool down the air to an undesirable temperature. Setting a minimum temperature the air can come off the coil at can stop this from happening and can also be used to control the minimum temperature the supply air can be provided at. Please note that setting the minimum offcoil temperature too high may cause unmet hours and other issues with your results, as the coil won't be able to cool down the air to a temperature required to condition the zone appropriately. Also, while there is no need to change this field from the default value when no controller is connected to the coil, if the minimum offcoil temperature is higher than the setpoint temperature the coil will only cool down the air passing through it to the offcoil temperature. Modifiers can be added to the minimum offcoil temperature using the Minimum Offcoil tab.
- **Bypass factor –** The Bypass Factor field determines the amount of air that will bypass the coil and thus will not be cooled by the coil. The value is entered as a factor between 0 and 1 and this factor is then multiplied against the flow rate of the air, just before the coil, to determine the amount of air that will bypass the coil. Modifiers can be added to the bypass factor using the Bypass Factor tab.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand. For the cooling coil this would mean it wouldn't be able to cool the air to the setpoint, it would fall short. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. There are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.

- Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
- Value – With this option the user will type in the duty of the component.

In the Duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

2.3 DX Coil



The DX coil component allows the user to model heating and cooling of the air by refrigerants. The DX coil can be used in conjunction with two controllers, one to control the coil's heating and one to control the coil's cooling. When using a controller to control the coil's heating, otherwise known as a heating controller, please create/connect it by using the circle underneath the plus sign. Similarly with a cooling controller, please create/connect to the circle underneath the minus sign. How the signal affects the DX coil depends on what option the user chooses in the Control Method field; please see that field's section below for more details. If no controller is used with the DX coil, the sensor used to determine the air temperature for the Setpoint fields is assumed to be directly after the coil. It should be noted that if the DX coil tries to both heat and cool the air in the same hour, either due to signals from its controllers or overlapping Setpoint fields, your system will produce an error during simulation.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

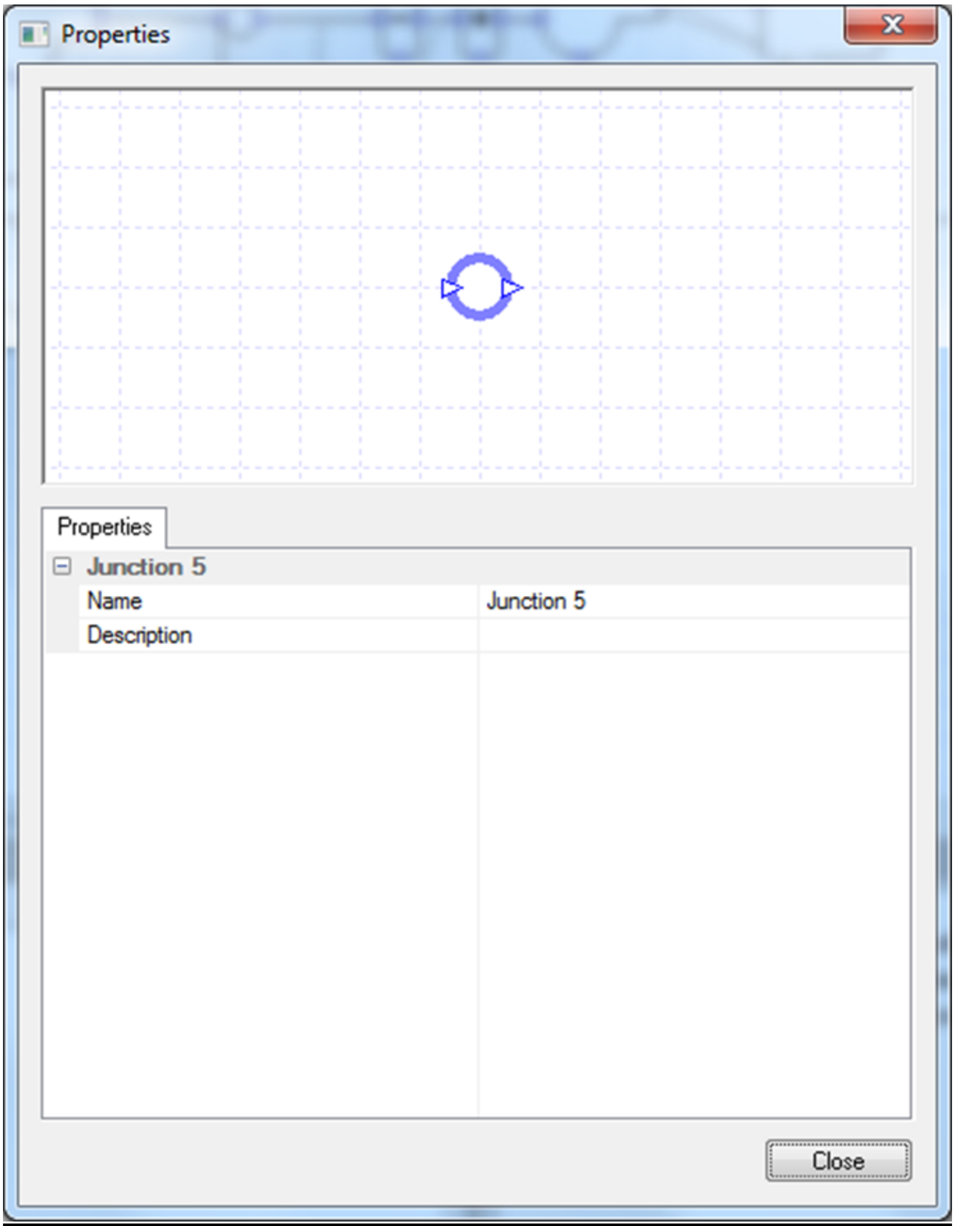
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the DX coil, this will mean that the air will flow through the coil unconditioned, even if there is a controller sending a non-zero signal to the coil. The default schedule option is always on, meaning that the component will operate 24/7.
- **Refrigerant Collection -** Collections are a way of grouping components that share the same source of energy. Once a component is added to a Refrigerant collection, any heating or cooling load it has will be added to that collection's refrigerant demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. As the DX coil uses refrigerants, you can only join refrigerant Groups.
- **Heating and cooling Setpoints –** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the air going through it to reach this setpoint. As the DX coil can provide both heating and cooling, you will have separate temperature setpoints for heating and cooling. When the air temperature is less than the heating setpoint, the DX coil will warm up the air to the setpoint. When the air temperature is above the cooling setpoint, the DX coil will cool the air down to the cooling setpoint. If the air temperature is between the heating and cooling setpoint, the DX coil will not heat or cool the air. Please note that the cooling setpoint should always be a higher temperature than the heating setpoint. If this is not the case then you will receive an overlapping limits error as otherwise your DX coil will be trying to heat and cool the air at the same time.
  Please note that when a controller is used in conjunction with the DX coil, the Setpoint fields will disappear from the properties. This is done because the DX coil's heating / cooling is being controlled by a controller. This means the DX coil will condition the air according to the signal it's sent rather than to the air temperature. It should also be noted that you cannot mix between controller and setpoint control. So, for instance, you cannot have the DX coil heating mode controlled by the Heating Setpoint field while the cooling mode is controlled by a controller. When the Setpoint fields are visible, modifiers can be added to the setpoints using the appropriate Setpoint tab.
- **Control Method –** The Control Method field only appears in the DX coil's properties list when a controller is attached to the component. It allows the user to decide how the coil will work when it receives a signal from a controller. The user can choose from the following two methods:
  - Partload – The Partload option allows the DX coil to run at a partload to meet a demand for the hour. The partload the DX coil runs at depends on the controller and the signal. The heating controller controls the coil by informing it of the amount of power the coil should provide to heat up the air. The controller does this by sending a signal, between zero and one, to the coil dictating the proportion of the coil's heating duty it should use to heat up the air. So, for example, if the coil received a signal of zero the coil would not warm up the air flowing through it. While if it received a signal of 1 the coil will warm up the air flowing through it using the maximum amount of power allowed from the Heating Duty field. Similarly the cooling controller controls the coil by informing it of the amount of power the coil should provide to cool down the air. The controller does this by sending a signal, between zero and one, to the coil dictating the proportion of the coil's cooling duty it should use to cool down the air. So, for example, if the coil received a signal of zero the coil would not cool down the air flowing through it. While if it received a signal of 1 the coil will cool down the air flowing through it using the maximum amount of power allowed from the Cooling Duty field.

  - On / Off – With this option the DX coil will only operate at full duty to meet any heating or cooling demand. The signal in this case informs the DX coil of the demand it must meet. In hours where it only needs to be partially on to meet the demand, TAS will model the unit as on at maximum for the required period and off for the rest of the hour. It will then take the average results over the hour (both the on and off period) and report them in the result section.
- **Minimum Offcoil –** The Minimum Offcoil field of the DX coil allows the user to set the minimum temperature the coil can cool the air passing through it down to, i.e. the minimum temperature of the air coming off the coil. This field comes in useful when using a controller to control the coil. As the coil will cool down the air according to the signal from the controller, it is possible that the coil will cool down the air to an undesirable temperature. Setting a minimum temperature the air can come off the coil at can stop this from happening and can also be used to control the minimum temperature the supply air can be provided at. Please note that setting the minimum offcoil temperature too high may cause unmet hours and other issues with your results, as the coil won't be able to cool down the air to a temperature required to condition the zone appropriately. Also, while there is no need to change this field from the default value when no controller is connected to the coil, if the minimum offcoil temperature is higher than the setpoint temperature the coil will only cool down the air passing through it to the offcoil temperature. Modifiers can be added to the minimum offcoil temperature using the Minimum Offcoil tab.
- **Maximum Offcoil –** The Maximum Offcoil field of the DX coil allows the user to set the maximum temperature the coil can heat the air passing through it up to, i.e. the maximum temperature of the air coming off the coil. This field comes in useful when using a controller to control the coil. As the coil will heat up the air according to the signal from the controller, it is possible that the coil will heat up the air to an undesirable temperature. Setting a maximum temperature the air can come off the coil at can stop this from happening and can also be used to control the maximum temperature the supply air can be provided at. Please note that setting the maximum offcoil temperature too low may cause unmet hours and other issues with your results, as the coil won't be able to warm up the air to a temperature required to condition the zone appropriately. Also, while there is no need to change this field from the default value when no controller is connected to the coil, if the maximum offcoil temperature is lower than the setpoint temperature the coil will only heat up the air passing through it to the offcoil temperature. Modifiers can be added to the maximum offcoil temperature using the Maximum Offcoil tab.
- **Bypass factor –** When the DX coil is cooling the air, the bypass factor helps to determine the amount of air that will bypass the coil and thus will not be heated or cooled by the coil. The value is entered as a factor between 0 and 1 and this factor is then multiplied against the flow rate of the air just before the coil to determine the amount of air that will bypass the coil. Modifiers can be added to the bypass factor using the Bypass Factor tab.
- **Heating and Cooling duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand**.** In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Please note that for a DX Coil, the heating and cooling duties are set separately. This allows you to model a refrigerant coil that just provides heating or cooling by setting the other's duty to zero. In TAS Systems, there are 3 options for the duty:
  - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.

- Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
- Value – With this option the user will type in the duty of the component.

In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

2.4 Junction

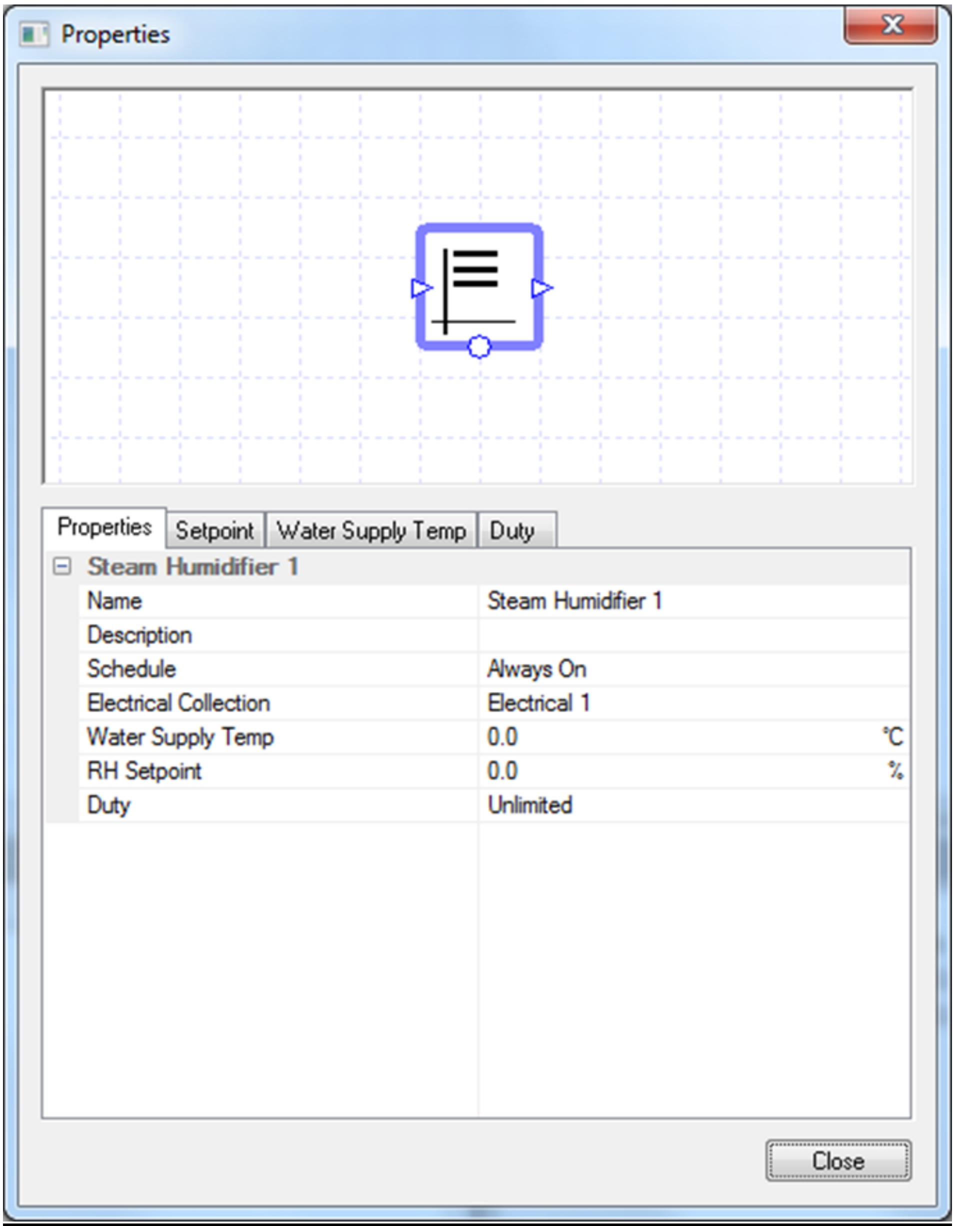


The Junction has two uses. The first use is to model where the fresh air enters the system and where the exhaust air leaves the system. In these cases the junction will only have one duct connected to it and it will turn blue, to indicate it is an external junction.
The second use of a junction is to split or merge air paths. When being used to split up an air path, the user may need to use a damper or fan on the new air paths to set the design flow rates along these new paths. You cannot use controllers with junctions.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

2.5 Steam Humidifier



A steam humidifier increases the humidity of the air passing through it by adding steam to it. Please note that this process humidifies the air while keeping the air temperature constant. You can use a controller with the steam humidifier to control how the humidifier behaves. For a steam humidifier, the controller controls the humidifier by informing it of the amount of power the humidifier should provide to humidify the air. The controller does this by sending a signal, between zero and one, to the humidifier dictating the proportion of the humidifier's duty it should use to humidify the air. So, for example, if the humidifier received a signal of zero the humidifier would not humidify the air flowing through it. While if it received a signal of 1 the humidifier will humidify the air flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the relative humidity for the Setpoint field is assumed to be directly after the Humidifier.

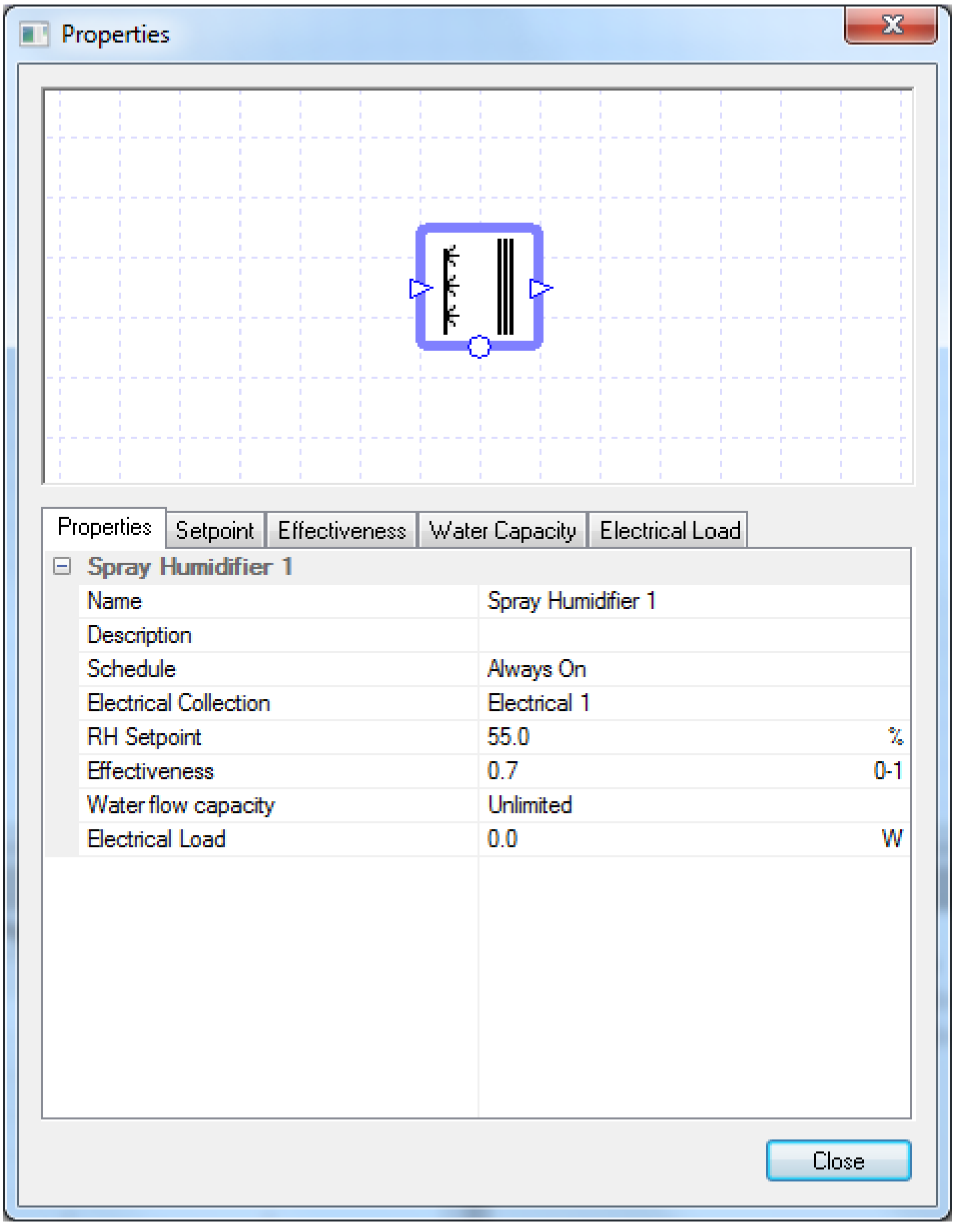
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the steam humidifier, this will mean that the air will flow through the humidifier without being humidified, even if there is a controller sending a non-zero signal to the humidifier. The default schedule option is always on, meaning that the component will operate 24/7.
- **Electrical Collection -** Collections are a way of grouping components that share the same source of energy. Once a component is added to an electrical collection, any electrical load it has will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. The Steam Humidifier component in TAS Systems can only join Electrical Groups.
- **Water Supply Temperature –** The Water Supply Temperature field allows the user to specify the temperature of the water being supplied to the humidifier. Please note that as the supply water temperature increases, the energy demand of the steam humidifier decreases. This is because less energy is required to produce the steam. Modifiers can be added to the water supply temperature using the Water Supply Temp tab.
- **Relative Humidity (RH) Setpoint -** When a Relative Humidity (RH) is entered into the RH setpoint field, the component will increase the RH of the air to this setpoint. If the air's RH is greater than the RH setpoint, there will be no change in the humidity of the air as the steam humidifier does not dehumidify the air. Please note that when a controller is used in conjunction with the steam humidifier, the Setpoint field will disappear from the properties. This is done because the steam humidifier is being controlled by a controller and will humidify the air when the controller sends a signal informing the humidifier to do so. When the RH Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand**.** For a humidifier this would mean it wouldn't be able to increase the RH of the air to the setpoint, it would fall short. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. There are 3 options for setting the duty.
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

2.6 Spray Humidifier



A spray humidifier increases the humidity of the air by spraying water into it. The spray humidifies the air by evaporative cooling, meaning the spray humidifier humidifies along lines of constant enthalpy on the Psychrometric Chart. You can use a controller with the spray humidifier to control how the humidifier behaves. For a spray humidifier, the controller controls the humidifier by informing it of the amount of water it should add to the air to humidify it. The controller does this by sending a signal, between zero and one, to the humidifier dictating the proportion of the humidifier's water flow capacity it should use as the water flow rate to humidify the air. So, for example, if the humidifier received a signal of zero the humidifier would not humidify the air flowing through it. While if it received a signal of 1, the humidifier will humidify the air flowing through it by adding the maximum amount of water allowed from the Water Flow Capacity field. If no controller is used, the sensor used to determine the relative humidity for the RH Setpoint field is assumed to be directly after the humidifier.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the spray humidifier, this will mean that the air will flow through the humidifier without being humidified, even if there is a controller sending a non-zero signal to the humidifier. The default schedule option is always on, meaning that the component will operate 24/7.
- **Electrical Collection -** Collections are a way of grouping components that share the same source of energy. Once a component is added to an electrical collection, any electrical load it has will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. The Spray Humidifier component in TAS Systems can only join Electrical Groups.
- **Relative Humidity (RH) Setpoint -** When a relative humidity is entered into the RH Setpoint field, the component will increase the RH of the air to this setpoint. If the air's RH is greater than the RH setpoint, there will be no change in the humidity of the air as the spray humidifier does not dehumidify the air. Please note that when a controller is used in conjunction with the humidifier, the Setpoint field will disappear from the properties. This is done because the humidifier is being controlled by a controller and will humidify the air when the controller sends a signal informing the humidifier to do so. When the RH Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Effectiveness** – This option allows the user to input how effective the spray humidifier is. The effectiveness needs to be entered as a factor between 0 and 1. The effectiveness of the spray humidifier can be found using the following formula:

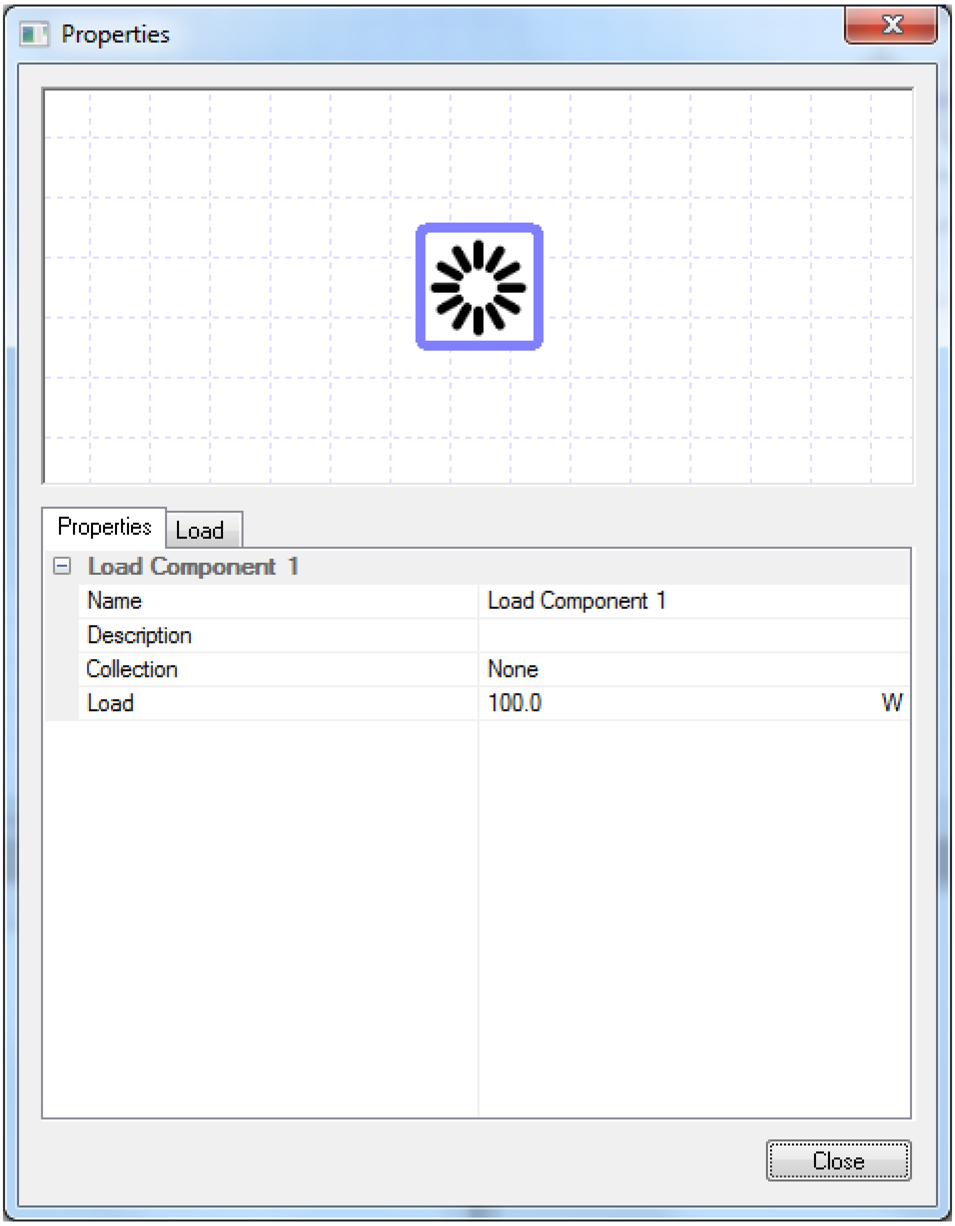
$$\boldsymbol{Effectiveness = \frac{T_{in} - T_{out}}{T_{in} - T_{sat}} = \frac{H_{out} - H_{in}}{H_{sat} - H_{in}}},$$

  where $\boldsymbol{T}$ is the air temperature and $\boldsymbol{H}$ is the humidity ratio. Please note that the subscript on each letter denotes where that measurement is taken. A subscript $\boldsymbol{in}$ denotes that the measurement is taken on the air entering the Spray Humidifier; a subscript $\boldsymbol{out}$ denotes that the measurement is taken on the air exiting the spray humidifier and the subscript $\boldsymbol{sat}$ denotes the temperature or humidity ratio of when the air would be saturated when following the line of constant enthalpy. In the Effectiveness tab you are able to apply a modifier to this field.
- **Water Flow Capacity** – The Water Flow Capacity field allows the user to enter the Water Flow Capacity of the Steam Humidifier. If the water flow capacity is set too low, the humidifier will not be able to humidify the air to the setpoint. In TAS Systems there are 3 options to choose from to set the water flow capacity:
    - Unlimited – This means there is no limit on the water flow, meaning the humidifier can spray as much water into the air as it needs to. Please note that this option cannot be used when a controller is connected to the component.
    - Value – The user can enter a value to provide a maximum limit to the water flow.
    - Sized - Allows the user to size the Water Flow Capacity on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.

  In the Water Flow Capacity tab, you will have these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Electrical Load –** The Electrical Load field of the spray humidifier allows the user to account for any electrical load in the exchanger, for instance the load associated with spraying the water into the air stream. This load is then passed onto the plant room collection, which is chosen in the Electrical Collection field. The user can type in a value into this field and the

component will pass this amount of electrical load onto the plant room collection for each hour of the simulation. As it is very unlikely that the user will want to pass on the same load for every hour of the simulation, it is recommended that the user goes to the Electrical Load tab and creates a modifier for this field. With the various modifiers available, the user will be able to vary the loads throughout the simulation.

2.7 Load Component



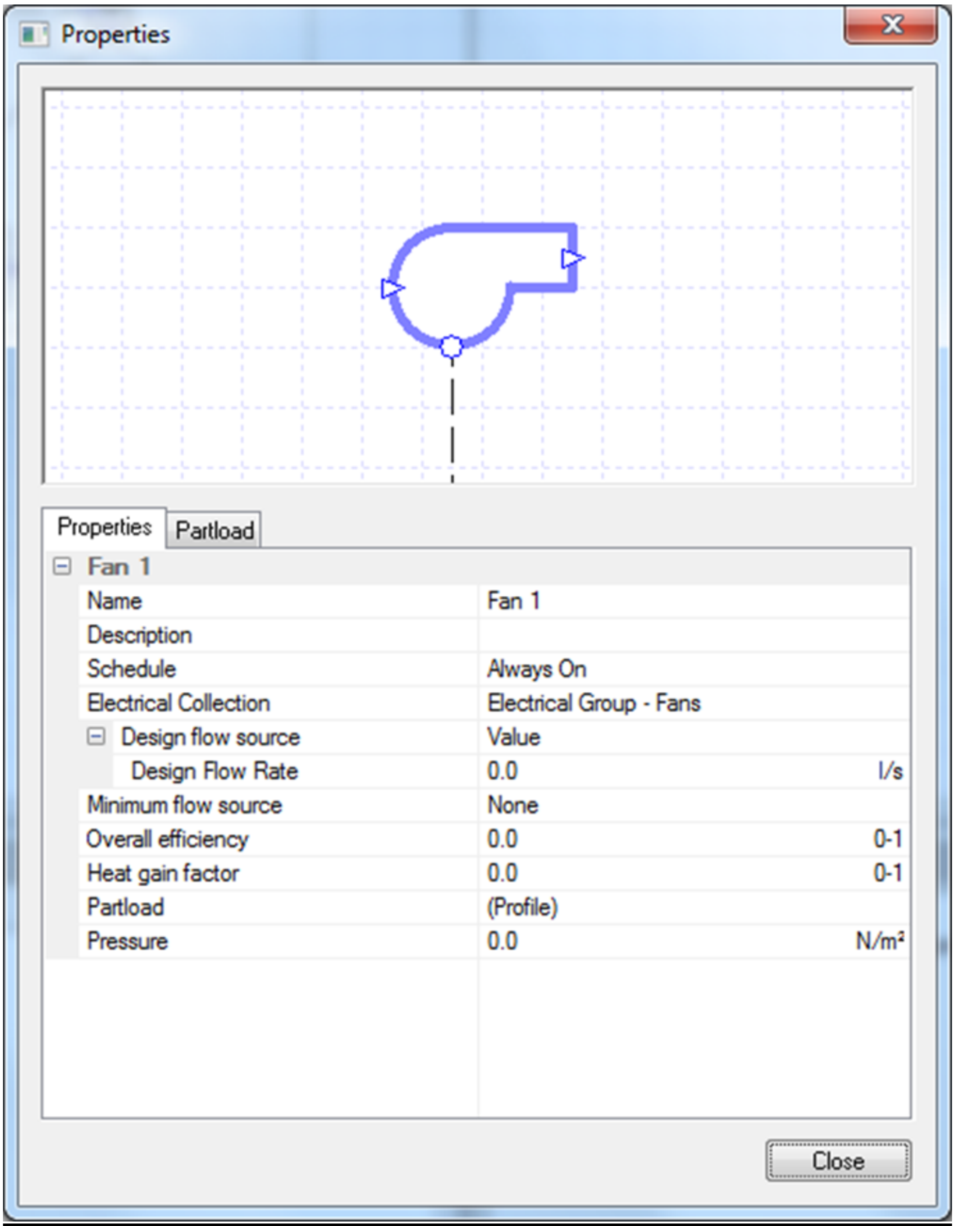
The Load component is a stand-alone component that allows the user to model additional loads that will not be modelled as a part of their air side system. The additional load modelled by the component is then added to the plant room collection chosen in the Collection field, so that the software can include it in the energy consumption calculations. As the Load component is a stand-alone component, it has no ports on it to connect up to air-side systems and also cannot be used with controllers.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Collection –** In the Collection field, the user will be able to choose one of the plant room collections they have created in their plant room system. When a collection is chosen, any load entered into this component for the hour will be passed onto the collection and will be reported in the collection's demand.

- **Load –** The Load field allows the user to enter in the amount of load they would like the component to pass onto the plant room collection. The user can type in a value into the field and the component will pass this amount of load onto the plant room collection for each hour of the simulation. As it is very unlikely that the user will want to pass on the same load for every hour of the simulation, it is recommended that the user goes to the Load tab and creates a modifier for this field. With the various modifiers available, the user will be able to vary the loads throughout the simulation.

2.8 Fan



The Fan component in TAS Systems is one of the foundations of any system that requires air flow through ducts, as it is the fan component that provides this air flow. Due to this, any system that cannot be represented by an unconnected zone component will need at least one Fan component within the system. Controllers can be used with the Fan component, and they will control how much pressure the fan will exert on the air flowing through it. The signal received by the fan dictates the proportion of the maximum pressure increase the fan will increase the pressure to. Please note that this maximum pressure increase is not the value entered in the fan's Pressure field. Instead the software will work the maximum pressure increase out by taking into account the Pressure field of the fan, along with the design flow rates and design pressure drops of other components around the system. Also, as pressure and flow rate are linked, controlling the pressure with a controller will also mean that the flow rate will vary. If it is only the fans, and not the dampers, with controllers attached, how the signal received by the fan affects the flow rate is shown using the equation below:

$$\textbf{Flow Rate at Signal} = \textbf{Design Flow Rate} * \sqrt{\textbf{Signal}}.$$

Please note that fans and dampers, being controlled by different controllers, can be used in conjunction to model certain systems; for instance a VAV system. In this instance the fan's controller will control the pressure increase through the fan while the damper's controller will control the capacity through the damper. The flow rate in this instance will not be controlled by either controller on its own; it will be the result of the pressures and capacities around the system. Please see the examples in the TAS Systems Controller guide for more on how to do this.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the fan, this will mean that the fan will not provide any air flow. If all fans in your system are scheduled to be off in the same hour then there will be no air flow in your system. The default schedule option is always on, meaning that the component will operate 24/7.
- **Electrical Collection -** Collections are a way of grouping components that share the same source of energy. Once a component is added to an electrical collection, any electrical load it has will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Fans in TAS Systems can only join Electrical groups.
- **Design Flow Source –** The Design Flow Source option allows the user to choose how they wish to set the design flow rate. The design flow rate, along with the other design values, are used during the design flow calculation to fix the capacities of the flow components (fans, dampers and air-side economisers) without a fixed capacity. When modelling a Constant Air Volume system, this design flow calculation will lead to the design flow rate being the flow rate of the air path the component is on. When modelling a Variable Air Volume system, the design flow calculation will lead to the design flow rate tending to be the maximum allowed flow rate for the air path the component is on. However, there can be instances, where the max flow rate will exceed the design flow rate. Please note that it is recommended that all flow components (fans, dampers and air-side economisers) on the same air path have the same design flow rate, as setting different design flow rates will cause inconsistent design flow rate errors. In TAS Systems, the user has the following options for the design flow source.
    - None – Upon choosing this option the user will be asked to enter the capacity of the fan.
    - Value – Upon choosing this option the user will be asked to enter the design flow rate.
    - All Attached Zones Flow Rate – With this option TAS will add up the flow rate of every zone in this system connected to the fan by ducts and use this as the design flow rate.
    - All Attached Zones Fresh Air – With this option TAS will add up the fresh air rate of every zone in the system connected to the fan by ducts and use this as the design flow rate.
    - All Attached Zones Sized – With this option, TAS will allow you to use sizing methods to calculate the design flow rate. Each method below is run on every zone in this system connected to the fan by ducts. The software then sums the sized flow rate for each zone to produce the design flow rate. The seven sizing methods available are:
        - Per Floor Area – Here TAS Systems asks the user to enter a value in $l/s/m^2$ (cfm / ft^2 in U.S. customary units). This value is then multiplied by the area of

the zone to give the flow rate in l/s (cfm in U.S. customary units). Please note that the user can use the size fraction field to over-size or under-size this value.

- Per Volume – Here TAS Systems asks the user to enter a value in l/s/m^3 (cfm / ft^3 in U.S. customary units). The value is then multiplied by the volume of the zone to give the flow rate. Please note that the user can use the size fraction field to over-size or under-size this value.
- ACH – The Value field here stands for how many air changes per hour the user wishes to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person – TAS Systems asks the user to enter an air rate per person. TAS then uses the following formula to give the sized flow rate:

$$\textbf{\textit{Sized Flow Rate}} = \frac{\textbf{\textit{Peak Occupancy Sensible Gain}} + \textbf{\textit{Peak Occupancy Latent Gain}}}{\textbf{\textit{Metabolic Rate}}} * \textbf{\textit{Area of zone}} * \textbf{\textit{Value}},$$

  where the: Peak Occupancy Sensible Gain, Peak Occupancy Latent Gain and Metabolic Rate are all read from the zone's internal condition; while Value is the value entered in Systems for the air rate per person. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person and Area – The Peak Person and Area method allows the user to size the flow rate on both the Peak Person and Per Floor Area methods. The user is asked to enter in their flow rates per person and per floor area. TAS then uses the value entered into the Per Person field in the Peak Person method and the value entered into the Per Area field in the Per Floor Area method. TAS will then add the results of both of these methods together and report the sum as the flow rate.
- Peak Internal Condition – Please note that the Peak Internal Condition method works like the Peak Person method, but instead of asking the user for the air rate per person it uses the Outside Air rate from the zone's Internal Condition instead. Please note that the user can use the size fraction field to over-size or under-size this value.
- Delta T – The Delta T method is the default method for sizing the flow rate for a zone. Upon choosing this method, the user is asked to enter a heating delta T and a cooling delta T. The heating delta T value entered should be:

$$\textbf{\textit{Max Temp of the supply air}} - \textbf{\textit{The zone's thermostat lower limit}},$$

  where the zone's thermostat is read from the Internal Condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air heated to this maximum temperature to keep the zone's temperature above the lower limit. Please note that this is done using the heating loads from the TSD file, as this sizing is done before the simulation. The cooling delta T works in a similar way with the zone's thermostat upper limit. The cooling delta T value entered should be equal to:

$$\textbf{\textit{The Zone's thermostat upper limit}} - \textbf{\textit{Min temp of the supply air}},$$

  where the zone's thermostat is read from the internal condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air cooled to this minimum temperature to keep the zone's temperature below the upper limit. Please note that this is done using the cooling loads from the TSD file, as this sizing is done before the simulation. Upon working out the flow rate needed for the heating delta T and the cooling delta T, TAS will take the max of the two as the flow rate for the zone. Please note that the values entered here do not impact the supply air temperatures. It is just used to work out the sized flow rate.

After entering the delta T values, the user will be asked to choose the design condition (or just the simulation data) to choose what days to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

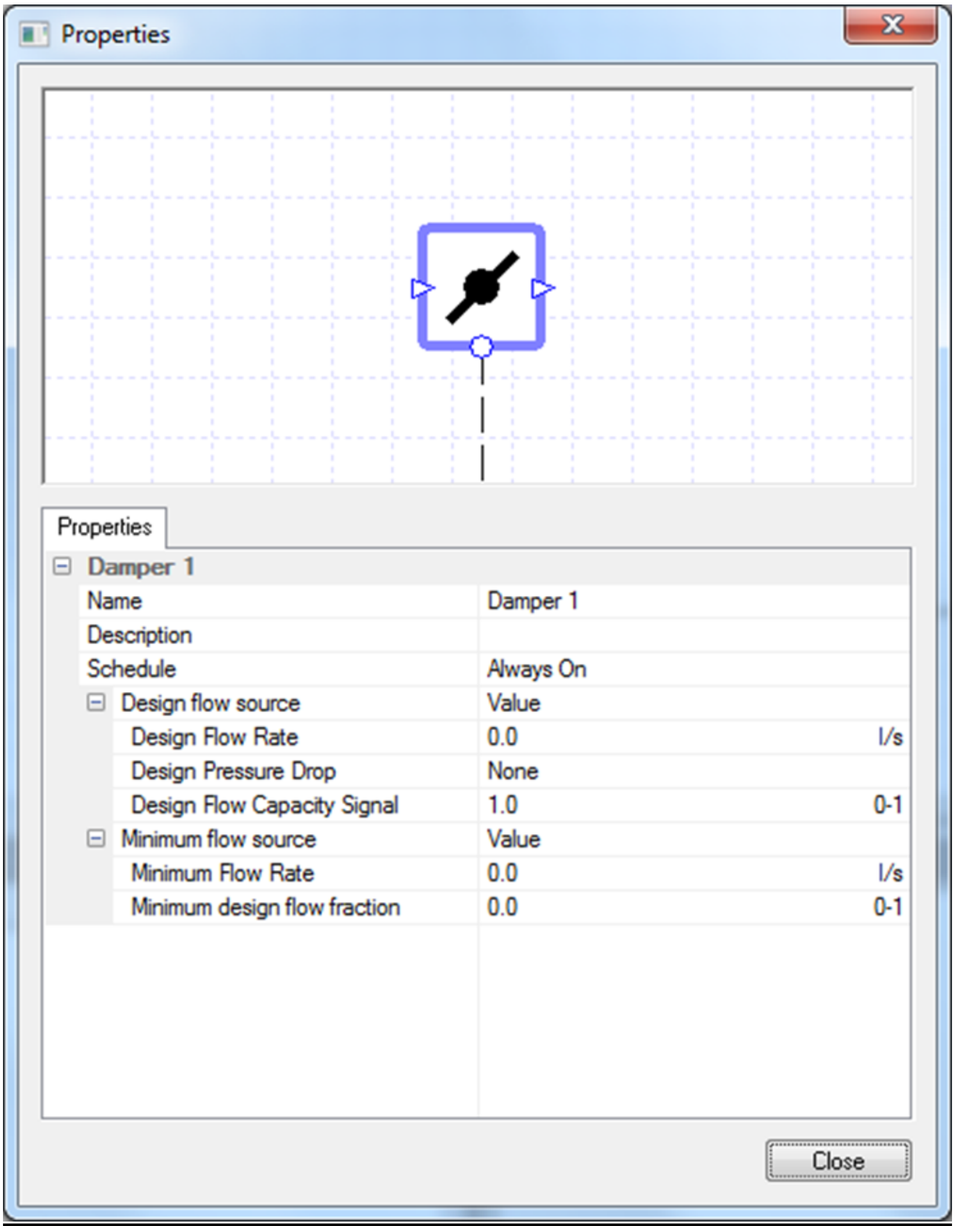
- Nearest Zone Flow Rate – With this option TAS will take the flow rate of the nearest zone to the component and use this as the design flow rate.
- Nearest Zone Fresh Air – With this option TAS takes the fresh air rate of the nearest zone to the component and uses this as its design flow rate.
- Nearest Zone Sized - With this option, TAS will allow you to use sizing methods to calculate the design flow rate. The selected sizing method will be run on the zone nearest to the component and the resulting sized value will be the design flow rate. Please note that the seven sizing methods available have already been discussed in the All Attached Zones Sized option above.

- **Minimum Flow Source –** The Minimum Flow Source option only appears when a controller is attached to the fan, as this option is only required when the flow rate is varying. The Minimum Flow Source option allows the user to choose how they wish to set the minimum flow rate of the system. The Minimum Flow Source option gives the same eight options as the Design Flow Source option but instead sets the Minimum Flow Rate field. Please note there are some differences between the two:
  - The 'none' option does not give you any other options. It means that the flow rate can decrease as needed.
  - The 'All Attached Zones Sized' and 'Nearest Zone Sized' methods come with three additional sizing methods: Hourly Person, Hourly Person and Area, and Hourly Internal Condition. These sizing methods work like the peak methods but are calculated each hour, allowing the user to model systems with a variable fresh air requirement based on occupancy.
  - All options, apart from the 'none' option, allow for a minimum design flow fraction to be set. The minimum design flow fraction allows the user to set the minimum flow rate as a fraction of the design flow rate. Please note that if the minimum design flow fraction and the minimum flow rate both have values in them, TAS will take the maximum of the two as the minimum flow rate. It will also enter the maximum into the Minimum Flow Rate field after a simulation.

  To ensure that the minimum flow rate set here is met, TAS works out the signal the fan requires to meet this flow rate and then sets it as the minimum signal of all attached controllers, ensuring the flow rate doesn't drop below this level. It is recommended that all fans on the same air path have the same minimum flow rate.
- **Overall Efficiency –** The overall efficiency of the fan is the product of its: motor efficiency, electrical efficiency, belt efficiency and aerodynamic efficiency (as there will be aerodynamic losses in the fan). Please note that this does not include any partload efficiency, which is set by the Partload field. The user will need to enter the overall efficiency as a factor between zero and one.
- **Heat Gain Factor –** The heat gain factor of the fan is the proportion of energy used to power the fan that is converted to heat. The generated heat will warm up the air that passes through the fan.
- **Partload –** Clicking on the Partload field will take you to the Partload tab. From this tab, you will be able to edit the Partload profile of the component by using the graph or the table. To see how to edit the Partload profiles, please watch the Profiles video in the TAS Systems User Guide.
- **Pressure –** The Pressure field allows the user to enter the fan pressure. The formula for fan pressure is:

$$\boldsymbol{Fan\ Pressure\ =\ SFP\ *\ 1000 * Overall\ Efficiency}\ .$$

2.9 Damper



The Damper component in Systems is used to regulate the air flow. They are mainly used after splitting up air paths to modulate the air flow on each path. Controllers can be used with the damper, and they will control the capacity of the damper. The signal the component receives dictates the proportion of the maximum capacity that the damper will close to, where the maximum capacity is set by either the Design Flow Rate field or the Capacity field of the damper. So for example, if the damper receives a signal of 1 from the controller, the damper will open up and allow air through it according to its maximum capacity. If the damper receives a signal of zero from the controller, the damper will close completely and will not allow any air to pass through it. It should be noted that as the capacity of the damper varies, so will the flow rate and pressure of the system. Please note that Fans and Dampers, being controlled by different controllers, can be used in conjunction to model certain systems; for instance a VAV system. In this instance the fan's controller will control the pressure increase through the fan while the damper's controller will control the capacity through the damper. The flow rate in this instance will not be controlled by either controller on its own; it will be the result of the pressures and capacities around the system. Please see the examples in the TAS Systems Controller guide for more on how to do this.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the damper, this will mean that the damper will close and will not allow any air to flow through it. This will mean any other component on the same air path as the damper will not receive any air flow in this hour. The default schedule option is always on, meaning that the component will operate 24/7.
- **Design Flow Source –** The Design Flow Source option allows the user to choose how they wish to set the design flow rate. The design flow rate, along with the other design values, are used during the design flow calculation to fix the capacities of the flow components (fans, dampers and air-side economisers) without a fixed capacity. When modelling a Constant Air Volume system, this design flow calculation will lead to the design flow rate being the flow rate of the air path the component is on. When modelling a Variable Air Volume system, the design flow calculation will lead to the design flow rate tending to be the maximum allowed flow rate for the air path the component is on. However, there can be instances, where the max flow rate will exceed the design flow rate. Please note that it is recommended that all flow components on the same air path have the same design flow rate, as setting different design flow rates will cause inconsistent design flow rate errors. In TAS Systems, the user has the following options for the design flow source.
    - None – Upon choosing this option the user will be asked to enter the capacity of the flow.
    - Value – Upon choosing this option the user will be asked to enter the design flow rate.
    - All Attached Zones Flow Rate – With this option TAS will add up the flow rate of every zone in this system connected to the damper by ducts and use this as the design flow rate.
    - All Attached Zones Fresh Air – With this option TAS will add up the fresh air rate of every zone in the system connected to the damper by ducts and use this as the design flow rate.
    - All Attached Zones Sized – With this option, TAS will allow you to use sizing methods to calculate the design flow rate. Each method below is run on every zone in this system connected to the damper by ducts. The software then sums the sized flow rate for each zone to produce the design flow rate. The seven sizing methods available are:
        - Per Floor Area – Here TAS Systems asks the user to enter a value in $l/s/m^2$ (cfm / ft^2 in U.S. customary units). This value is then multiplied by the area of the zone to give the flow rate in l/s (cfm in U.S. customary units). Please note that the user can use the size fraction field to over-size or under-size this value.
        - Per Volume – Here TAS Systems asks the user to enter a value in $l/s/m^3$ (cfm / ft^3 in U.S. customary units). The value is then multiplied by the volume of the zone to give the flow rate. Please note that the user can use the size fraction field to over-size or under-size this value.
        - ACH – The Value field here stands for how many air changes per hour the user wishes to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

- Peak Person – TAS Systems asks the user to enter an air rate per person. TAS then uses the following formula to give the sized flow rate:

$$\textit{Sized Flow Rate} = \frac{\textit{Peak Occupancy Sensible Gain} + \textit{Peak Occupancy Latent Gain}}{\textit{Metabolic Rate}} * \textit{Area of zone} * \textit{Value},$$

  where the: Peak Occupancy Sensible Gain, Peak Occupancy Latent Gain and Metabolic Rate are all read from the zone's internal condition; while Value is the value entered in Systems for the air rate per person. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person and Area – The Peak Person and Area method allows the user to size the flow rate on both the Peak Person and Per Floor Area methods. The user is asked to enter in their flow rates per person and per floor area. TAS then uses the value entered into the Per Person field in the Peak Person method and the value entered into the Per Area field in the Per Floor Area method. TAS will then add the results of both of these methods together and report the sum as the flow rate.
- Peak Internal Condition – Please note that the Peak Internal Condition method works like the Peak Person method, but instead of asking the user for the air rate per person it uses the Outside Air rate from the zone's Internal Condition instead. Please note that the user can use the size fraction field to over-size or under-size this value.
- Delta T – The Delta T method is the default method for sizing the flow rate for a zone. Upon choosing this method, the user is asked to enter a heating delta T and a cooling delta T. The heating delta T value entered should be:

$$\textit{Max Temp of the supply air} - \textit{The zone's thermostat lower limit},$$

  where the zone's thermostat is read from the Internal Condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air heated to this maximum temperature to keep the zone's temperature above the lower limit. Please note that this is done using the heating loads from the TSD file, as this sizing is done before the simulation. The cooling delta T works in a similar way with the zone's thermostat upper limit. The cooling delta T value entered should be equal to:

$$\textit{The Zone's thermostat upper limit} - \textit{Min temp of the supply air},$$

  where the zone's thermostat is read from the internal condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air cooled to this minimum temperature to keep the zone's temperature below the upper limit. Please note that this is done using the cooling loads from the TSD file, as this sizing is done before the simulation. Upon working out the flow rate needed for the heating delta T and the cooling delta T, TAS will take the max of the two as the flow rate for the zone. Please note that the values entered here do not impact the supply air temperatures. It is just used to work out the sized flow rate.

  After entering the delta T values, the user will be asked to choose the design condition (or just the simulation data) to choose what days to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

- Nearest Zone Flow Rate – With this option TAS will take the flow rate of the nearest zone to the component and use this as the design flow rate.
- Nearest Zone Fresh Air – With this option TAS takes the fresh air rate of the nearest zone to the component and uses this as its design flow rate.
- Nearest Zone Sized - With this option, TAS will allow you to use sizing methods to calculate the design flow rate. The selected sizing method will be run on the zone nearest to the component and the resulting sized value will be the design flow rate.

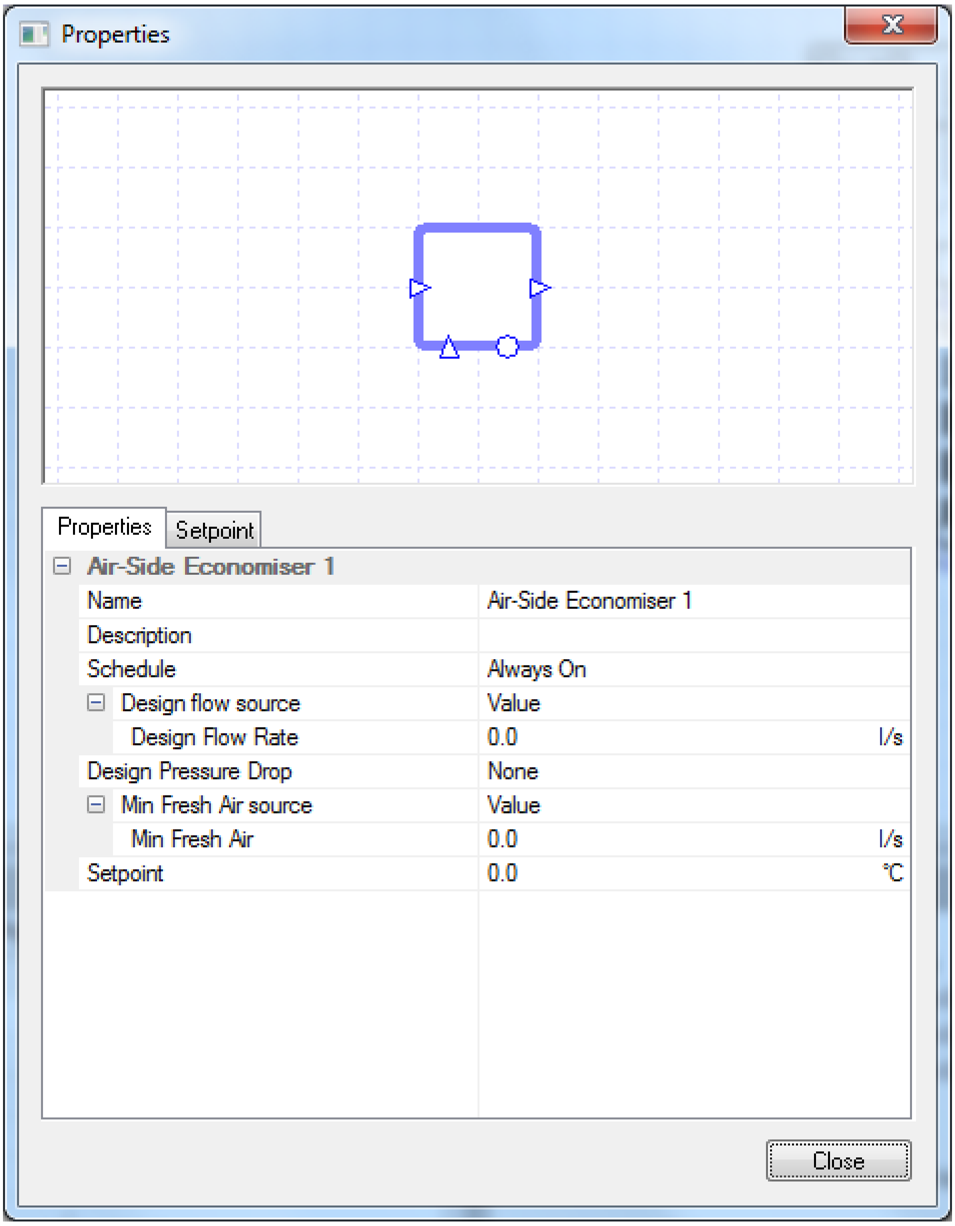
Please note that the seven sizing methods available have already been discussed in the All Attached Zones Sized option above.

- **Design Pressure Drop –** The Design Pressure Drop field, like the Design Flow Rate field, is used during the design flow calculations to fix the capacity of the damper; if the user has not already fixed the capacity. The field sets the drop in pressure, at the design flow rate, of the air as it passes through the damper. The user has the following two options to choose from in this field:
  - None – When this option is chosen, TAS will calculate the pressure drop through the component in a similar manner to how it calculates all other pressure drops through the other air – side components.
  - Value – When this option is chosen, the user is required to enter in the pressure drop at the damper. This pressure drop cannot be bigger than or equal to the sum of all fan components' Pressure fields. If the air will flow through multiple dampers/ air-side economisers, then the sum of these components' Design Pressure Drop fields must not be bigger than or equal to the sum of all fan components' pressure fields. Entering in a higher value will lead to an error calculating the capacities within the user's system. The value entered here will be the pressure drop when the flow rate is at the design flow rate. Please note that if you have a variable flow rate, the pressure drop will be bigger or smaller for decreased flow rates depending on if it is the damper or the fan varying the flow rate.
- **Minimum Flow Source –** The Minimum Flow Source option only appears when a controller is attached to the damper, as this option is only required when varying the flow rate at a damper. This option allows the user to choose how they wish to set the minimum flow rate of the air path the damper is on. The user should note that when using the damper to vary the flow rate they will also need to control the pressure of any fan in the system; an example of how to do this can be found in the TAS Systems Controller guide. The Minimum Flow Source option gives the same eight options as the Design Flow Source option but instead sets the minimum flow source. Please note that there are some differences between the two:
  - The 'none' option does not give you any other options. It means that the flow rate can decrease as needed.
  - The 'All Attached Zones Sized' and 'Nearest Zone Sized' methods come with three additional sizing methods: Hourly Person, Hourly Person and Area, and Hourly Internal Condition. These sizing methods work like the peak methods but are calculated each hour, allowing the user to model systems with a variable fresh air requirement based on occupancy.
  - All options, apart from the 'none' option, allow for a minimum design flow fraction to be set. The minimum design flow fraction allows the user to set the minimum flow rate as a fraction of the design flow rate. Please note that if the minimum design flow fraction and the minimum flow rate both have values in them, TAS will take the maximum of the two as the minimum flow rate. It will also enter the maximum into the Minimum Flow Rate field after a simulation.

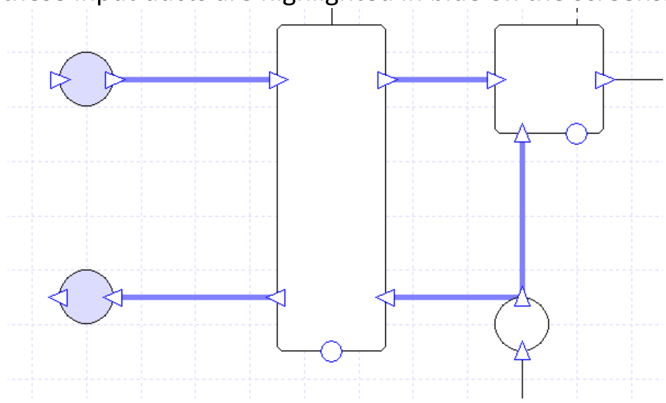
  To ensure that the minimum flow rate is met, TAS works out the signal the damper requires to meet this flow rate and then sets it as the minimum signal of all attached controllers, ensuring the flow rate doesn't drop below this level. It is recommended that all fans on the same air path have the same minimum flow rate.
- **Design Flow Capacity Signal –** The Design Flow Capacity Signal option only appears when a controller is attached to the component. The design flow capacity signal allows the user to make the design flow rate of the damper correspond to a certain signal from the controller. For instance if the Design Flow Capacity Signal field has the value x entered into it then when

the controller's signal reads x, the flow rate through the damper will be the design flow rate. By design the factor is set to one by default and it is recommended that it is kept that way.

2.10 Air-Side Economiser



An air-side economiser should be used in circuits where the air is being re-circulated. The user can set up the minimum amount of fresh air that must enter the circuit along with a target temperature to aim for when mixing the air streams together. As the air-side economiser controls the flow rate, you do not need to use a damper with it. Please note that the air-side economiser takes the air entering the port on the side as being fresh air and the air entering the port at the bottom as the re-circulated air. Controllers can be used with the air-side economiser, and they will control how much the air-side economiser will mix the two air streams together. If a Setpoint Passthrough Standard controller is connected to the air-side economiser, then the signal sent to the air-side economiser will not be between zero and one but instead the reading taken by the sensor. This reading is then used as the temperature setpoint the air-side economiser will attempt to mix the two air streams to. If a Standard controller sending a signal between zero and one is attached to the air-side economiser, the signal specifies the proportion of re-circulated air in the system. A signal of zero means that the air-side economiser will only allow for fresh air to enter the system while a signal of one means that the air-side economiser will only allow for the minimum amount of fresh air to enter the system with the rest being re-circulated. A signal in-between the two indicates the two streams are being mixed, with a lower signal implying that more fresh air makes up the supply air. If no controller is used, the sensor used to determine the air temperature for the Setpoint field is assumed to be placed in the supply air directly after the air-side economiser. It should be noted that the air-side economiser assumes that there are no flow components between the input ducts and the nearest junctions; these input ducts are highlighted in blue on the screenshot below:



Placing a flow component on these ducts will cause issues with how the air-side economiser works; for instance, the minimum fresh air requirement may not be met.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** This option allows the user to apply a schedule to the air-side economiser. The way the Air-Side Economiser component works with the Schedule field is unique compared to the other components. During scheduled hours the air-side economiser will operate by mixing the two air streams such that the minimum fresh air requirement is always met. During the out of schedule hours, the air-side economiser will operate according to the choice set in the "Mode When Off" property.
- **Design Flow Source –** The Design Flow Source option allows the user to choose how they wish to set the design flow rate. The design flow rate, along with the other design values, are used during the design flow calculation to fix the capacities of the flow components (fans, dampers and air-side economisers) without a fixed capacity. When modelling a Constant Air Volume system, this design flow calculation will lead to the design flow rate being the flow rate of the air path the component is on. When modelling a Variable Air Volume system, the design flow calculation will lead to the design flow rate tending to be the maximum allowed flow rate for the air path the component is on. However, there can be instances, where the max flow rate will exceed the design flow rate. Please note that it is recommended that all flow components on the same air path have the same design flow rate, as setting different design flow rates will cause inconsistent design flow rate errors. In TAS Systems, the user has the following options for the Design Flow Source:
    - None – Upon choosing this option the user will be asked to enter the capacity of the flow.
    - Value – Upon choosing this option the user will be asked to enter the design flow rate.
    - All Attached Zones Flow Rate – With this option TAS will add up the flow rate of every zone in this system connected to the air-side economiser by ducts and use this as the design flow rate.
    - All Attached Zones Fresh Air – With this option TAS will add up the fresh air rate of every zone in the system connected to the air-side economiser by ducts and use this as the design flow rate.

- All Attached Zones Sized – With this option, TAS will allow you to use sizing methods to calculate the design flow rate. Each method below is run on every zone in this system connected to the air-side economiser by ducts. The software then sums the sized flow rate for each zone to produce the design flow rate. The seven sizing methods available are:
  - Per Floor Area – Here TAS Systems asks the user to enter a value in $l/s/m^2$ (cfm / ft^2 in U.S. customary units). This value is then multiplied by the area of the zone to give the flow rate in l/s (cfm in U.S. customary units). Please note that the user can use the size fraction field to over-size or under-size this value.
  - Per Volume – Here TAS Systems asks the user to enter a value in $l/s/m^3$ (cfm / ft^3 in U.S. customary units). The value is then multiplied by the volume of the zone to give the flow rate. Please note that the user can use the size fraction field to over-size or under-size this value.
  - ACH – The Value field here stands for how many air changes per hour the user wishes to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.
  - Peak Person – TAS Systems asks the user to enter an air rate per person. TAS then uses the following formula to give the sized flow rate:

$$\textbf{Sized Flow Rate} = \frac{\textbf{Peak Occupancy Sensible Gain} + \textbf{Peak Occupancy Latent Gain}}{\textbf{Metabolic Rate}} * \textbf{Area of zone} * \textbf{Value},$$

    where the: Peak Occupancy Sensible Gain, Peak Occupancy Latent Gain and Metabolic Rate are all read from the zone's internal condition; while Value is the value entered in Systems for the air rate per person. Please note that the user can use the size fraction field to over-size or under-size this value.
  - Peak Person and Area – The Peak Person and Area method allows the user to size the flow rate on both the Peak Person and Per Floor Area methods. The user is asked to enter in their flow rates per person and per floor area. TAS then uses the value entered into the Per Person field in the Peak Person method and the value entered into the Per Area field in the Per Floor Area method. TAS will then add the results of both of these methods together and report the sum as the flow rate.
  - Peak Internal Condition – Please note that the Peak Internal Condition method works like the Peak Person method, but instead of asking the user for the air rate per person it uses the Outside Air rate from the zone's Internal Condition instead. Please note that the user can use the size fraction field to over-size or under-size this value.
  - Delta T – The Delta T method is the default method for sizing the flow rate for a zone. Upon choosing this method, the user is asked to enter a heating delta T and a cooling delta T. The heating delta T value entered should be:

$$\textbf{Max Temp of the supply air} - \textbf{The zone's theromstat lower limit},$$

    where the zone's thermostat is read from the Internal Condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air heated to this maximum temperature to keep the zone's temperature above the lower limit. Please note that this is done using the heating loads from the TSD file, as this sizing is done before the simulation. The cooling delta T works in a similar way with the zone's thermostat upper limit. The cooling delta T value entered should be equal to:

$$\textbf{The Zone's theromsat upper limit} - \textbf{Min temp of the supply air},$$

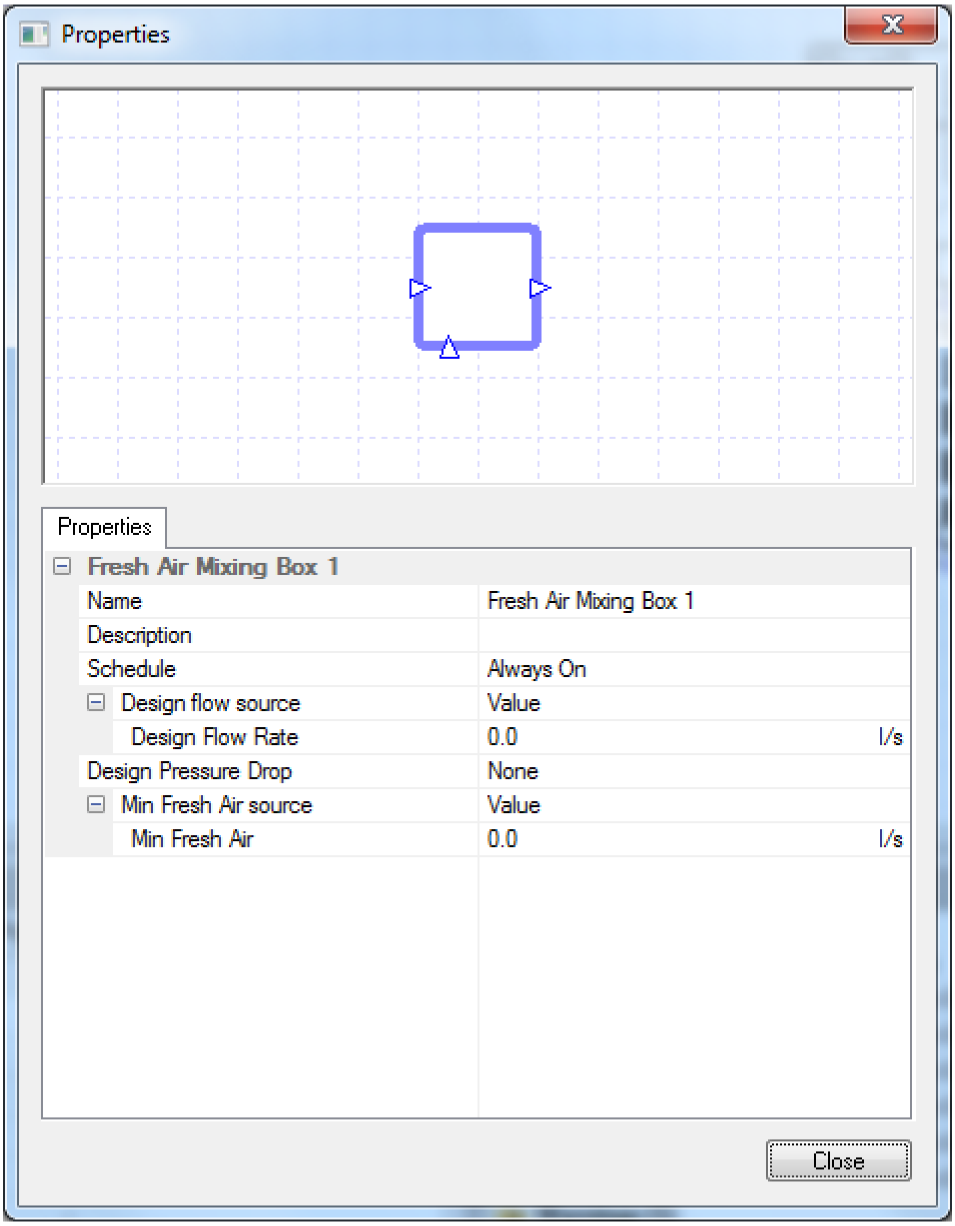
    where the zone's thermostat is read from the internal condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air cooled to this minimum temperature to keep the zone's

temperature below the upper limit. Please note that this is done using the cooling loads from the TSD file, as this sizing is done before the simulation. Upon working out the flow rate needed for the heating delta T and the cooling delta T, TAS will take the max of the two as the flow rate for the zone. Please note that the values entered here do not impact the supply air temperatures. It is just used to work out the sized flow rate.
After entering the delta T values, the user will be asked to choose the design condition (or just the simulation data) to choose what days to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

  - Nearest Zone Flow Rate – With this option TAS will take the flow rate of the nearest zone to the component and use this as the design flow rate.
  - Nearest Zone Fresh Air – With this option TAS takes the fresh air rate of the nearest zone to the component and uses this as its design flow rate.
  - Nearest Zone Sized - With this option, TAS will allow you to use sizing methods to calculate the design flow rate. The selected sizing method will be run on the zone nearest to the component and the resulting sized value will be the design flow rate. Please note that the seven sizing methods available have already been discussed in the All Attached Zones Sized option above.
- **Minimum Fresh Air Source –** The Minimum Fresh Air Source option allows the user to choose how they wish to obtain the minimum fresh air rate. The minimum fresh air rate is the minimum amount of fresh air that the air-side economiser will add to the re-circulated air, where the fresh air is the air coming from the duct connected to the input port on the side of the air-side economiser. The options to choose from are the same as the Design Flow Source options, except:
  - The ‘none’ option means that there is no requirement that fresh air is added to the circuit. This could mean for days on end no fresh air would enter the circuit.
  - The ‘All Attached Zones Sized’ and ‘Nearest Zone Sized’ methods come with three additional sizing methods: Hourly Person, Hourly Person and Area, and Hourly Internal Condition. These sizing methods work like the peak methods but are calculated each hour, allowing the user to model systems with a variable fresh air requirement based on occupancy.
- **Design Pressure Drop –** The Design Pressure Drop field, like the Design Flow Rate field, is used during the design flow calculations to fix the capacity of the air-side economiser; if the user has not already fixed the capacity. The field sets the drop in pressure, at the design flow rate, the air receives when it passes through the air-side economiser. The user has the following two options to choose from in this field:
  - None – When this option is chosen, TAS will calculate the pressure drop through the component in a similar manner as to how it calculates all other pressure drops through the other air – side components.
  - Value – When this option is chosen, the user is required to enter in the pressure drop at the air-side economiser. This pressure drop cannot be bigger than or equal to the sum of all fan components’ Pressure fields. If the air will flow through multiple dampers/ air-side economiser then the sum of these components’ Design Pressure Drop fields must not be bigger than or equal to the sum of all fan components’ Pressure fields. Entering in a higher value will lead to an error calculating the capacities within the user’s system. The value entered here will be the pressure drop when the flow rate is at the design flow rate. Please note that if you have a variable flow rate, the pressure drop will be bigger or smaller for decreased flow rates depending on if it is the damper or the fan varying the flow rate.

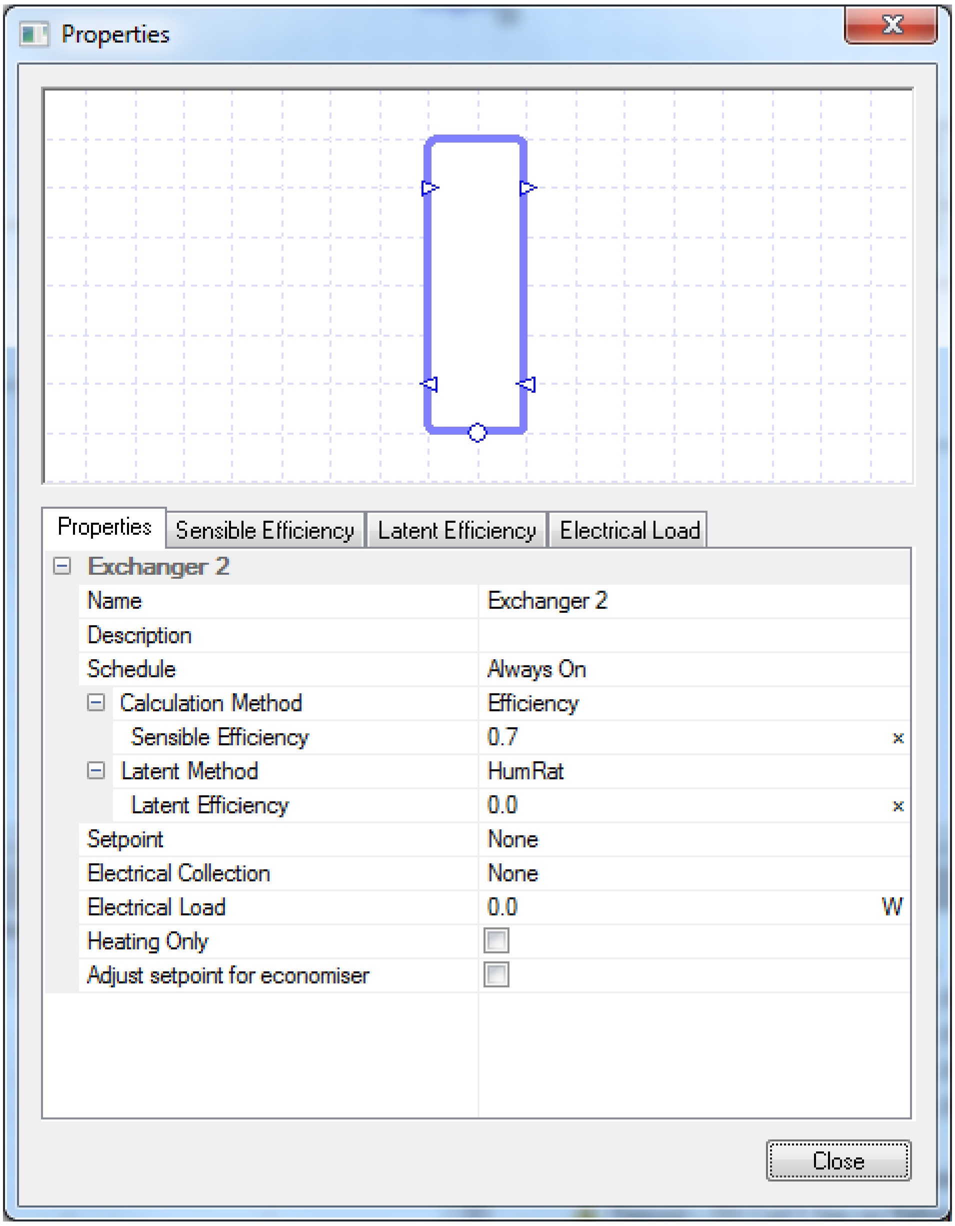
- **Setpoint –** When a temperature is entered into the setpoint field, the component will attempt to regulate the temperature of the air going through it to reach this setpoint. In the case of the air-side economiser, it will try to merge the two air streams together to reach the setpoint. As the air-side economiser has no heating or cooling capability, it will not always be able to meet this setpoint. Please note that when a controller is used in conjunction with the air-side economiser, the Setpoint field will disappear from the properties. This is done because the air-side economiser is being controlled by a controller and will mix the air streams based on the signal sent by the controller. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Mode When Off** – The Mode When Off property only appears in the air-side economiser's properties when a schedule other than "Always On" is applied. The property allows the user to set how the air-side economiser works during out of schedule hours when the fans are active and creating an air flow. These out of scheduled hours will depend on the schedule chosen by the user in the Schedule field. The user gets three options with this field:
    - Through – The "Through" option tells the air-side economiser that during un-scheduled hours the air should not be re-circulated and any air flow needed should be provided by fresh air.
    - Recirc – The "Recirc" option tells the air-side economiser that during un-scheduled hours the air should be re-circulated only, no fresh air should be introduced.
    - No Minimum – The "No Minimum" option tells the air-side economiser that during un-scheduled hours it can still mix the air streams to reduce the loads, but it no longer needs to meet the minimum fresh air requirement.

2.11 Fresh Air Mixing Box



A fresh air mixing box is a simplified version of the air-side economiser. Instead of varying the amount of fresh air introduced into the system to reduce any heating/cooling loads, the fresh air mixing box will always bring in the amount of fresh air specified in the Minimum Fresh Air field during scheduled hours (i.e. it acts like an air-side economiser which has a controller always sending a signal of one). Due to the fresh air mixing box not mixing the two air streams to meet a temperature it does not have a setpoint field and cannot be connected to a controller. However, apart from the above differences, the properties of the fresh air mixing box are the same as the air side economiser and will not be discussed in this section.

2.12 Exchanger



A heat exchanger allows for the transfer of heat from one air stream to the other air stream. The ports at the top should be used for the supply air, while the ports at the bottom should be used for the exhaust air. You can input both the sensible and latent efficiencies of the exchanger. Controllers can be used with the heat exchanger, and they will control how much the exchanger will exchange heat between the two air streams. If a Setpoint Passthrough Standard controller is connected to the heat exchanger, then the signal sent to the exchanger will not be between zero and one but instead the reading taken by the sensor. This reading is then used as the temperature setpoint the exchanger will attempt to get the supply air to. If a Standard controller sending a signal between zero and one is attached to the exchanger the signal specifies the proportion of exhaust air, on the lower air path, that flows through the exchanger. A signal of zero means that all exhaust air will bypass the exchanger, stopping any heat transfer, while a signal of one means that all exhaust air will flow through the exchanger, allowing for the maximum amount of heat transfer to take place. A signal in-between the two implies a partial bypass of the exhaust air, with a lower signal indicating more exhaust air bypassing the exchanger and reducing the amount of heat transfer. Please note that if a controller isn't used and the Setpoint field is set to on, the sensor determining the air temperature for the Setpoint field is assumed to be placed directly after the component in the supply air (after the exit port on the top).

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the exchanger, this will mean that the exchanger will not exchange heat between the two air streams. Instead the air streams will just flow through the exchanger unaltered. The default schedule option is always on, meaning that the component will operate 24/7.
- **Calculation Method –** In TAS Systems, there are three options to calculate the maximum rate of heat transfer for the exchanger:
  - Efficiency – The first calculation method is efficiency. Upon choosing this option the user is asked to enter the sensible efficiency of the exchanger. The sensible efficiency of the exchanger is calculated by the following formula when the supply air's flow rate is equal to the exhaust air's flow rate:

    $$\boldsymbol{Sensible\ Efficiency} = \frac{\boldsymbol{Supply\ Air\ Temp - Fresh\ Air\ Temp}}{\boldsymbol{Return\ Air\ Temp - Fresh\ Air\ Temp}}$$

    Where: $\boldsymbol{Supply\ Air\ Temp}$ is the temperature of the supply air immediately after going through the exchanger, $\boldsymbol{Return\ Air\ Temp}$ is the temperature of the return air before entering the exchanger and $\boldsymbol{Fresh\ Air\ Temp}$ is the temperature of the supply air before entering the exchanger.
    Please note that in the rare cases where the exchanger is used with the air entering it at the top and bottom at different flow rates, the efficiency is worked out with the following equation:

    $$\boldsymbol{Efficiency} = \frac{\left(\boldsymbol{TempAfter_{f1} - TempBefore_{f1}}\right)\boldsymbol{FlowRate_{f1}}}{\left(\boldsymbol{TempBefore_{f2} - TempBefore_{f1}}\right) * \mathbf{min}\left(\boldsymbol{FlowRate_{f1}}, \boldsymbol{FlowRate_{f2}}\right)},$$

    Where: $\boldsymbol{TempAfter}$ is the temperature of the air exiting the exchanger on the selected air flow, $\boldsymbol{TempBefore}$ is the temperature of the air entering the exchanger on the selected air flow, $\boldsymbol{FlowRate}$ is the flow rate of the selected air flow, $\boldsymbol{Min(x\ , y)}$ means to take the minimum value of the two entries, and the subscript $\boldsymbol{f1}$ and $\boldsymbol{f2}$ relates as to whether it is the first or second air stream this value should be taken from.
    Please note that this equation will sometimes give you a negative efficiency, which would be caused in this case by the air stream $\boldsymbol{f1}$ transferring heat to the air stream $\boldsymbol{f2}$. The efficiency entered into the efficiency field should always be the absolute value.
    In the Sensible Efficiency Tab, you can add a modifier to the Sensible Efficiency.
  - NTU Method – The second calculation method is the NTU method. Upon choosing this method, the user will need to input: The heat transfer surface area, the heat transfer coefficient and the Exchanger type. After entering these details, TAS will then work out the rate of heat transfer of the exchanger.
  - Duty – The third calculation method is duty. This option models a heat exchanger that has a built in heat pump that is used to transfer heat from one air stream to the other. As there is a built in heat pump, the user will need to provide the following information:

  - Duty - The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand. For the heat pump in the heat exchanger this would mean it would no longer exchange heat between the two air streams. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. In the Duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
  - Bypass Factor - The Bypass Factor field determines the amount of air that will bypass the heat pump. The value is entered as a factor between 0 and 1 and this factor is then multiplied against the flow rate of the air going through the exchanger. Modifiers can be added to the bypass factor using the Bypass Factor tab.
- **Latent Method –** With the Latent method field, you are provided with two options to calculate the latent energy transfer of the exchanger:
  - HumRat – Upon choosing this option, the user is asked to enter a latent efficiency. The latent efficiency is calculated by the following formula when the supply air's flow rate is equal to the exhaust air's flow rate:

$$\boldsymbol{Latent\ Efficiency} = \frac{\boldsymbol{Supply\ Humidity\ Ratio - Fresh\ Air\ Humidity\ Ratio}}{\boldsymbol{Return\ Humidity\ Ratio - Fresh\ Air\ Humidity\ Ratio}}$$

Where: $\boldsymbol{Supply\ Humidity\ Ratio}$ is the humidity ratio of the supply air immediately after going through the exchanger, $\boldsymbol{Return\ Humidity\ Ratio}$ is the return humidity ratio of the return air before entering the exchanger and the $\boldsymbol{Fresh\ Air\ Humidity\ Ratio}$ is the humidity ratio of the supply air before entering the exchanger.

Please note that in the rare cases where the exchanger is used with the air entering it at the top and bottom at different flow rates, the latent efficiency is worked out with the following equation:

$$\boldsymbol{Efficiency} = \frac{(\boldsymbol{HumRatAfter_{f1} - HumRatBefore_{f1}})\boldsymbol{FlowRate_{f1}}}{(\boldsymbol{HumRatBefore_{f2} - HumRatBefore_{f1}}) * \mathbf{min}(\boldsymbol{FlowRate_{f1}, FlowRate_{f2}})},$$

Where: $\boldsymbol{HumRatAfter}$ is the humidity ratio of the air exiting the exchanger on the selected air flow, $\boldsymbol{HumRatBefore}$ is the humidity ratio of the fluid entering the exchanger on the selected air flow, $\boldsymbol{FlowRate}$ is the flow rate of the selected air flow, $\boldsymbol{Min(x, y)}$ means to take the minimum value of the two entries, and the subscript $\boldsymbol{f1}$ and $\boldsymbol{f2}$ relates as to whether it is the first or second air stream this value should be taken from.

Please note that this equation will sometimes give you a negative efficiency, which would be caused in this case by the air stream $\boldsymbol{f1}$ transferring heat to the air stream $\boldsymbol{f2}$. The efficiency entered into the efficiency field should always be the absolute value.

In the Latent Efficiency tab you can add a modifier to the Latent Efficiency.
  - Enthalpy – Upon choosing this option, the user is asked to enter enthalpy efficiency. The enthalpy efficiency is calculated using the following formula when the supply air's flow rate is equal to the exhaust air's flow rate :

$$\boldsymbol{Enthalpy\ Efficiency} = \frac{\boldsymbol{Supply\ Enthalpy - Fresh\ Air\ Enthalpy}}{\boldsymbol{Return\ Enthalpy - Fresh\ Air\ Enthalpy}},$$

Where: $\boldsymbol{Supply\ Enthalpy}$ is the enthalpy of the supply air immediately after going through the exchanger, $\boldsymbol{Return\ Enthalpy}$ is the enthalpy of the return air before entering the exchanger and $\boldsymbol{Fresh\ Air\ Enthalpy}$ is the enthalpy of the supply air before entering the exchanger.

Please note that in the rare cases where the exchanger is used with the air entering it at the top and bottom at different flow rates, the latent efficiency is worked out with the following equation:

$$Efficiency = \frac{(EnthalpyAfter_{f1} - EnthalpyBefore_{f1})FlowRate_{f1}}{(EnthalpyBefore_{f2} - EnthalpyBefore_{f1}) * \min(FlowRate_{f1}, FlowRate_{f2})},$$

Where: $EnthalpyAfter$ is the enthalpy of the air exiting the exchanger on the selected air flow, $EnthalpyBefore$ is the enthalpy of the fluid entering the exchanger on the selected air flow, $FlowRate$ is the flow rate of the selected air flow, $Min(x, y)$ means to take the minimum value of the two entries, and the subscript $f1$ and $f2$ relates as to whether it is the first or second air stream this value should be taken from.

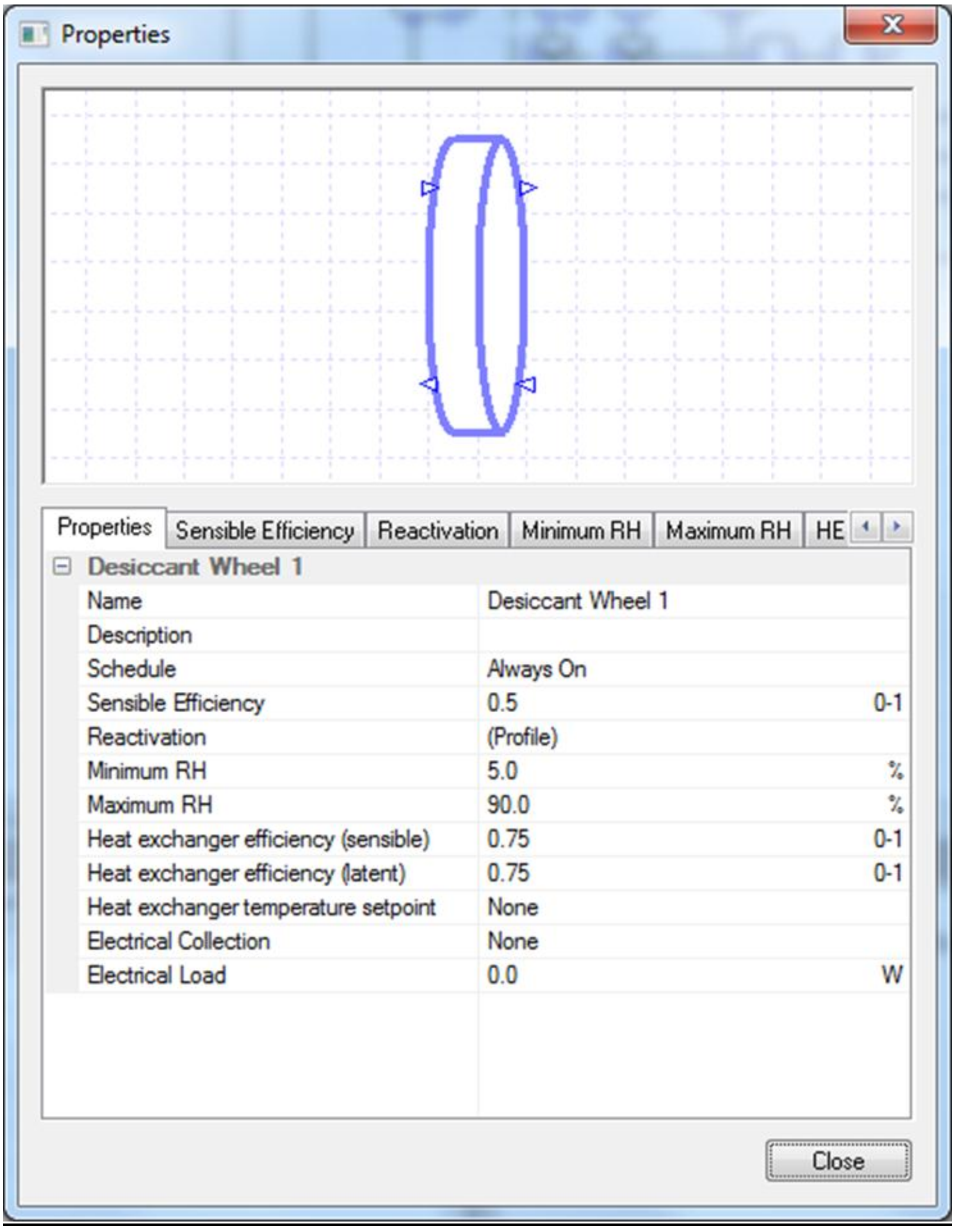
Please note that this equation will sometimes give you a negative efficiency, which would be caused in this case by the air stream $f1$ transferring heat to the air stream $f2$. The efficiency entered into the efficiency field should always be the absolute value.

In the Enthalpy Efficiency tab you can add a modifier to the Latent Efficiency.

- **Setpoint –** With the exchanger, you have two options with the Setpoint: None or On. Upon choosing the "None" option, the heat exchanger will exchange heat depending on the rate of heat transfer calculated from the calculation method. Upon choosing the "On" option, you will be asked to enter a temperature as its value. When a temperature is entered into the Setpoint Value field, the component will attempt to regulate the temperature of the air going through it, by allowing the exhaust air to bypass the exchanger, so that the supply air leaving the exchanger is at this setpoint. In the case of the exchanger it will transfer heat to warm up or cool down the supply air to the setpoint. As the exchanger only transfers heat from the hotter air mass to the cooler one, sometimes it will not be able to meet the temperature setpoint. Factors that will affect this include: the fresh air and extract air temps and the maximum rate of heat transfer. Please note that when a controller is used in conjunction with the heat exchanger, the Setpoint field will disappear from the properties. This is done because the air-side economiser is being controlled by a controller and will exchange heat between the air streams based on the signal sent by the controller. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Electrical Collection –** Collections are a way of grouping components that share the same source of energy. Once a component is added to an electrical collection, any electrical load it has will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. For the heat exchanger, the electrical collection chosen here will have the loads entered in the Electrical Load field passed onto it.
- **Electrical Load –** The Electrical Load field of the heat exchanger allows the user to account for any electrical load in the exchanger, for instance from the pump for the run around coil. This load is then passed onto the plant room collection, which is chosen in the Electrical Collection field. The user can type in a value into this field and the component will pass this amount of electrical load onto the plant room collection for each hour of the simulation. As it is very unlikely that the user will want to pass on the same load for every hour of the simulation, it is recommended that the user goes to the Load tab and creates a modifier for this field. With the various modifiers available, the user will be able to vary the loads throughout the simulation.
- **Heating Only –** When this tick box is ticked, the heat exchanger will only be able to exchange heat if this will warm up the supply air. If your heat exchanger can exchange heat to both heat and cool down the supply air, then you should leave this option un-ticked.
- **Adjust Setpoint for Economiser –** When this tick box is ticked and there is an air-side economiser directly after the heat exchanger, the exchanger will try and exchange heat

between the air streams to control the temperature of the air after the air-side economiser to the temperature entered into the Setpoint field. Doing this eliminates the rare scenarios where the heat exchanger and air side economiser could fight against each other, increasing the heating and cooling loads.

2.13 Desiccant Wheel



A desiccant wheel is a rotary heat exchanger with a coating of desiccant on it. The supply air for the zone should use the top ports of the desiccant wheel while the exhaust air should use the bottom ports. When one of the air streams is at a temperature, set by the reactivation profile, such that the desiccant is fully reactivated then the desiccant wheel allows for heat and moisture to be transferred between the two air streams along lines of constant enthalpy. When both air streams temperatures are at a temperature such that the desiccant is not reactivated at all, the desiccant wheel acts like a heat exchanger. Depending on the profile used for the reactivation, the desiccant can be partially reactivated. When the desiccant is partially reactivated the desiccant wheel's properties are a mixture of when the desiccant is fully reactivated and when the desiccant is not reactivated at all. Please note that a controller cannot be used with a desiccant wheel.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the desiccant wheel, this will mean that the desiccant wheel will not exchange heat or moisture between the two air streams. Instead the air streams will just flow through the desiccant wheel unaltered. The default schedule option is always on, meaning that the component will operate 24/7.
- **Sensible Efficiency –** This field sets the sensible efficiency of the desiccant wheel when the desiccant is fully reactivated. The sensible efficiency is calculated by the following formula when the supply air's flow rate is equal to the exhaust air's flow rate:

$$\boldsymbol{Sensible\ Efficiency} = \frac{\boldsymbol{Supply\ Air\ Temp - Fresh\ Air\ Temp}}{\boldsymbol{Return\ Air\ Temp - Fresh\ Air\ Temp}}$$

  Where: $\boldsymbol{Supply\ Air\ Temp}$ is the temperature of the supply air immediately after going through the desiccant wheel, $\boldsymbol{Return\ Air\ Temp}$ is the temperature of the return air before entering the desiccant wheel and $\boldsymbol{Fresh\ Air\ Temp}$ is the temperature of the supply air before entering the desiccant wheel.

  Please note that in the rare cases where the desiccant wheel is used with the air entering it at the top and bottom at different flow rates, the efficiency is worked out with the following equation:

$$\boldsymbol{Efficiency} = \frac{(\boldsymbol{TempAfter_{f1} - TempBefore_{f1}})\boldsymbol{FlowRate_{f1}}}{(\boldsymbol{TempBefore_{f2} - TempBefore_{f1}}) * \mathbf{min}(\boldsymbol{FlowRate_{f1}, FlowRate_{f2}})},$$

  Where: $\boldsymbol{TempAfter}$ is the temperature of the air exiting the exchanger on the selected air flow, $\boldsymbol{TempBefore}$ is the temperature of the air entering the exchanger on the selected air flow, $\boldsymbol{FlowRate}$ is the flow rate of the selected air flow, $\boldsymbol{Min(x, y)}$ means to take the minimum value of the two entries, and the subscript $\boldsymbol{f1}$ and $\boldsymbol{f2}$ relates as to whether it is the first or second air stream this value should be taken from.

  Please note that this equation will sometimes give you a negative efficiency, which would be caused in this case by the air stream $\boldsymbol{f1}$ transferring heat to the air stream $\boldsymbol{f2}$. The efficiency entered into the Efficiency field should always be the absolute value. The user can place a modifier on this field using the Sensible Efficiency tab.
- **Minimum RH –** This field sets the minimum relative humidity that either airstream can leave the desiccant wheel with when the desiccant is reactivated. Please note that the user can place a modifier on this field using the Minimum RH tab.
- **Maximum RH -** This field sets the maximum relative humidity that either airstream can leave the desiccant wheel with when the desiccant is reactivated. Please note that the user can place a modifier on this field using the Maximum RH tab.
- **Reactivation –** The Reactivation field takes the user to the Reactivation tab where they are asked to create a profile to set up when the desiccant is reactivated. When using the profile, an entry of zero for the Reactivation field indicates when the desiccant is not reactivated. This means that the desiccant wheel acts as a heat exchanger using the properties from the Heat Exchanger fields. An entry of one in the Reactivation field indicates when the desiccant is fully reactivated. This means the desiccant wheel transfers both heat and moisture along lines of constant enthalpy and uses the Sensible Efficiency and RH fields. Values in the reactivation field between zero and one indicate where the desiccant is partially reactivated. In this case the desiccant wheel uses a mixture of both properties. By default a profile has been set up based on dry bulb temperature, as the reactivation profile would normally have the desiccant activated at higher dry bulb temperatures. TAS will only match the air stream (supply or exhaust) with the highest temperature against the profile. Please note that while technically you could delete the profile and enter a value in for the Reactivation field, it is not recommended to do so and you will receive a warning if you try.

- **Heat Exchanger Efficiency (Sensible) –** This field allows the user to input the sensible efficiency of the desiccant wheel when the desiccant is not activated. The sensible efficiency of the desiccant wheel is calculated by the following formula when the supply air's flow rate is equal to the exhaust air's flow rate :

$$\textbf{Sensible Efficiency} = \frac{\textbf{Supply Air Temp} - \textbf{Fresh Air Temp}}{\textbf{Return Air Temp} - \textbf{Fresh Air Temp}}$$

Where: $\textbf{Supply Air Temp}$ is the temperature of the supply air immediately after going through the desiccant wheel, $\textbf{Return Air Temp}$ is the temperature of the return air before entering the desiccant wheel and $\textbf{Fresh Air Temp}$ is the temperature of the supply air before entering the desiccant wheel.
Please note that in the rare cases where the desiccant wheel is used with the air entering at the top and bottom of the desiccant wheel at different flow rates, the efficiency is worked out with the following equation:

$$\textbf{Efficiency} = \frac{(\textbf{TempAfter}_{f1} - \textbf{TempBefore}_{f1})\textbf{FlowRate}_{f1}}{(\textbf{TempBefore}_{f2} - \textbf{TempBefore}_{f1}) * \min(\textbf{FlowRate}_{f1}, \textbf{FlowRate}_{f2})},$$

Where: $\textbf{TempAfter}$ is the temperature of the air exiting the exchanger on the selected air flow, $\textbf{TempBefore}$ is the temperature of the air entering the exchanger on the selected air flow, $\textbf{FlowRate}$ is the flow rate of the selected air flow, $\textbf{Min}(\boldsymbol{x}, \boldsymbol{y})$ means to take the minimum value of the two entries, and the subscript $\boldsymbol{f1}$ and $\boldsymbol{f2}$ relates as to whether it is the first or second air stream this value should be taken from.
Please note that this equation will sometimes give you a negative efficiency, which would be caused in this case by the air stream $\boldsymbol{f1}$ transferring heat to the air stream $\boldsymbol{f2}$. The efficiency entered into the Efficiency field should always be the absolute value. The user can place a modifier on this field using the HE eff (Sensible) tab.

- **Heat Exchanger Efficiency (Latent) –** This field allows the user to input the latent efficiency of the desiccant wheel when the desiccant is not activated. The latent efficiency is calculated by the following formula when the supply air's flow rate is equal to the exhaust air's flow rate :

$$\textbf{Latent Efficiency} = \frac{\textbf{Supply Humidity Ratio} - \textbf{Fresh Air Humidity Ratio}}{\textbf{Return Humidity Ratio} - \textbf{Fresh Air Humidity Ratio}}$$

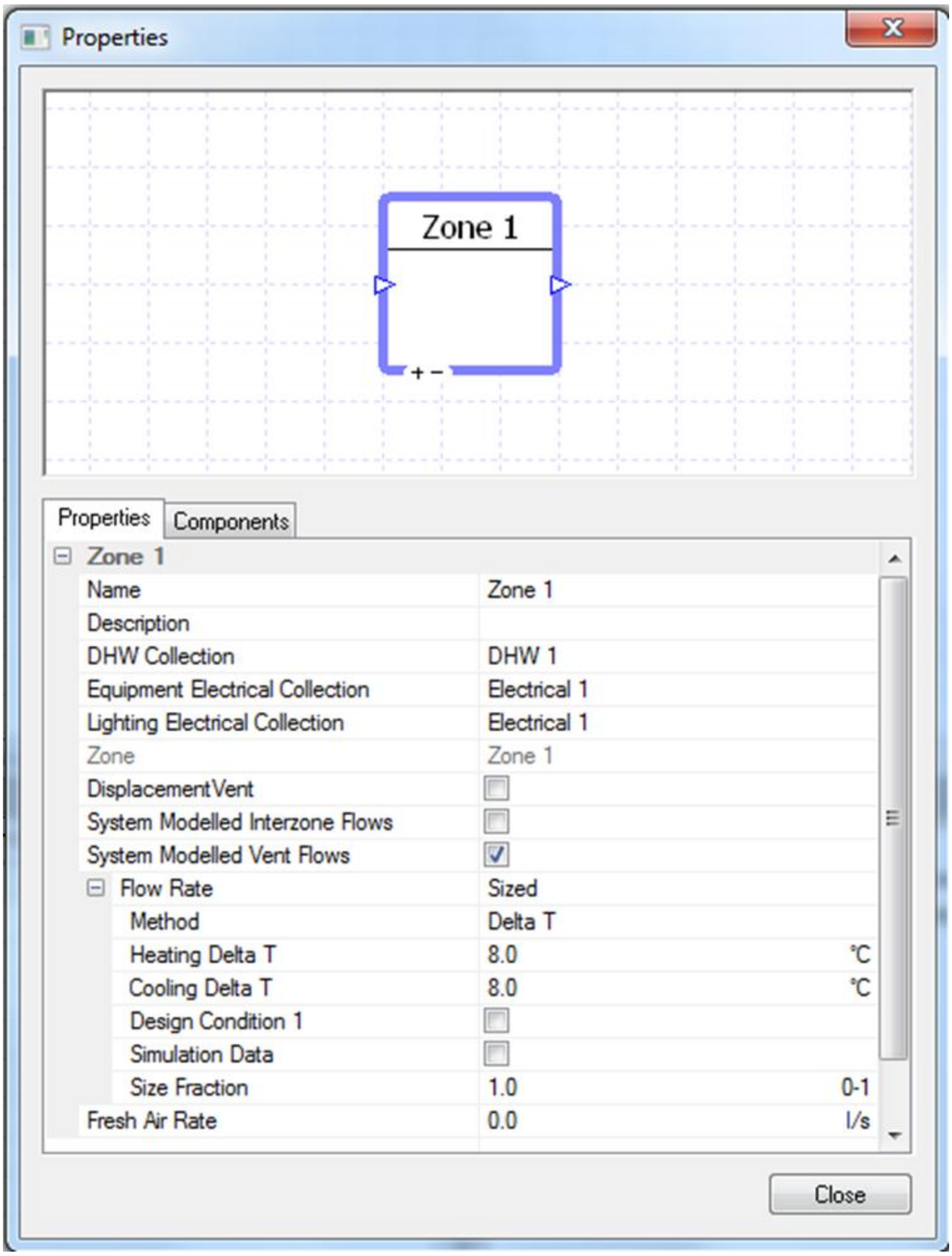
Where: $\textbf{Supply Humidity Ratio}$ is the humidity ratio of the supply air immediately after going through the desiccant wheel, $\textbf{Return Humidity Ratio}$ is the humidity ratio of the return air before entering the desiccant wheel and $\textbf{Fresh Air Humidity Ratio}$ is the humidity ratio of the supply air before entering the desiccant wheel.
Please note that in the rare cases where the desiccant wheel is used with the air entering at the top and bottom of the desiccant wheel at different flow rates, the latent efficiency is worked out with the following equation:

$$\textbf{Efficiency} = \frac{(\textbf{HumRatAfter}_{f1} - \textbf{HumRatBefore}_{f1})\textbf{FlowRate}_{f1}}{(\textbf{HumRatBefore}_{f2} - \textbf{HumRatBefore}_{f1}) * \min(\textbf{FlowRate}_{f1}, \textbf{FlowRate}_{f2})},$$

Where: $\textbf{HumRatAfter}$ is the humidity ratio of the air exiting the exchanger on the selected air flow, $\textbf{HumRatBefore}$ is the humidity ratio of the fluid entering the exchanger on the selected air flow, $\textbf{FlowRate}$ is the flow rate of the selected air flow, $\textbf{Min}(\boldsymbol{x}, \boldsymbol{y})$ means to take the minimum value of the two entries, and the subscript $\boldsymbol{f1}$ and $\boldsymbol{f2}$ relates as to whether it is the first or second air stream this value should be taken from.
Please note that this equation will sometimes give you a negative efficiency, which would be caused in this case by the air stream $\boldsymbol{f1}$ transferring heat to the air stream $\boldsymbol{f2}$. The efficiency entered into the Efficiency field should always be the absolute value. The user can place a modifier on this field using the HE eff (latent) tab.

- **Heat Exchanger Temperature Setpoint –** This field allows the user to input the optional temperature setpoint of the desiccant wheel when the desiccant is not activated. You have two options with the Setpoint: None or On. Upon choosing the 'none' option, the desiccant wheel will exchange heat depending on the rate of heat transfer calculated from the Heat Exchanger Efficiency (Sensible) field. Upon choosing the 'On' option, you will be asked to enter a temperature as its value. When a temperature is entered into the setpoint value field, the component will attempt to regulate the temperature of the air going through it to reach this setpoint. In the case of the desiccant wheel, it will transfer heat from the hotter air flow to the cooler one to warm up or cool down the supply air to the setpoint. As this field is only used by the desiccant wheel when it is in heat exchanger mode, sometimes it will not be able to meet the temperature setpoint. Factors that will affect this include: the fresh air and extract air temps and the maximum rate of heat transfer.
- **Electrical Collection –** Collections are a way of grouping components that share the same source of energy. Once a component is added to an electrical collection, any electrical load it has will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. For the desiccant wheel, the electrical collection chosen here will have the loads entered in the Electrical load field passed onto it.
- **Electrical Load –** The Electrical Load field of the desiccant wheel allows the user to account for any electrical load in the desiccant wheel, for instance from the wheel's motor. This load is then passed onto the plant room collection, which is chosen in the Electrical Collection field. The user can type in a value into this field and the component will pass this amount of electrical load onto the plant room collection for each hour of the simulation. As it is very unlikely that the user will want to pass on the same load for every hour of the simulation, it is recommended that the user goes to the Load tab and creates a modifier for this field. With the various modifiers available, the user will be able to vary the loads throughout the simulation.

2.14 Zone



The Zone Component in TAS Systems is used to model a zone's placement in the systems schematic. The user will need to drag a zone from the TSD Data folder, from the Project Explorer in Systems and not from the TSD itself, onto the Zone Component to tell TAS Systems what zone the component is modelling. If the user groups some components together, and the group includes a Zone Component, they can then drag a Zone Group onto the grouped components to model an air handling unit serving multiple zones. Please note that you can drag and drop zone components from the library into the zone component. These Zone Components will be discussed separately. While you cannot use a controller with the zone component, you can connect sensors from other controllers to the zone by placing the controller on the +- symbol at the bottom of the component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **DHW Collection -** Collections are a way of grouping components that share the same source of energy. Once a zone component is added to a DHW collection, any DHW load the zone has (decided by the zone's internal condition) will be added to that collection's DHW demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. For a zone, the DHW demand can only be placed in DHW groups.
- **Equipment Electrical Collection –** Collections are a way of grouping components that share the same source of energy. Once a zone component is added to an Equipment Electrical Collection, any electrical load from equipment the zone has (decided by the equipment gains from the zone's internal condition) will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. For a zone, the equipment electrical demand can only be placed in Electrical groups.
- **Lighting Electrical Collection –** Collections are a way of grouping components that share the same source of energy. Once a zone component is added to a Lighting Electrical Collection, any electrical load from the lighting the zone has (decided by the lighting gains from the zone's internal condition) will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. For a zone, the lighting electrical demand can only be placed in Electrical groups.
- **Zone –** The Zone field displays the name of the zone being modelled by the component. It is greyed out in the properties list as it cannot be edited from here. To change the zone being modelled by the component, the user should select the new zone from the TSD data folder and drag and drop it onto the component.
- **Displacement Vent –** The Displacement Ventilation option allows the user to model displacement ventilation in their zone by ticking this option. With displacement ventilation, your conditioned cool supply air is supplied to the zone at floor level while warmer air is extracted from the zone at ceiling level. Please note that when using the Displacement Ventilation option in conjunction with zone components, the effect of the displacement ventilation will only occur in the hours the zone components are not actively meeting any heating or cooling demand.
- **System Modelled Interzone Flows –** This option only affects models with IZAMs. When the option is ticked, TAS Systems will ignore any IZAMs in the TBD and the air movement will have to be modelled in TAS Systems. If the option is un-ticked then the IZAMs will be taken into account. The default option for this field is un-ticked.
- **System Modelled Vent Flows –** This option affects the ventilation values in the Internal Condition for each zone. When this option is ticked, TAS Systems will ignore the ventilation values in the TBD and the ventilation will have to be modelled in TAS Systems. The default option for this is ticked.
- **Flow Rate –** The Flow Rate field of a Zone component is a parameter that can be used by flow components (fans, dampers and air-side economisers) to set the flow rate around the system. Please note that just entering a flow rate here does not mean that this will be the flow rate through the zone. Instead, it will depend on the design flow rate of the nearest flow component, which can be set up to read the value specified in this field.
  There are 3 options on setting the flow rate target of a zone. The first one, not used, indicates that the target flow rate is not used in the simulation. The second option, value, allows the user to type in the flow rate into this field. The final option is sized. Upon choosing this option, the user is asked to choose from the following seven sizing methods:

- Per Floor Area – Here TAS Systems asks the user to enter a value in $l/s/m^2$ (cfm / ft^2 in U.S. customary units). This value is then multiplied by the area of the zone to give the flow rate in l/s (cfm in U.S. customary units). Please note that the user can use the size fraction field to over-size or under-size this value.
- Per Volume – Here TAS Systems asks the user to enter a value in $l/s/m^3$ (cfm / ft^3 in U.S. customary units). The value is then multiplied by the volume of the zone to give the flow rate. Please note that the user can use the size fraction field to over-size or under-size this value.
- ACH – The Value field here stands for how many air changes per hour the user wishes to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person – TAS Systems asks the user to enter an air rate per person. TAS then uses the following formula to give the sized flow rate:

$$\boldsymbol{Sized\ Flow\ Rate} = \frac{\boldsymbol{Peak\ Occupancy\ Sensible\ Gain + Peak\ Occupancy\ Latent\ Gain}}{\boldsymbol{Metabolic\ Rate}} * \boldsymbol{Area\ of\ zone} * \boldsymbol{Value},$$

  where the: Peak Occupancy Sensible Gain, Peak Occupancy Latent Gain and Metabolic Rate are all read from the zone’s internal condition and value is the value entered in Systems for the air rate per person. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person and Area – The Peak Person and Area method allows the user to size the flow rate on both the Peak Person and Per Floor Area methods. The user is asked to enter in their flow rates per person and per floor area. TAS then uses the value entered into the Per Person field in the Peak Person method and the value entered into the Per Area field in the Per Floor Area method. TAS will then add the results of both of these methods together and report the sum as the flow rate.
- Peak Internal Condition – Please note that the Peak Internal Condition method works like the Peak Person method, but instead of asking the user for the air rate per person it uses the Outside Air rate from the zone’s Internal Condition instead. Please note that the user can use the size fraction field to over-size or under-size this value.
- Delta T – The Delta T method is the default method for sizing the flow rate for a zone. Upon choosing this method, the user is asked to enter a heating delta T and a cooling delta T. The heating delta T value entered should be:

$$\boldsymbol{Max\ Temp\ of\ the\ supply\ air - The\ zone's\ thermostat\ lower\ limit},$$

  where the zone’s thermostat is read from the Internal Condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air heated to this maximum temperature to keep the zone’s temperature above the lower limit. Please note that this is done using the heating loads from the TSD file, as this sizing is done before the simulation.

  The cooling delta T works in a similar way with the zone’s thermostat upper limit. The cooling delta T value entered should be equal to:

$$\boldsymbol{The\ Zone's\ thermostat\ upper\ limit - Min\ temp\ of\ the\ supply\ air},$$

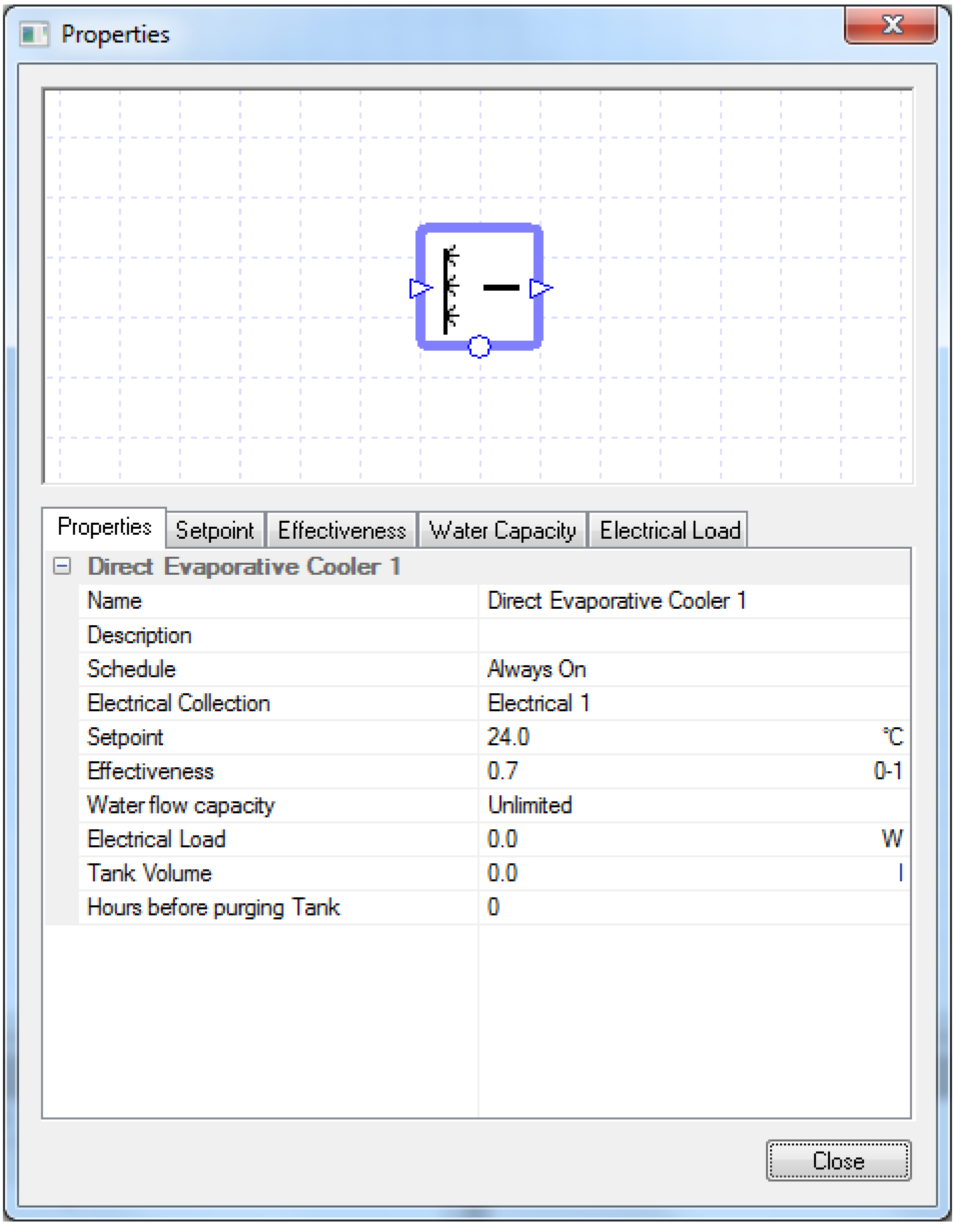
  where the zone’s thermostat is read from the internal condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air cooled to this minimum temperature to keep the zone’s temperature below the upper limit. Please note that this is done using the cooling loads from the TSD file, as this sizing is done before the simulation.

  Upon working out the flow rate needed for the heating delta T and the cooling delta T, TAS will take the max of the two as the flow rate for the zone. Please note that the values entered here do not impact the supply air temperatures. It is just used to work out the sized flow rate.

After entering the delta T values, the user will be asked to choose the design condition (or just the simulation data) to choose what days to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

- **Fresh Air Rate** – The Fresh Air Rate field of a zone is a parameter that can be used by flow components (fans, dampers and air-side economisers) to help set the minimum fresh air requirement for the system. Please note that just entering a value here does not mean that this will be the amount of fresh air supplied to the zone. Instead it will depend on your system set up, especially the flow component setup.
  There are three options for setting the Fresh Air Rate field of a zone component. On the whole, the three methods are exactly the same as discussed in the Flow Rate Field. However with the sizing method there are an extra 3 sizing methods to choose from: Hourly Person, Hourly Person and Area, and Hourly Internal Condition. These sizing methods work like the peak methods but are calculated for each hour, allowing the user to model systems with a variable fresh air requirement based on occupancy.

2.15 Direct Evaporative Cooler



A direct evaporative cooler cools the air flowing through it by spraying water into it. The spray cools the air by evaporative cooling, leading to an increase of humidity along lines of constant enthalpy on the Psychrometric Chart. You can use a controller with the direct evaporative cooler to control how the cooler behaves. For a direct evaporative cooler, the controller controls the cooler by informing it of the amount of water it should add to the air to cool it. The controller does this by sending a signal, between zero and one, to the cooler dictating the proportion of the cooler's water flow capacity it should use as the water flow rate to cool the air. So, for example, if the cooler received a signal of zero the cooler would not cool the air flowing through it. While if it received a signal of 1, the cooler will cool the air flowing through it by adding the maximum amount of water allowed from the Water Flow Capacity field. If no controller is used, the sensor used to determine the temperature for the Setpoint field is assumed to be directly after the cooler.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the direct evaporative cooler, this will mean that the air will flow through the cooler without being cooled, even if there is a controller sending a non-zero signal to the cooler. The default schedule option is always on, meaning that the component will operate 24/7.
- **Electrical Collection -** Collections are a way of grouping components that share the same source of energy. Once a component is added to an electrical collection, any electrical load it has will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. The Direct Evaporative Cooler component in TAS Systems can only join Electrical Groups.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the air going through it to reach the setpoint. In the case of the direct evaporate cooler, it will cool the air so it reaches the setpoint, but it will not be able to warm the air to reach this setpoint. To warm the air you would need another component, for instance a heating coil. Please note that when a controller is used in conjunction with the cooler, the Setpoint field will disappear from the properties. This is done because the cooler is being controlled by a controller and will cool down the air when the controller sends a signal informing the coil to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Effectiveness** – This option allows the user to input how effective the direct evaporative cooler is. The effectiveness needs to be entered as a factor between 0 and 1. The effectiveness of the direct evaporative cooler can be found using the following formula:

$$\boldsymbol{Effectiveness} = \frac{\boldsymbol{T_{in}} - \boldsymbol{T_{out}}}{\boldsymbol{T_{in}} - \boldsymbol{T_{sat}}} = \frac{\boldsymbol{H_{out}} - \boldsymbol{H_{in}}}{\boldsymbol{H_{sat}} - \boldsymbol{H_{in}}},$$

  where $\boldsymbol{T}$ is the air temperature and $\boldsymbol{H}$ is the humidity ratio. Please note that the subscript on each letter denotes where that measurement is taken. A subscript $\boldsymbol{in}$ denotes that the measurement is taken on the air entering the direct evaporative cooler; a subscript $\boldsymbol{out}$ denotes that the measurement is taken on the air exiting the direct evaporative cooler and the subscript $\boldsymbol{sat}$ denotes the temperature or humidity ratio of when the air would be saturated when following the line of constant enthalpy. In the Effectiveness tab you are able to apply a modifier to this field.
- **Water Flow Capacity** – The Water Flow Capacity field allows the user to enter the Water Flow Capacity of the direct evaporative cooler. If the water flow capacity is set too low, the cooler will not be able to cool the air to the setpoint. In TAS Systems there are 3 options to choose from to set the water flow capacity:
  - Unlimited – This means there is no limit on the water flow, meaning the cooler can spray as much water into the air as it needs to. Please note that this option cannot be used when a controller is connected to the component.
  - Value – The user can enter a value to provide a maximum limit to the water flow.
  - Sized - Allows the user to size the Water Flow Capacity on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.

  In the Water Flow Capacity tab, you will have these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Electrical Load –** The Electrical Load field of the direct evaporative cooler allows the user to account for any electrical load in the cooler, for instance the load associated with spraying

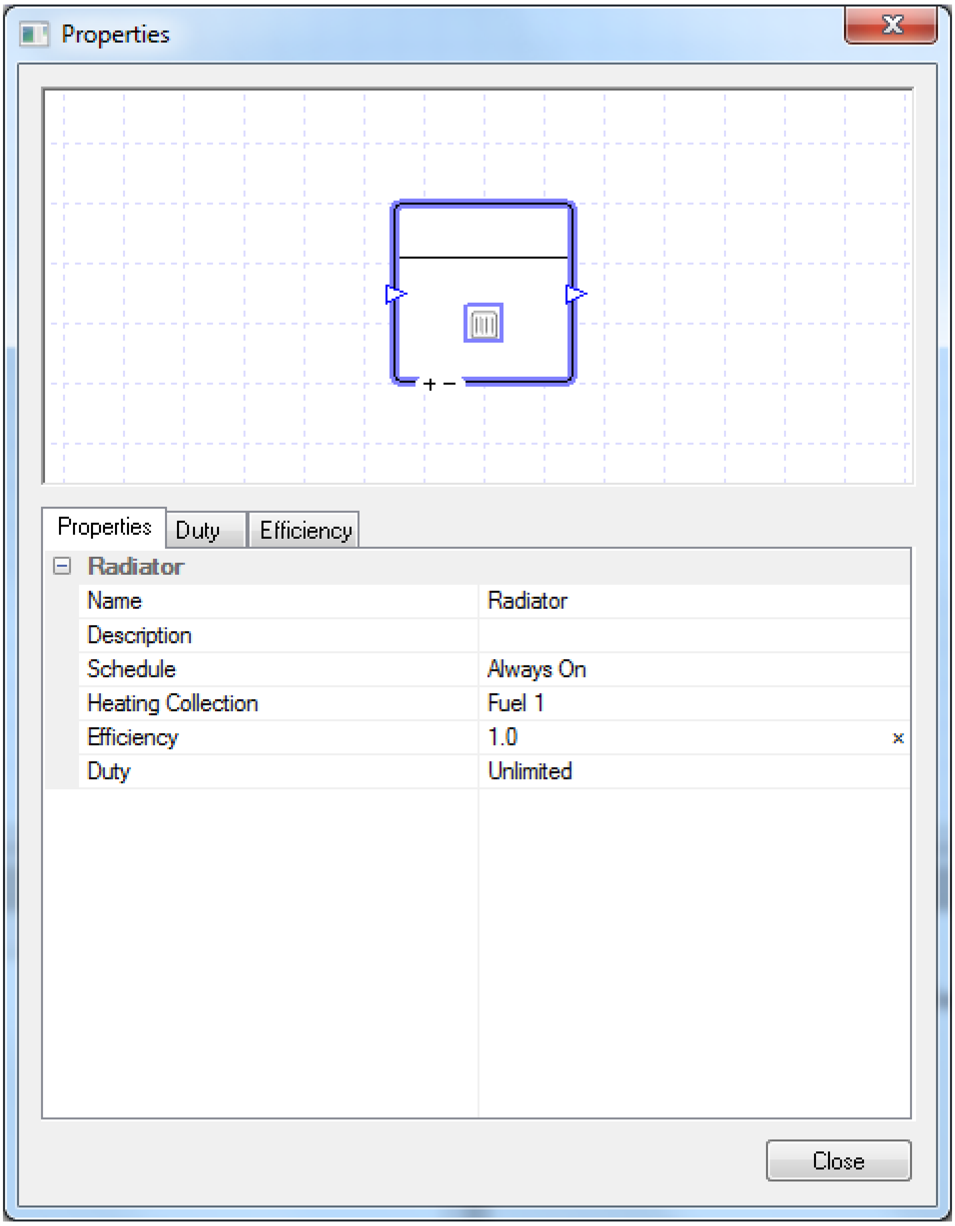
the water into the air stream. This load is then passed onto the plant room collection, which is chosen in the Electrical Collection field. The user can type in a value into this field and the component will pass this amount of electrical load onto the plant room collection for each hour of the simulation. As it is very unlikely that the user will want to pass on the same load for every hour of the simulation, it is recommended that the user goes to the Electrical Load tab and creates a modifier for this field. With the various modifiers available, the user will be able to vary the loads throughout the simulation.

- **Tank Volume –** The Tank Volume field requires the user to enter the volume of the evaporative dry cooler's tank.
- **Hours Before Purging Tank –** The Hours Before Purging Tank field allows the user to enter the amount of hours the cooler will be inactive for before the tank is emptied. If zero is entered into this field, then the tank will never be purged.

3 Zone Components

The zone components listed here can only be placed within a zone. They are used to model in space heating and cooling and can be used to supply additional heating or cooling when the supply air cannot condition the zone appropriately.

3.1 Radiator



The Radiator component in TAS only provides heating to the space. It heats the room up by convection and radiation, where the radiator's radiant proportion is set in the internal condition of each zone. Please note that the user will need to edit the radiant proportion of the heating emitter in the zone's internal condition to model the radiator correctly. You cannot use controllers with this component and it will always heat the space to the lower limit of the zone's thermostat.

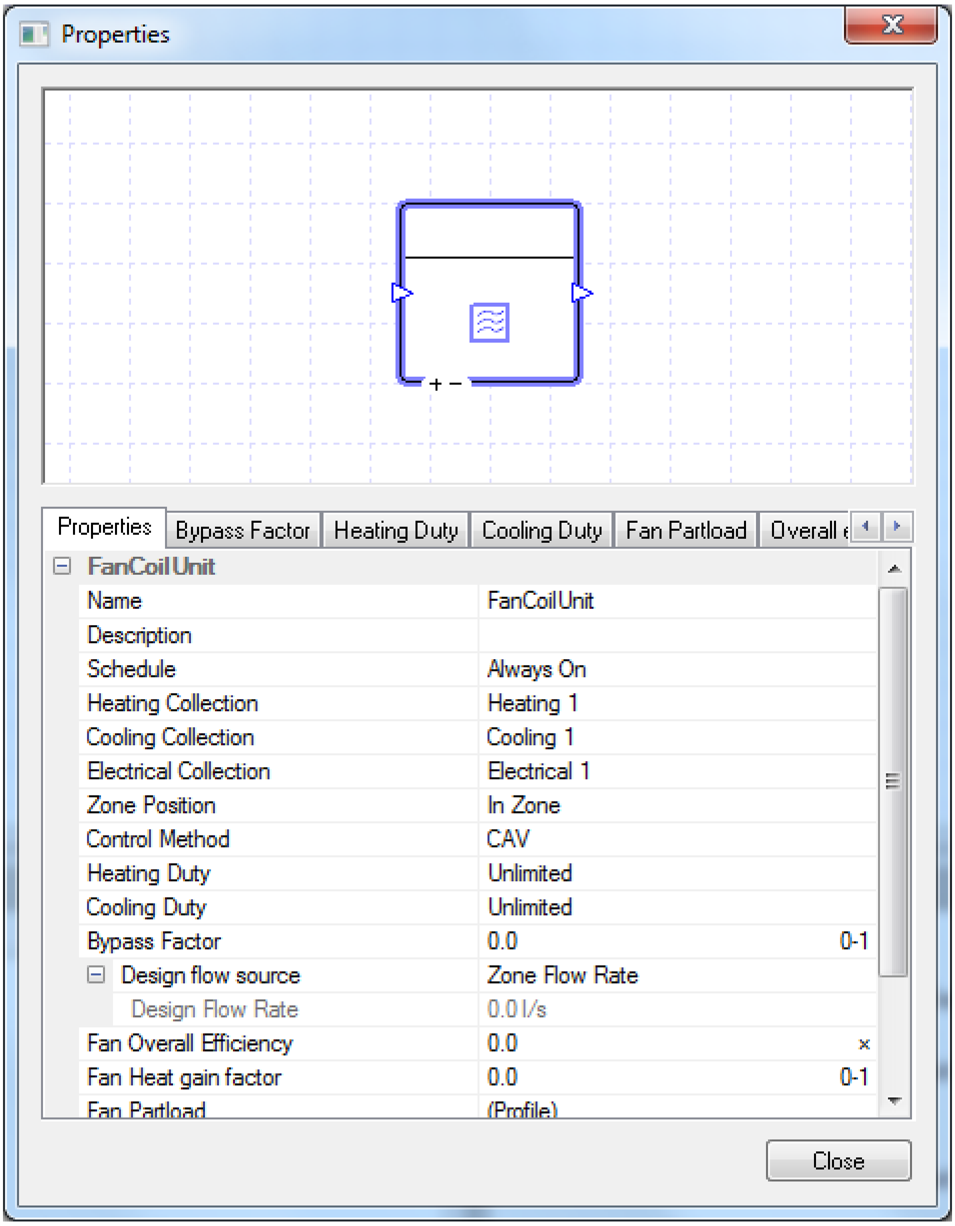
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the radiator, it will not heat up the zone even if the temperature of the zone is below the lower limit of the zone's thermostat. The default schedule option is always on, meaning that the component will operate 24/7.
- **Heating Collection –** Collections are a way of grouping components that share the same source of energy. Once a component is added to a Heating, Fuel or Electrical collection, any heating load it has will be added to that collection's demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Please note you will receive a warning telling you this. With the Heating Collection, you will be able to join: Heating, Fuel and Electrical groups. Your choice of group will depend on how the heat is being generated. For instance if the heat is being provided to the radiator from a boiler, you would choose a Heating group but if the radiator was generating the heat itself by burning fuel, you would choose a Fuel group.
- **Efficiency** – The Efficiency field of a heating component only appears when a user chooses: an electrical group, a fuel group or the "None" option in the Heating Collection field; as choosing one of these options allows the user to model a component using its energy source directly at the component to produce heat. The Efficiency field allows the user to enter how efficient this process is, as a factor. This field will not have any effect on the air-side results apart from the consumption results for this component now appearing in the results section of this component. Please note while the option appears for the "none" choice in the Heating Collection field, any loads from the component will be discarded. The user will be able to add a modifier to this field by going to the Efficiency tab which also appears when the appropriate options are chosen in the Heating Collection Field.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand**.** For the Radiator this would mean it wouldn't be able to heat the air to the lower limit of the thermostat, it would fall short. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. There are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

3.2 Fancoil Unit



A Fancoil Unit consists of: a heating coil, a cooling coil and a fan. It can be used as the source of heating and cooling for a zone and it can also be used to provide additional heating and cooling to a zone if the supply air hasn't been able to condition the zone appropriately. In a set hour, the Fancoil can only heat or cool the zone; it cannot do both at the same time. You cannot use controllers with this component and it will always heat and cool the space to the lower and upper limit of the zone's thermostat. It should be noted that the fancoil uses the radiant proportion from the zones internal condition in the TBD to determine the proportion of radiant energy the emitter produces. In the case of a fancoil unit, this would most likely be set to zero.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they

should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the fancoil, neither the fan nor the coils will operate. This means that the fancoil unit will not condition the air entering it and air will only flow through the fancoil unit if it is a terminal unit and supply air is being introduced to the zone. The default schedule option is always on, meaning that the component will operate 24/7.

- **Heating Collection –** Collections are a way of grouping components that share the same source of energy. Once the Fancoil unit is added to a Heating, Fuel or Electrical collection, any heating load it has will be added to that collection's demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Please note you will receive a warning telling you this. With the Heating Collection, you will be able to join: Heating, Fuel and Electrical groups. Your choice of group will depend on how the heat is being generated. For instance if the heat is being provided to the coil from a boiler, you would choose a Heating group but if the coil was generating the heat itself by burning fuel, you would choose a Fuel group.
- **Cooling Collection –** Collections are a way of grouping components that share the same source of energy. Once a Fancoil unit is added to a Cooling collection, any cooling load it has will be added to that collection's cooling demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Please note you will receive a warning telling you this. Cooling Coils can only join Cooling Groups.
- **Electrical Collection –** Collections are a way of grouping components that share the same source of energy. For a Fancoil unit, upon choosing an electrical collection, any electrical load due to the fan will be added to that collection's electrical demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Fans in TAS Systems can only join Electrical groups.
- **Zone Position –** This field allows the user to set the position and type of Fancoil Unit. There are three options to choose from:
    - In Zone – The unit is placed in the zone and is not connected to the supply air duct.
    - Terminal Unit – The unit is placed on the supply air duct, meaning that the supply air passes through the unit before entering the zone. The fan in this unit is placed on the air path after the supply air has been merged with the air entering the Fancoil unit from the zone.
    - Parallel Fan Terminal Unit – Like the Terminal Unit, the unit is placed on the supply air duct but the fan is placed in the air path of air entering the Fancoil unit from the zone, before it is merged with the supply air. The placement of the fan leads to a reduced Electrical load compared to a Terminal Unit.
- **Control Method –** This field allows the user to set the control method of the Fancoil unit. There are three options to choose from:
    - CAV – With this option the Fancoil unit will have air entering it with a constant flow rate. The air is then heated and cooled to maintain the temperature in the zone. Please note that during scheduled hours if no heating or cooling is required the fan will still operate and air will still flow through the unit at the constant flow rate.
    - VAV – With this option the Fancoil unit will have air entering it with a variable flow rate. The air is then heated and cooled to maintain the temperature in the zone. Please note that during scheduled hours if no heating or cooling is required the fan will still operate and air will still flow through the unit at the value set in the minimum flow rate.
    - On / Off – With this option the Fancoil unit will only turn on when there is a need for heating and cooling during scheduled hours. During hours where it only needs to be partially on, TAS will work out for how many minutes it will need to be on for and

then takes the average air flow over the hour to display in the hourly figure in the results. When the unit is on, air will enter the Fancoil unit at a constant flow rate.

- **Heating and Cooling Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Please note that for a Fancoil unit, the heating and cooling duties are set separately. This allows you to model a Fancoil unit which just supplies heating or cooling by setting the others duty to zero. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Heating Efficiency** – The Heating Efficiency field of a heating component only appears when a user chooses: an electrical group, a fuel group or the “None” option in the Heating Collection field; as choosing one of these options allows the user to model a component using its energy source directly at the component to produce heat. The Heating Efficiency field allows the user to enter how efficient this process is, as a factor. This field will not have any effect on the air-side results apart from the consumption results for this component will also appear in the results section of this component. Please note while the option appears for the “none” choice in the Heating Collection field, any loads from the component will be discarded. The user will be able to add a modifier to this field by going to the Heating Efficiency tab which also appears when the appropriate options are chosen in the Heating Collection Field.
- **Bypass Factor –** The Bypass Factor field determines the amount of air that will bypass the coils and thus will not be heated or cooled by the coil. The value is entered as a factor between 0 and 1 and this factor is then multiplied against the air flow rate of the air just before the coil to determine the amount of air that will bypass the coils. Modifiers can be added to the bypass factor using the Bypass factor tab.
- **Design Flow Source –** The Design Flow Source option for a zone component allows the user to set the design flow rate for the component. When the Fancoil Unit control method is set to “CAV” or “On/Off” the Design Flow Rate will be the flow rate of air through the Fancoil unit. When the control method is set to “VAV” the design flow rate will be the maximum allowed flow rate of air through the Fancoil unit, with the minimum flow rate set by the Minimum Flow Source option. With the Fancoil unit component, the user has the following options for the Design Flow Source field:
    - Value – Upon choosing this option the user will be asked to enter the design flow rate.
    - Sized - Upon choosing this option, the user is asked to choose from the following seven sizing methods to set the design flow Rate:
        - Per Floor Area – Here TAS Systems asks the user to enter a value in $l/s/m^2$ (cfm / ft^2 in U.S. customary units). This value is then multiplied by the area of the zone to give the flow rate in l/s (cfm in U.S. customary units). Please note that the user can use the size fraction field to over-size or under-size this value.
        - Per Volume – Here TAS Systems asks the user to enter a value in $l/s/m^3$ (cfm / ft^3 in U.S. customary units). The value is then multiplied by the volume of

the zone to give the flow rate. Please note that the user can use the size fraction field to over-size or under-size this value.

- ACH – The Value field here stands for how many air changes per hour the user wishes to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person – TAS Systems asks the user to enter an air rate per person. TAS then uses the following formula to give the sized flow rate:

$$\textit{Sized Flow Rate} = \frac{\textit{Peak Occupancy Sensible Gain} + \textit{Peak Occupancy Latent Gain}}{\textit{Metabolic Rate}} * \textit{Area of zone} * \textit{Value},$$

where the: Peak Occupancy Sensible Gain, Peak Occupancy Latent Gain and Metabolic Rate are all read from the zone's internal condition and value is the value entered in Systems for the air rate per person. Please note that the user can use the size fraction field to over-size or under-size this value.

- Peak Person and Area – The Peak Person and Area method allows the user to size the flow rate on both the Peak Person and Per Floor Area methods. The user is asked to enter in their flow rates per person and per floor area. TAS then uses the value entered into the Per Person field in the Peak Person method and the value entered into the Per Area field in the Per Floor Area method. TAS will then add the results of both of these methods together and report the sum as the flow rate.
- Peak Internal Condition – Please note that the Peak Internal Condition method works like the Peak Person method, but instead of asking the user for the air rate per person it uses the Outside Air rate from the zone's Internal Condition instead. Please note that the user can use the size fraction field to over-size or under-size this value.
- Delta T – The Delta T method is the default method for sizing the flow rate for a zone. Upon choosing this method, the user is asked to enter a heating delta T and a cooling delta T. The heating delta T value entered should be:

$$\textit{Max Temp of the supply air} - \textit{The zone's thermostat lower limit},$$

where the zone's thermostat is read from the Internal Condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air heated to this maximum temperature to keep the zone's temperature above the lower limit. Please note that this is done using the heating loads from the TSD file, as this sizing is done before the simulation. The cooling delta T works in a similar way with the zone's thermostat upper limit. The cooling delta T value entered should be equal to:

$$\textit{The Zone's thermostat upper limit} - \textit{Min temp of the supply air},$$

where the zone's thermostat is read from the internal condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air cooled to this minimum temperature to keep the zone's temperature below the upper limit. Please note that this is done using the cooling loads from the TSD file, as this sizing is done before the simulation. Upon working out the flow rate needed for the heating delta T and the cooling delta T, TAS will take the max of the two as the flow rate for the zone. Please note that the values entered here do not impact the supply air temperatures. It is just used to work out the sized flow rate.

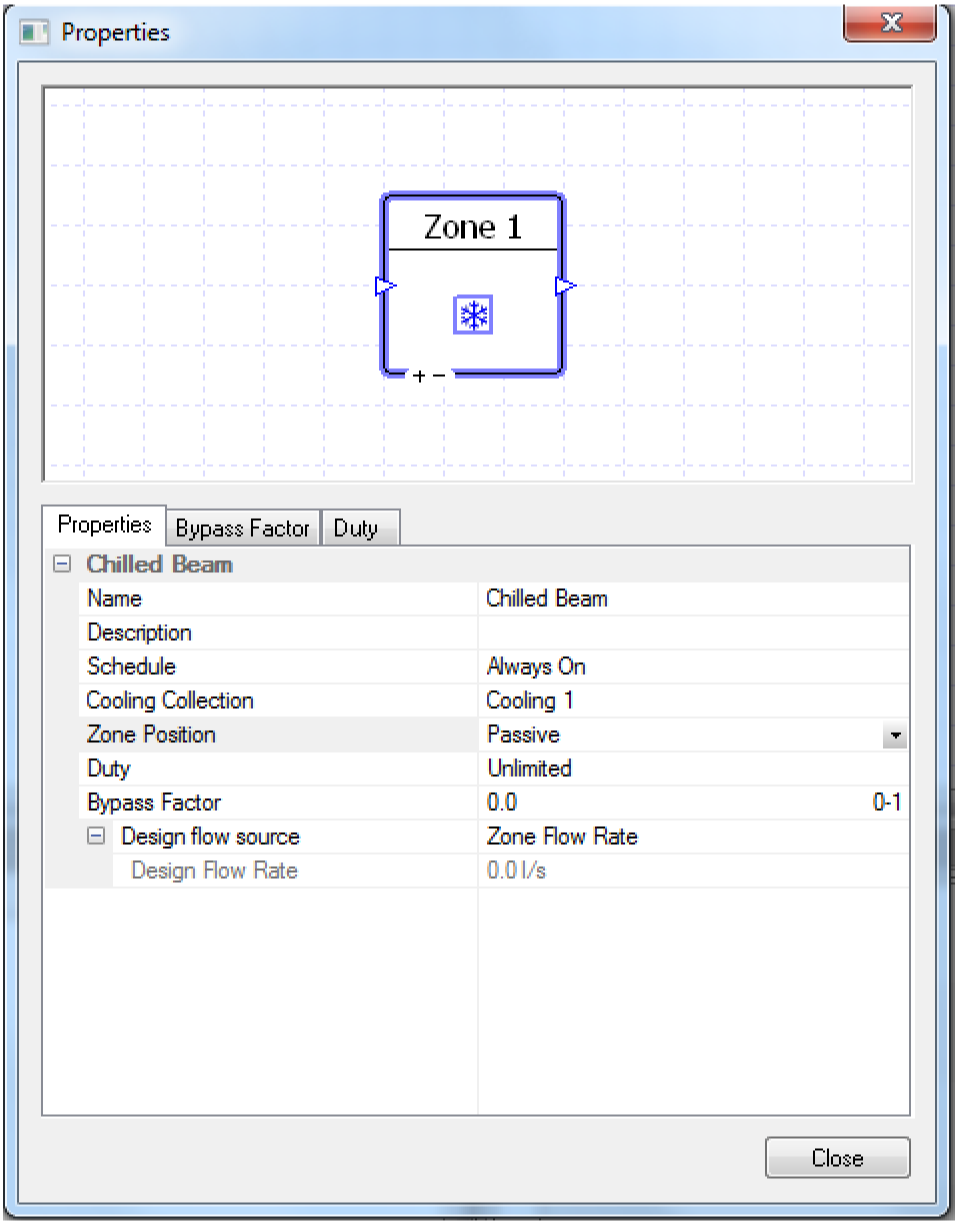
After entering the delta T values, the user will be asked to choose the design condition (or just the simulation data) to choose what days to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

- Zone Flow Rate – When this option is chosen, TAS will take the zone's target flow rate as the flow rate through the component.

  - Zone Fresh Air – When this option is chosen, TAS will take the zone's target Fresh Air rate as the flow rate through the component.
- **Minimum Flow Source –** The Minimum Flow Source option for a zone component allows the user to set the minimum flow rate for the component. Please note that for a Fancoil unit this option only appears when the Control Method option is set to "VAV". The Minimum Flow Source field gives the same four options as the Design Flow Source field but sets the minimum flow rate instead of the design flow rate. However, it should be noted that with the sized method there are an extra 3 sizing methods to choose from: Hourly Person, Hourly Person and Area, and Hourly Internal Condition. These sizing methods work like the peak methods but are calculated for each hour.
- **Fan Overall Efficiency –** The Fan Overall Efficiency field of the Fancoil unit allows the user to enter the overall efficiency of the fan in the Fancoil unit. The overall efficiency of the fan is the product of its: motor efficiency, electrical efficiency, belt efficiency and aerodynamic efficiency (as there will be aerodynamic losses in the fan). Please note that this does not include any partload efficiency, which is set by the Partload field. The user will need to enter the overall efficiency as a factor between zero and one.
- **Fan Heat Gain Factor –** The heat gain factor of the fan is the proportion of energy used to power the fan that is converted to heat. The generated heat will warm up the air that passes through the fan.
- **Fan Part Load -** Clicking on the Fan Partload field will take you to the Fan Partload Tab. From this tab, you will be able to edit the Partload profile of the Fancoil unit's fan by using the graph or the table. To see how to edit the Partload profiles, please watch the Profiles video in the TAS Systems User Guide.
- **Fan Pressure –** The Fan Pressure field allows the user to enter the Fan Pressure. The formula for Fan Pressure is:

$$\boldsymbol{Fan\ Pressure\ =\ SFP\ *\ 1000 * Overall\ Efficiency\ .}$$

3.3 Chilled Beam



A Chilled Beam component can be used to model an active or passive chilled beam. Cold water is pumped through the beam and cools the surrounding air, which then cools down the space by convection. Please note that the user will need to edit the radiant proportion of the cooling emitter in the zone's internal condition to model the chilled beam correctly. You cannot use controllers with this component and it will always cool the space to the upper limit of the zones thermostat.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the chilled beam, this will mean that the beam will not cool down the

zone if the zone's temperature increases above the upper limit. The default schedule option is always on, meaning that the component will operate 24/7.

- **Cooling Collection –** Collections are a way of grouping components that share the same source of energy. Once a component is added to a Cooling collection, any cooling load it has will be added to that collection's cooling demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Please note you will receive a warning telling you this. Cooling Coils can only join Cooling Groups.
- **Zone Position –** This field allows the user to set up the position of the chilled Beam in their zone. The two options are:
  - Active – The active chilled beam is connected to the supply air ductwork. This means that some of the supply air to the zone enters through the chilled beam. This induces warm zone air into the chilled beam to be cooled.
  - Passive – The passive chilled beam is suspended from the ceiling. It relies on natural convection for air to pass through it.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand**.** For the chilled beam this would mean it wouldn't be able to cool the air to the upper limit of the zones thermostat, it would fall short. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
  - Unlimited – Unlimited means the component is always able to meet the demand.
  - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
  - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Bypass factor –** The Bypass Factor field determines the amount of air that will bypass the beam and thus will not be cooled by the beam. The value is entered as a factor between 0 and 1 and this factor is then multiplied against the air flow rate of the air just before the beam to determine the amount of air that will bypass the beam. Modifiers can be added to the bypass factor using the Bypass factor tab.
- **Design Flow Source** – The design flow source option for a zone component allows the user to set the flow rate of the air through the component. For a chilled beam, this will be the amount of air that passes through the beam. With a zone component, the user has the following options for the Design Flow Source field:
  - Value – Upon choosing this option the user will be asked to enter the design flow rate.
  - Sized - Upon choosing this option, the user is asked to choose from the following seven sizing methods to set the design flow Rate:
    - Per Floor Area – Here TAS Systems asks the user to enter a value in $l/s/m^2$ (cfm / ft^2 in U.S. customary units). This value is then multiplied by the area of the zone to give the flow rate in l/s (cfm in U.S. customary units). Please note that the user can use the size fraction field to over-size or under-size this value.
    - Per Volume – Here TAS Systems asks the user to enter a value in $l/s/m^3$ (cfm / ft^3 in U.S. customary units). The value is then multiplied by the volume of the zone to give the flow rate. Please note that the user can use the size fraction field to over-size or under-size this value.

- ACH – The Value field here stands for how many air changes per hour the user wishes to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person – TAS Systems asks the user to enter an air rate per person. TAS then uses the following formula to give the sized flow rate:

$$\boldsymbol{Sized\ Flow\ Rate} = \frac{\boldsymbol{Peak\ Occupancy\ Sensible\ Gain + Peak\ Occupancy\ Latent\ Gain}}{\boldsymbol{Metabolic\ Rate}} * \boldsymbol{Area\ of\ zone} * \boldsymbol{Value},$$

  where the: Peak Occupancy Sensible Gain, Peak Occupancy Latent Gain and Metabolic Rate are all read from the zone's internal condition and value is the value entered in Systems for the air rate per person. Please note that the user can use the size fraction field to over-size or under-size this value.
- Peak Person and Area – The Peak Person and Area method allows the user to size the flow rate on both the Peak Person and Per Floor Area methods. The user is asked to enter in their flow rates per person and per floor area. TAS then uses the value entered into the Per Person field in the Peak Person method and the value entered into the Per Area field in the Per Floor Area method. TAS will then add the results of both of these methods together and report the sum as the flow rate.
- Peak Internal Condition – Please note that the Peak Internal Condition method works like the Peak Person method, but instead of asking the user for the air rate per person it uses the Outside Air rate from the zone's Internal Condition instead. Please note that the user can use the size fraction field to over-size or under-size this value.
- Delta T – The Delta T method is the default method for sizing the flow rate for a zone. Upon choosing this method, the user is asked to enter a heating delta T and a cooling delta T. The heating delta T value entered should be:

$$\boldsymbol{Max\ Temp\ of\ the\ supply\ air - The\ zone's\ thermostat\ lower\ limit},$$

  where the zone's thermostat is read from the Internal Condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air heated to this maximum temperature to keep the zone's temperature above the lower limit. Please note that this is done using the heating loads from the TSD file, as this sizing is done before the simulation. The cooling delta T works in a similar way with the zone's thermostat upper limit. The cooling delta T value entered should be equal to:

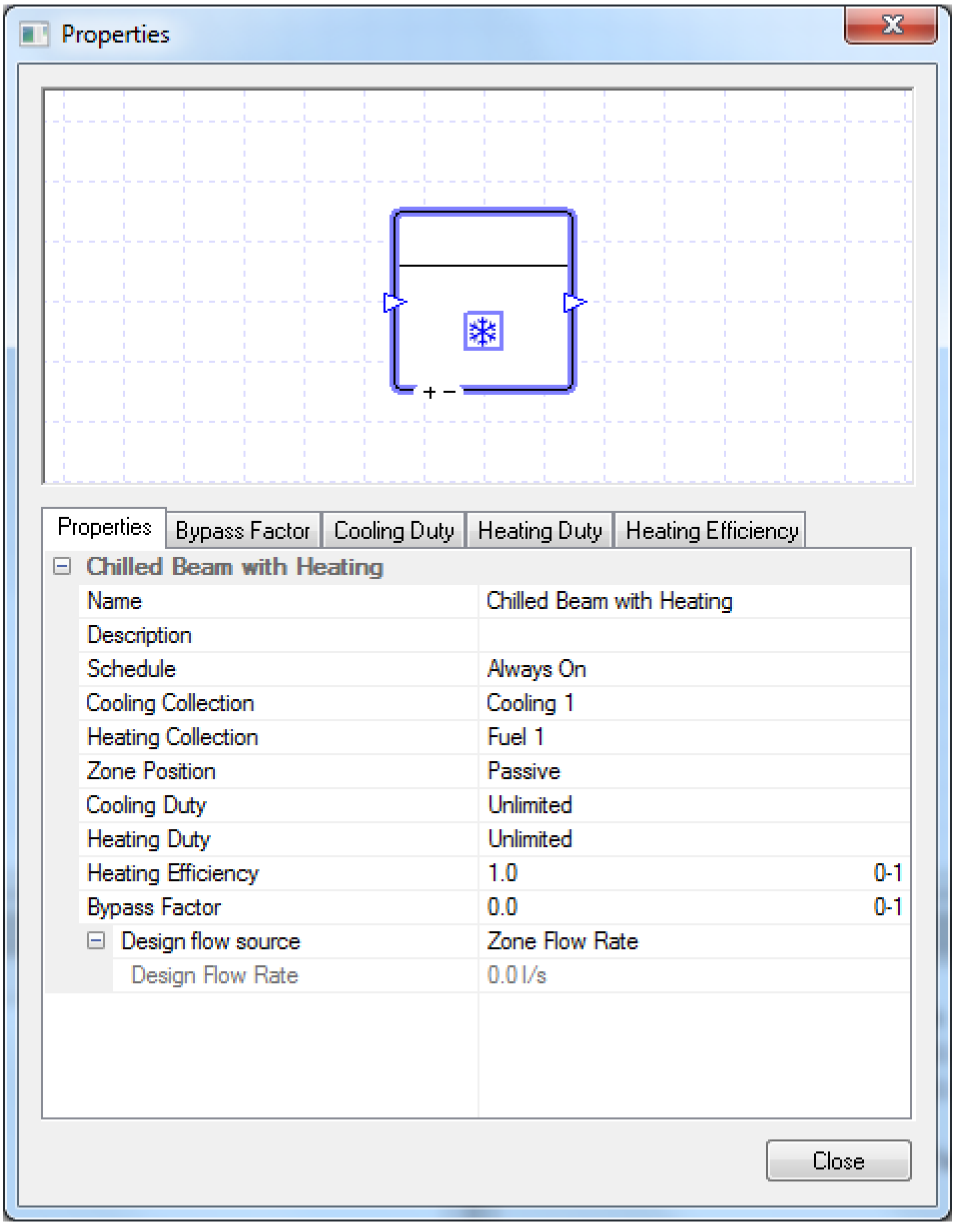
$$\boldsymbol{The\ Zone's\ thermostat\ upper\ limit - Min\ temp\ of\ the\ supply\ air},$$

  where the zone's thermostat is read from the internal condition of the zone. TAS Systems will then work out what is the maximum flow rate required with the supply air cooled to this minimum temperature to keep the zone's temperature below the upper limit. Please note that this is done using the cooling loads from the TSD file, as this sizing is done before the simulation. Upon working out the flow rate needed for the heating delta T and the cooling delta T, TAS will take the max of the two as the flow rate for the zone. Please note that the values entered here do not impact the supply air temperatures. It is just used to work out the sized flow rate.

  After entering the delta T values, the user will be asked to choose the design condition (or just the simulation data) to choose what days to size the flow rate on. Please note that the user can use the size fraction field to over-size or under-size this value.

- Zone Flow Rate – When this option is chosen, TAS will take the zone's target flow rate as the flow rate through the component.
- Zone Fresh Air – When this option is chosen, TAS will take the zone's target Fresh Air rate as the flow rate through the component.

3.4 Chilled Beam with Heating



The Chilled Beam with Heating component acts similarly to the Chilled Beam component with the difference being it can also heat the air. Just as with the chilled beam, you will need to set up the correct radiant proportion for the heating and cooling emitter in the building simulator file. You can also use this component to model a heated beam, by setting the cooling duty of the component to zero. As this component's properties are very similar to the chilled beam, I will only discuss the additional options. Please note that the Bypass Factor field will only affect the results when the beam is cooling and the Cooling Duty field for the chilled beam with heating is the same as the Duty field for the chilled beam.

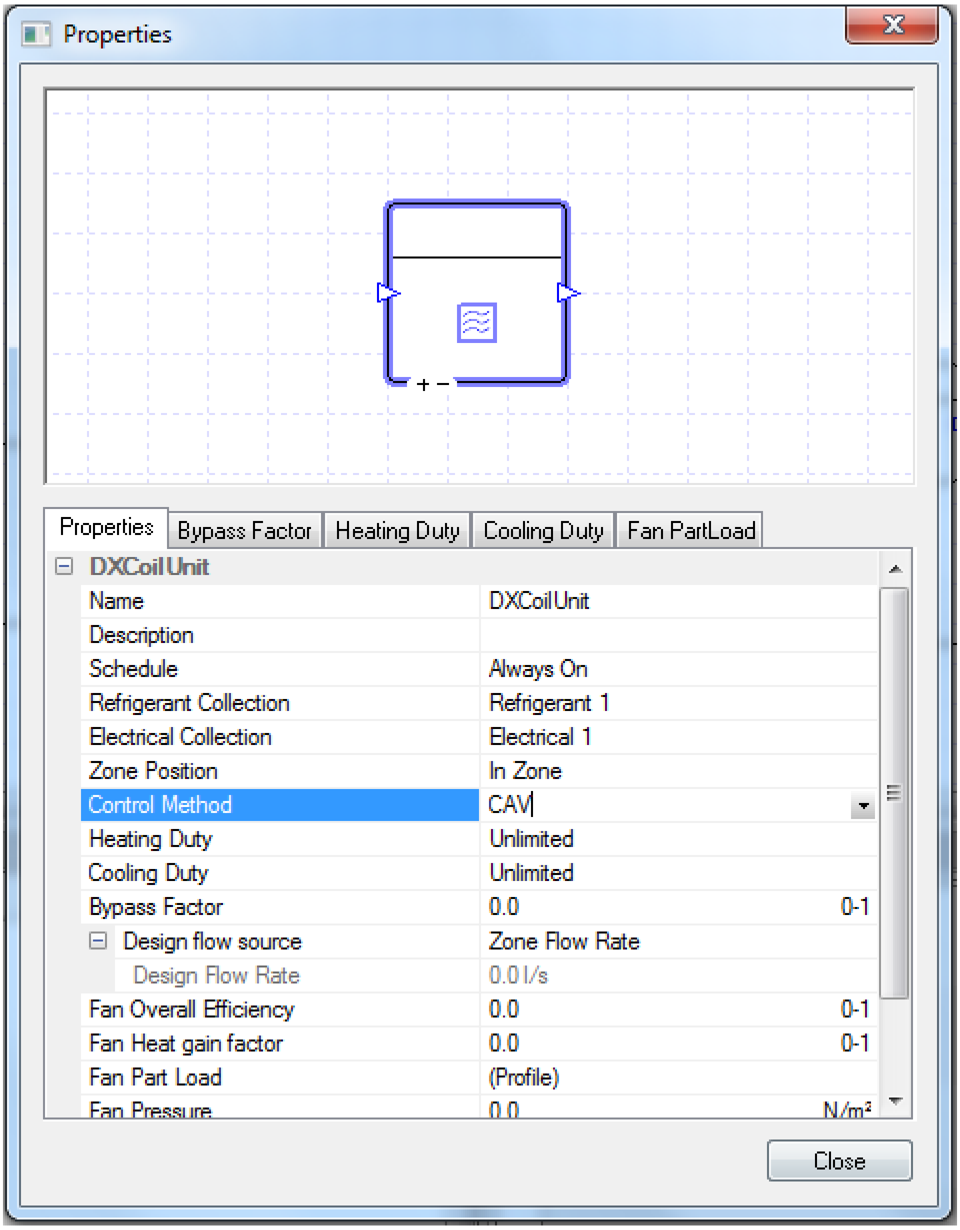
Properties:

- **Heating Collection –** Collections are a way of grouping components that share the same source of energy. Once the chilled beam with heating is added to a Heating, Fuel or Electrical collection, any heating load it has will be added to that collection's demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. Please note you will receive a warning telling you this. For the chilled beam with heating, the heating collection only deals with the energy used by the beam for heating. With the Heating Collection field, you will be able to join: Heating, Fuel and Electrical groups. For instance if the heat is being provided to the beam from a boiler, you would choose a heating group but if the beam was generating the heat itself by burning fuel, you would choose a fuel group.

- **Heating Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand**.** For the chilled beam with heating, setting the heating duty too low would mean it wouldn't be able to heat the air to the upper limit of the zones thermostat, it would fall short. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction. Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Heating Efficiency** – The Heating Efficiency field of a heating component only appears when a user chooses: an electrical group, a fuel group or the "None" option in the Heating Collection field; as choosing one of these options allows the user to model a component using its energy source directly at the component to produce heat. The Heating Efficiency field allows the user to enter how efficient this process is, as a factor. This field will not have any effect on the air-side results apart from the consumption results for this component will also appear in the results section of this component. Please note while the option appears for the "none" choice in the Heating Collection field, any loads from the component will be discarded. The user will be able to add a modifier to this field by going to the Heating Efficiency tab which also appears when the appropriate options are chosen in the Heating Collection Field.

3.5 DX Coil Unit



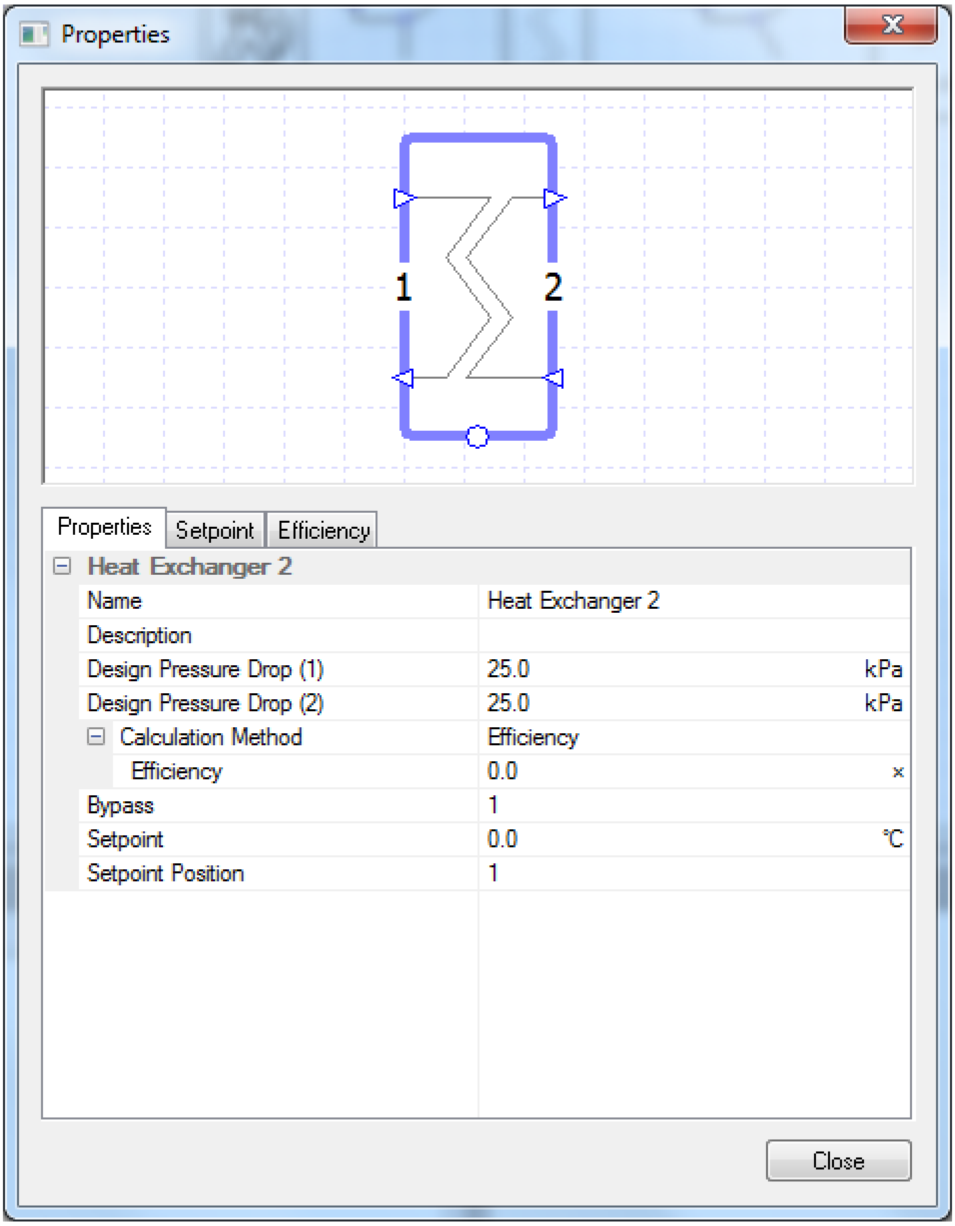
A DX Coil unit is a Fancoil unit but instead of having a heating coil and a cooling coil, it has a DX coil which provides both heating and cooling. As the DX Coil unit properties are mostly the same as the Fancoil unit I will only discuss the one property that isn't in the Fancoil unit's properties, the Refrigerant Collection property. This replaces the Heating and Cooling Collection fields in the Fancoil unit's Properties. You cannot use controllers with this component and it will always heat and cool the space to the lower and upper limit of the zone's thermostat.

Property:

- **Refrigerant Collection -** Collections are a way of grouping components that share the same source of energy. Once a component is added to a Refrigerant collection, any heating or cooling load it has will be added to that collection's refrigerant demand in the plant room. If a component is not assigned to a collection, the energy it uses will be discarded and not reported in the results. As the DX Coil uses refrigerants, you can only join refrigerant Groups.

4 Plant Room Components

4.1 Heat Exchanger



The Plant Room heat exchanger exchanges heat between two different fluid loops. Unlike the heat exchanger in the air-side components, the fluid that enters on the supply port on one side will exit the heat exchanger on the return port on the same side. This means that the heat exchanger has two loops, which we shall call Loop 1 and Loop 2. When you click on the heat exchanger, the numbers 1 and 2 appear to detail which loop is Loop 1 and which loop is Loop 2. Properties that require different values for each loop put the loop's number in brackets after the field's name. Provided that a bypass has been set up on the heat exchanger, controllers can be used with the heat exchanger, and they will control how much the exchanger will exchange heat between the two fluid loops. If a standard controller sending a signal between zero and one is attached to the heat exchanger, the signal specifies the amount of fluid that will flow through the exchanger on the loop with the bypass. A signal of zero means that all fluid on the loop with the bypass will bypass the exchanger, allowing for no heat transfer, while a signal of one means that all fluid on the loop with the bypass will flow through the exchanger, allowing for the maximum amount of heat transfer. A signal in-between the two implies a partial bypass of the loop with the bypass, with a lower signal indicating more fluid bypassing the exchanger, reducing the amount of heat transfer. If no controller is used, the sensor used to measure the temperature of the fluid for the Setpoint field is assumed to be directly after the return port on the loop decided by the Setpoint Position field.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid when passing through the component on Loop 1. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on Loop 1's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) –** This field allows the user to enter the pressure drop of the fluid when passing through the component on Loop 2. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on Loop 2's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Calculation Method –** In TAS systems, there are two options to calculate the maximum rate of heat transfer for the Exchanger:
    - Efficiency – The first calculation method is efficiency. Upon choosing this option the user is asked to enter the efficiency of the exchanger. The efficiency of the exchanger is calculated by the following formula:

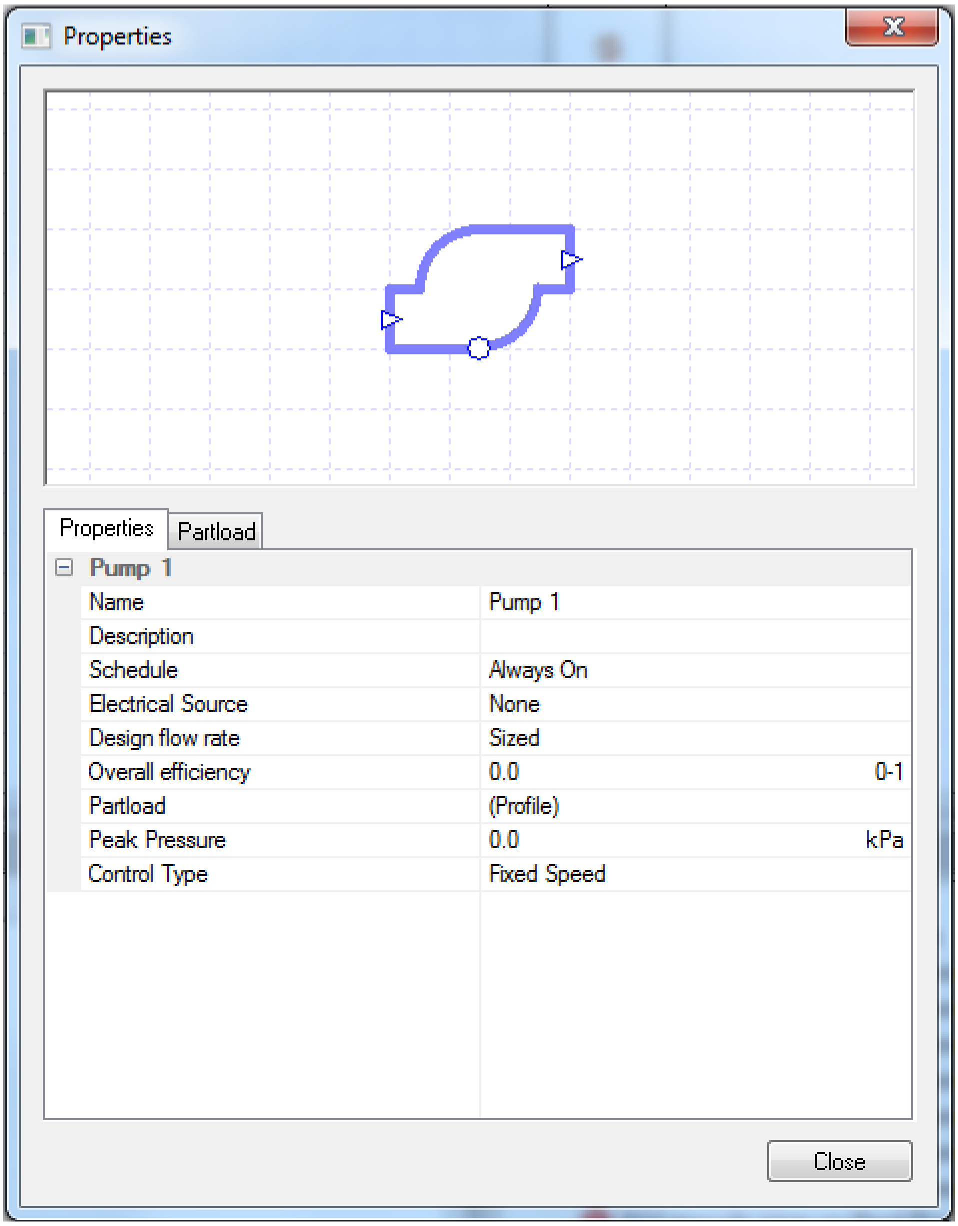
      $$\boldsymbol{Efficiency} = \mathbf{1} - \frac{\boldsymbol{Higher\ Supply\ Temp} - \boldsymbol{Lower\ Return\ Temp}}{\boldsymbol{Higher\ Supply\ Temp} - \boldsymbol{Lower\ Supply\ Temp}},$$

      where: $\boldsymbol{Higher\ Supply\ Temp}$ is the maximum of the two supply temperatures, $\boldsymbol{Lower\ Return\ Temp}$ is the minimum of the two return temperatures and $\boldsymbol{Lower\ Supply\ Temp}$ is the minimum of the two supply Temperatures. Please note that in the Efficiency Tab, you can add a modifier to the efficiency.
    - NTU Method – The second calculation method is the NTU method. Upon choosing this method, the user will need to input: The heat transfer surface area, the heat transfer coefficient and the Exchanger type. After entering these details, TAS will then work out the rate of heat transfer for the exchanger.
- **Bypass –** This field allows the user to set up a bypass of the heat exchanger for one of the loops when certain conditions are met; for instance, to stop a loop from exceeding a certain temperature. If the "none" option is chosen, no bypass will be modelled. If the user chooses "1", Loop 1 of the exchanger will be modelled with the bypass. If the user chooses "2", Loop 2 of the exchanger will be modelled with the bypass. When the bypass operates will be determined by the properties entered into the Setpoint Position field, or by any controllers connected to the heat exchanger.
- **Setpoint Fields –** The Setpoint fields only appear when a controller is not connected to the heat exchanger, and the number of setpoint fields will depend on the selected option in the Setpoint Position field. The setpoint entered into these fields will be used to control the bypass and thus will regulate the temperature on the fluid loops. Please note that how the setpoint will work will depend on the temperature of the fluid entering the exchanger on that loop. So if the temperature entering the exchanger on a loop was below the setpoint temperature, then if the heat exchanged could lead to the temperature of the fluid exiting the exchanger on that loop to go above the setpoint temperature, the bypass would be enforced to make sure the fluid leaves at the setpoint temperature. Similarly, if the temperature entering the exchanger on that loop was above the setpoint temperature, if

the heat exchanged could lead to the temperature of the fluid exiting the heat exchanger going below the setpoint temperature, the bypass would be enforced. When the setpoint fields are visible, modifiers can be applied to the setpoint using the appropriate Setpoint tabs.

- **Setpoint Position -** The Setpoint Position field is only available when a controller is not connected to the heat exchanger. The setpoint position allows the user to set under what conditions the exchanger's bypass will operate, from the following four options:
    - 1 – This option will control the heat exchanger's Loop 1. To do this, a sensor is placed directly after the return port on Loop 1 and one Setpoint field will appear. The setpoint entered into this field will be the temperature fluid on Loop 1 will be controlled to by the heat exchanger.
    - 2– This option will control the heat exchanger's Loop 2. To do this, a sensor is placed directly after the return port on Loop 2 and one Setpoint field will appear. The setpoint entered into this field will be the temperature fluid on Loop 2 will be controlled to by the heat exchanger.
    - 1 Over 2 – This option will control the temperature on the heat exchanger's Loop 1 but sets a fluid temperature for Loop 2 that cannot be passed, even if the temperature on Loop 1 has not yet reached the setpoint temperature. To do this, a sensor is placed directly after the return port on Loop 1 and the setpoint entered into the Setpoint field will be the temperature fluid on Loop 1 will be controlled to by the heat exchanger. A sensor will also be placed directly after the return port on Loop 2, and the setpoint entered into the Setpoint 2 field will be the temperature the fluid on Loop 2 cannot pass.
    - 2 Over 1 – This option will control the temperature on the heat exchanger's Loop 2 but sets a fluid temperature for Loop 1 that cannot be passed, even if the temperature on Loop 2 has not yet reached the setpoint temperature. To do this, a sensor is placed directly after the return port on Loop 2 and the setpoint entered into the Setpoint field will be the temperature fluid on Loop 2 will be controlled to by the heat exchanger. A sensor will also be placed directly after the return port on Loop 1, and the setpoint entered into the Setpoint 2 field will be the temperature the fluid on Loop 1 cannot pass.

4.2 Pump



Pump components in TAS Systems allow the user to provide and control the flow rate of the fluid in the Plant Room. Controllers can be used with the pump component, and will control how much the pump will exert pressure on the fluid flowing through it. The signal received by the pump dictates the proportion of the maximum pressure increase the pump will increase the pressure to. Please note that this maximum pressure increase is not the value entered in the pumps' Peak Pressure field. Instead the software will work the maximum pressure increase out by taking into account the Peak Pressure field of the pump, along with the design flow rates and design pressure drops of other components around the system. Also, as pressure and flow rate are linked, controlling the pressure with a controller will also mean that the flow rate will vary. How the signal received by the pump affects the flow rate is shown using the equation below

$$\boldsymbol{Flow\ Rate\ at\ signal = Design\ Flow\ Rate * \sqrt{signal}.}$$

Commonly the sensor associated with the pump is placed onto the Collection and the controller is sensing the load of the Collection, turning the pump off when there's no demand.

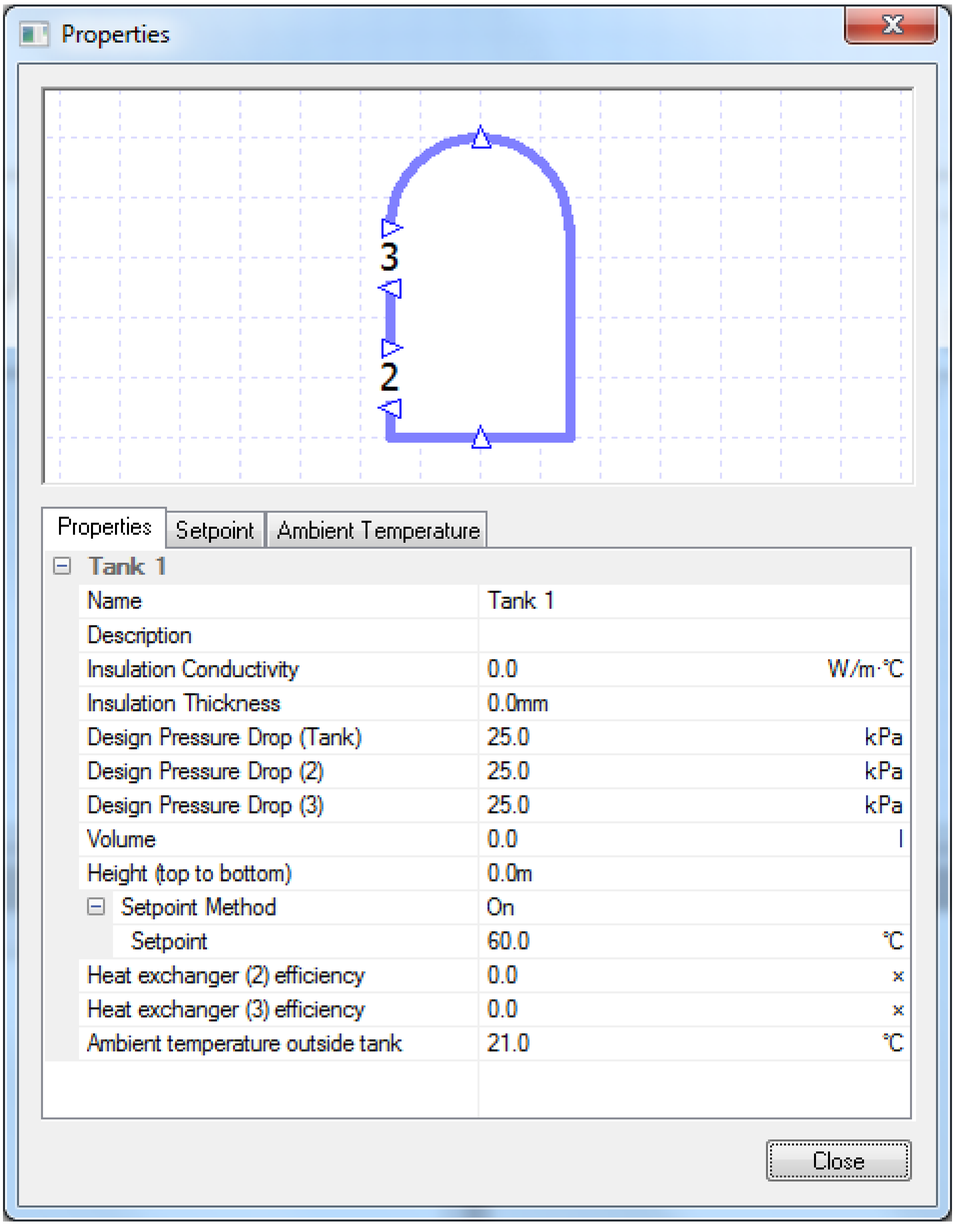
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the pump, this will mean that the pump will not provide any fluid flow. If all of your pumps in your system are scheduled to be off in the same hour then there will be no fluid flow in your system. The default schedule option is always on, meaning that the component will operate 24/7.
- **Electrical Source –** The Electrical Source field allows the user to decide the source of electricity for the Pumps. The options given here will be any fuel source that is in the fuel source folder, so the user will be able to choose a non-electrical fuel source but this is not advised. If the user chooses the “None” option, the user will receive a warning and the pump’s load will be discarded from the results.
- **Design Flow Rate –** The Design Flow Rate field allows the user to set the design flow rate of the plant room fluid path the pump is on. When the pump is on a loop with a collection with the Variable Flow Rate Option set to “No”, and the pump is not connected to a controller set up to vary the flow rate, then the design flow rate will be the flow rate of the fluid path. If; the Variable Flow Rate option is set to “Yes”, the pump is on the same circuit as a valve which is varying the flow, or the pump is connected to a controller set up to vary the flow then the design flow rate is the maximum flow rate of the fluid path. There are two options available to set the design flow rate:
    - Value – Upon choosing this option the user will enter the desired Design Flow Rate.
    - Auto – Upon choosing this option the design flow rate will be sized upon the properties of components / collections with a Design Flow Delta T field, which are on the same circuit loop as the Pump. If the component has “none” entered in the Design Flow Delta T field, it will be ignored for the sizing of the design flow rate by the pump. If a value is entered, TAS will size the design flow rate using the Design Flow Delta T field along with the peak load of the component. The sizing is done such that the fluid is kept within the delta T temp limit when passing through the component and so that the demand of the component / collection is met. Please note that for a collection it will just use the Design Flow Delta T field. If a Design Flow Delta T is entered into 2 or more components / collections on the same circuit as the pump, TAS will use the component / collection which results in the larger design flow rate to size the design flow rate.
- **Overall Efficiency –** The overall efficiency of the pump is its motor efficiency multiplied by the electrical efficiency. Please note that this does not include any partload efficiency, which is set by the Partload field. The user will need to enter the overall efficiency as a factor between zero and one.
- **Partload –** Clicking on the Partload field will take you to the Partload Tab. From this tab, you will be able to edit the Partload profile of the component by using the graph or the table. To see how to edit the Partload profiles, please watch the Profiles video in the TAS Systems User Guide.
- **Peak Pressure –** The Peak Pressure of the pump is the upper limit on the amount of pressure the pump can generate. The user should bear in mind that the peak pressure needs to satisfy the following equation, or otherwise they will receive a flow sizing error:

$$\boldsymbol{Peak\ Pressure\ of\ thePump} > \sum \begin{matrix}\boldsymbol{Pressure\ Drops\ of\ All}\\ \boldsymbol{Components\ on\ the\ Loop}\end{matrix}.$$

- **Control Type –** This field allows the user to set the control type of the pump. This option is only available when there is no controller attached to the pump. There are two options to choose from:
    - Variable Speed – With this option the pressure generated at the Pump will match the sum of the pressure drops of the circuit, even if the peak pressure is larger.
    - Fixed Speed – The pressure generated at the pump will depend on the Variable Flow Option at the collection. If the Variable Flow Option is set to "No", then the pressure generated at the pump will be equal to the sum of the pressure drops. If the Variable Flow Option is set to "Yes" then the Pressure generated at the Pump will vary between the Peak Pressure and the sum of all Pressure Drops on the circuit.

4.3 Tank



The Tank component allows for the storage of fluids in TAS Systems. The tank has three fluid loops which connect to it. The first loop, known as the Tank loop, models the fluid being stored in the tank. This loop enters the Tank component at the bottom port and exits the Tank component at the top port. The other two loops, called Loop 2 and Loop 3, attach to the side of the tank and are heat exchanger loops which allow for heat transfer to the stored fluid. When you click on the Tank component, the numbers 2 and 3 appear to denote which ports correspond to Loop 2 and which ports correspond to Loop 3. Please note that it is not required to use both heat exchanger loops, but you do need to use at least one of them. It is assumed that the tank is cylindrical. No controllers may be used with this component.

Properties:

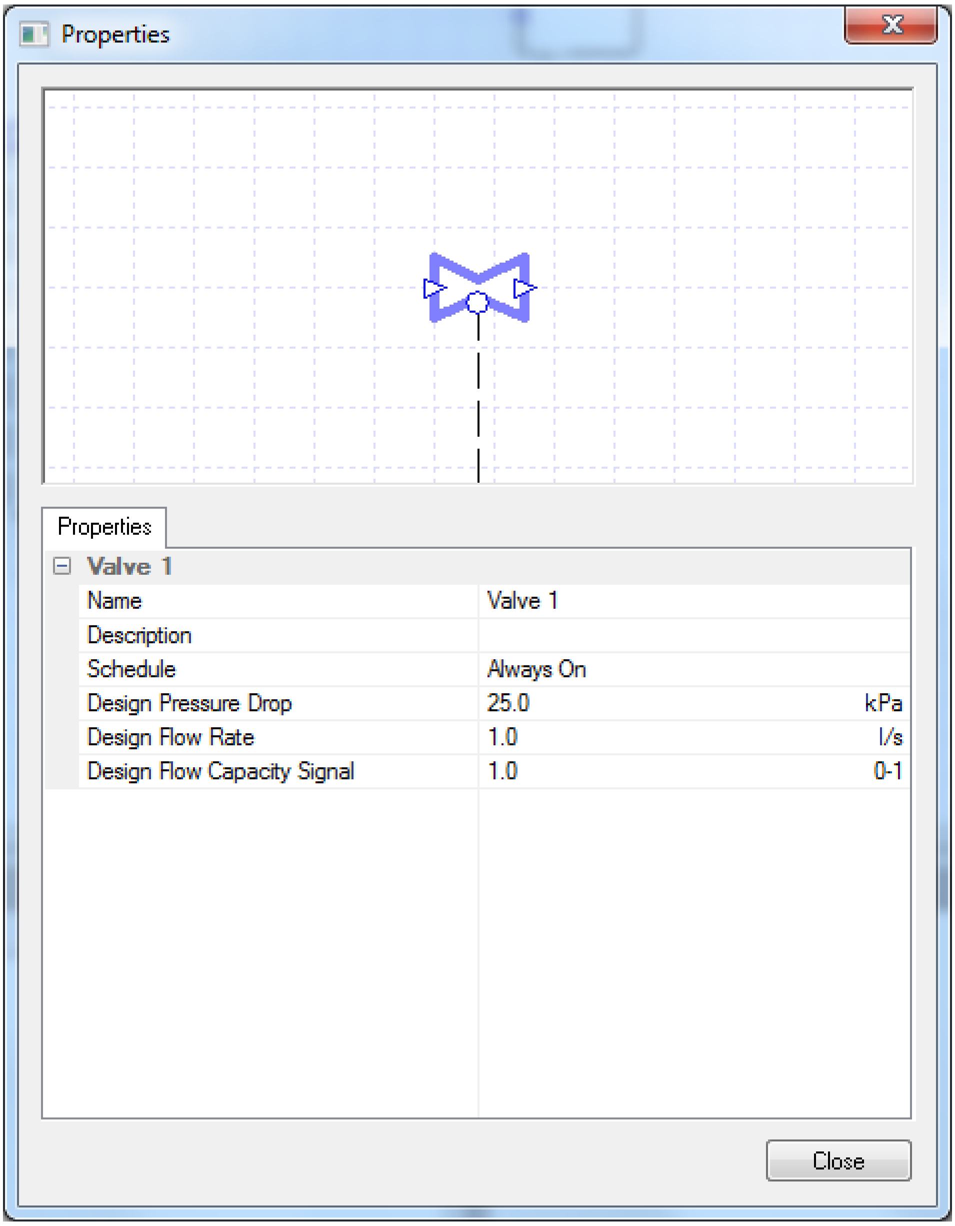
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

- **Insulation Conductivity –** The user will need to enter the thermal conductivity of the tank's insulation.
- **Insulation Thickness –** In this field, the user enters the thickness of the tank's insulation.
- **Design Pressure Drop (Tank) -** This field allows the user to enter the design pressure drop of the fluid being stored in the tank; which is the fluid entering the tank from the bottom port and exiting from the top port. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) –** This field allows the user to enter the design pressure drop for the fluid on Loop 2 when passing through the Tank component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (3) –** This field allows the user to enter the design pressure drop for the fluid on Loop 3 when passing through the Tank component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Volume –** In this field the user enters the volume of the tank.
- **Height (Top to Bottom) –** This field allows the user to enter the height of the tank.
- **Setpoint Method –** With this field the user can decide their setpoint method. There are two choices:
  - Off – When set to Off, there is no setpoint for the temperature of the fluid in the tank. This means that heat will be exchanged by the heat exchanger loops for all operating hours.
  - On – Upon choosing this option, the user will be asked to enter a temperature as a setpoint. Once the fluid in the tank hits this temperature setpoint, the exchanger loops will bypass the tank to avoid any additional heating.
- **Heat Exchanger (2)** – The Heat Exchanger (2) field allows the user to set the efficiency of the heat transfer between the tank and Loop 2. The efficiency is set by the following formula:

  $$\boldsymbol{Efficiency} = \frac{\boldsymbol{Loop2_{Return}} - \boldsymbol{Loop2_{Supply}}}{\boldsymbol{Tank_{Mean}} - \boldsymbol{Loop2_{Supply}}},$$

  Where: $\boldsymbol{Loop2_{Return}}$ is the temperature of the fluid leaving the tank on Loop 2, $\boldsymbol{Loop2_{Supply}}$ is the temperature of the fluid entering the tank on Loop 2 and $\boldsymbol{Tank_{Mean}}$ is the mean temperature of the tank for the hour which can be found by checking the temperature of the fluid leaving the tank at the top port.
- **Heat Exchanger (3) –** The Heat Exchanger (3) field allows the user to set the efficiency of the heat transfer between the tank and Loop 3. Please note that that the efficiency formula in the Heat Exchanger (2) field is used to calculate the efficiency here as well but with Loop 3 replacing all mentions of Loop 2.
- **Ambient Temperature Outside Tank –** The user will input here the average temperature of the area where the tank is stored. The user can go to the Ambient Temperature tab to set up a modifier with this field. They can also drag a zone, from the TSDData folder, onto the tank and Systems will use the zone's temperature as the ambient temperature for each hour of the simulation.

4.4 Valve



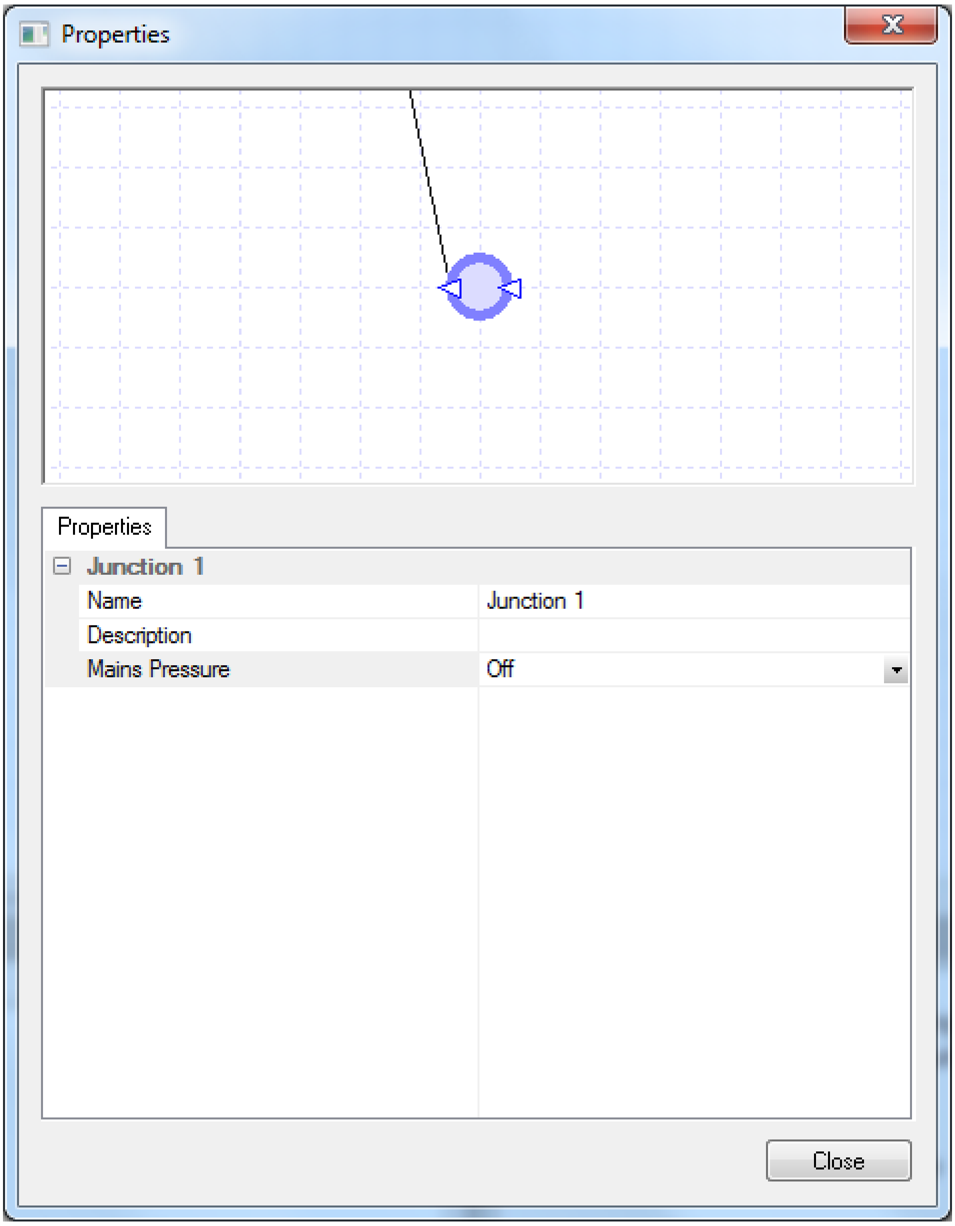
The Valve component for the plant room acts similarly to the Damper component for air side systems. They are mainly used after splitting fluid paths, modulating the flow on each new path. Please note that unlike the damper, the design flow rate must be entered manually here. So when splitting fluid paths you will need to check manually that the design flow rate on both sides, before and after the split, match. Controllers can be used with the valve, and they will control the capacity of the valve. The signal the valve receives dictates the proportion of the maximum capacity that the valve will close to, where the maximum capacity is set by either the Design Flow Rate field or the Capacity field of the valve. So for example, if the valve receives a signal of 1 from the controller, the valve will open up and allow fluid through it according to its maximum capacity. If the valve receives a signal of zero from the controller, the valve will close completely and will not allow any fluid to pass through it. It should be noted that as the capacity of the valve varies, the flow rate and pressure of the fluid around the system will also vary.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the valve, this will mean that the valve will close and will not allow any fluid to flow through it. This will mean any other component on the same fluid path as the valve will not receive any fluid flow in this hour. The default schedule option is always on, meaning that the component will operate 24/7.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid when passing through the valve. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this valve.
- **Design Flow Rate -** The Design Flow Rate field allows the user to set the design flow rate of the fluid path the valve is on. When the valve is on a loop with a collection with the Variable Flow Option set to "No", then the design flow rate will be the flow rate of the fluid path. If the Variable Flow Rate option is set to "Yes" then the design flow rate is the maximum flow rate of the fluid path. Please also note that the Design Flow Rate field for a pump and valve on the same fluid path must match. If the pump's Design Flow Rate field is set to "auto" and a value is entered in the valve's Design Flow Rate field, when the valve is on the same fluid path as the pump, then the pump will take the valve's design flow rate. There are two options to set the design flow rate:
    - Auto – Upon choosing this option the design flow rate will be sized upon the properties of components / collections with a Design Flow Delta T field, which are on the same circuit loop as the Valve. If the component has "none" entered in the Design Flow Delta T field, it will be ignored for the sizing of the design flow rate by the valve. If a value is entered, TAS will size the design flow rate using the Design Flow Delta T field along with the peak load of the component. The sizing is done such that the fluid is kept within the delta T temp limit when passing through the component and so that the demand of the component / collection is met. Please note that for a collection it will just use the Design Flow Delta T field. If a Design Flow Delta T is entered into 2 or more components / collections on the same circuit as the pump, TAS will use the component / collection which results in the larger design flow rate to size the design flow rate.
    - Value – Upon choosing this option the user will enter a value which will be used for the Design Flow Rate.
- **Design Flow Capacity Signal –** The Design Flow Capacity Signal option only appears when a controller is attached to the component. The design flow capacity signal allows the user to make the design flow rate of the damper correspond to a certain signal from the controller. For instance if the Design Flow Capacity Signal field has the value x entered into it then when the controller's signal reads x, the flow rate through the damper will be the design flow rate. By design the factor is set to one by default and it is strongly recommended that it is kept that way.

4.5 Junction



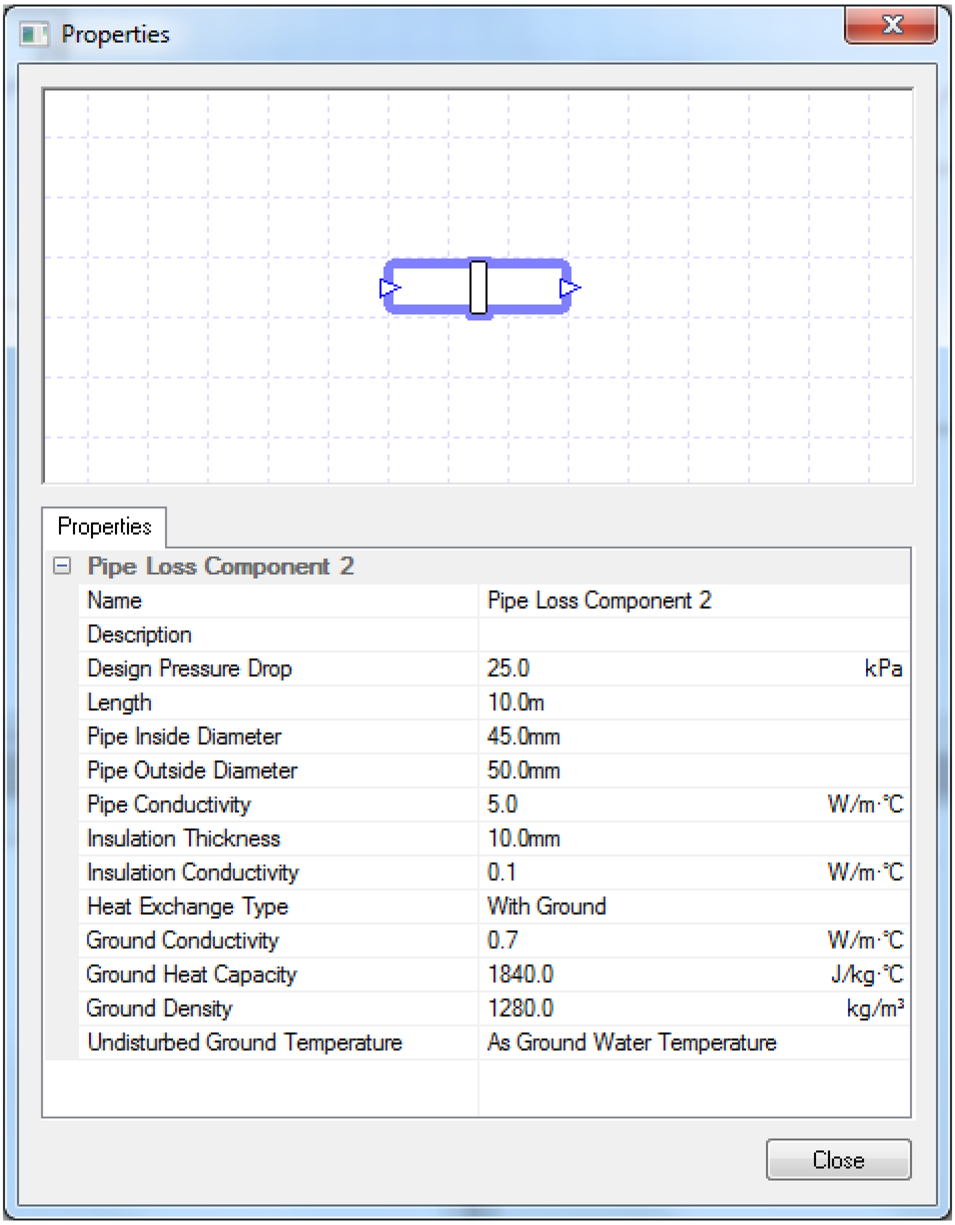
The Junction component has two uses. The first use is to model where the fluid enters the system and where the fluid leaves the system. In these cases the junction will only have one duct connected to it and it will turn blue, to indicate it is an external junction.
The second use of a junction is to split or merge air paths. When being used to split fluid paths, a valve should be used in conjunction with the junction. You cannot use controllers with junctions.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Mains Pressure –** This field allows the user to account for the fluid pressure their fluid is supplied at from the mains; due to this, this option only appears for external junctions. If the user uses the “Off” option then the fluid is not pressurised when provided from the mains. If the “On” option is chosen, the user needs to enter the pressure the fluid is supplied at from the mains. When a pressure is modelled here, the user will not need to model a pump on

the circuit loop, as the mains pressure will cause the fluid to flow. However, the user may want to model a valve on the circuit as a replacement to the pump, so they can turn off the flow when there is no demand.

4.6 Pipe Loss Component



The Pipe Loss Component allows the user to model the heat loss / gain that the fluid will experience when travelling through pipes. The user can model the pipes being placed in the ground or exposed to air. Please note that controllers cannot be used with this component.

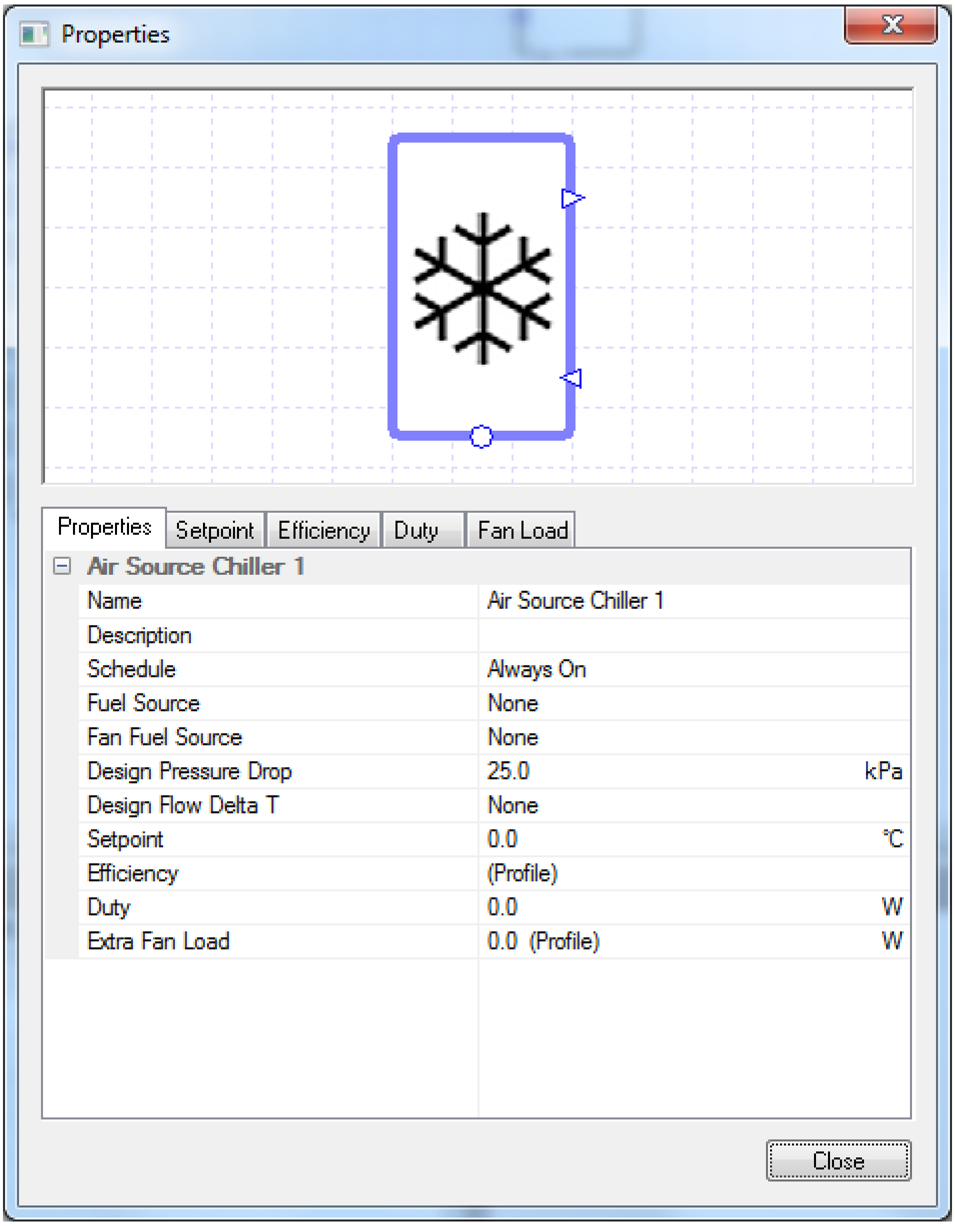
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid when it flows through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the component's circuit loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Length –** The user will input the length of the pipe in this field.
- **Pipe Inside Diameter –** The user will input the inside diameter of the pipe.

- **Pipe Outside Diameter –** The user should input the outside diameter of the pipe.
- **Pipe Conductivity –** Please enter the thermal conductivity of the pipe, not including the insulation.
- **Insulation thickness –** Please enter here the thickness of your insulation.
- **Insulation Conductivity –** Please enter here the thermal conductivity of the insulation.
- **Heat Exchange Type –** This field allows the user to tell TAS if the pipe is in the ground, by setting the heat exchanger type to "With Ground", or exposed to air, by setting the heat exchanger type to "With Air". Upon choosing the "With Ground" option, the user will have to enter the following information:
    - Ground Conductivity – This field requires the user to enter the thermal conductivity of the ground surrounding the pipe. If the soil type varies in the ground around the pipe then please take a weighted average for the ground conductivity.
    - Ground Heat Capacity - This field requires the user to enter the heat capacity of the ground surrounding the pipe. If the soil type varies in the ground around the pipe, then please take a weighted average for the specific heat capacity.
    - Ground Density - This field requires the user to enter the density of the ground that the pipe is placed into. If the soil type changes around the pipe then please take a weighted average for the density.
    - Undisturbed Ground Temperature – This will be the temperature of the ground before any heat exchange. This can be entered as a value or set by using the "Set to Ground Water Temp" option. If using the latter option, it will be set using the Water Temperature option in the simulation dialog box.

  If the user chooses the "With Air" option, they will be asked to enter the ambient air temperature of the air surrounding the pipe. The user can use the Ambient Air Temperature tab to set up a modifier or yearly profile for this temperature. They can also drag a zone, from the TSDData folder, onto the pipe and Systems will use the zone's temperature as the ambient temperature for each hour of the simulation.

4.7 Air Source Chiller



The Air Source Chiller component in TAS Systems allows users to model an air source chiller. The two ports on the chiller represent where the fluid ducts connect to the chiller. You can use a controller with the chiller to control how the chiller behaves. For a chiller, the controller controls it by informing it of the amount of power the chiller should use to cool down the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it. While if it received a signal of 1 the chiller will cool down the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after the return port of the chiller.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the air source chiller, this will mean that the fluid will flow through the chiller uncooled, even if there is a controller sending a non-zero signal to the chiller. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source –** With this field the user can choose the fuel source of the air source chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Fan Fuel Source –** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the Air Source Chiller's fan.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid flowing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the chillers circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the air source chiller, it will cool the fluid so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Efficiency –** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency. Normally the modifier chosen here would be a table modifier with a partload profile. The efficiency input for a chiller should be the EER.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For an air source chiller, it means it wouldn't be able to cool the fluid to the setpoint, it would fall

short.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:

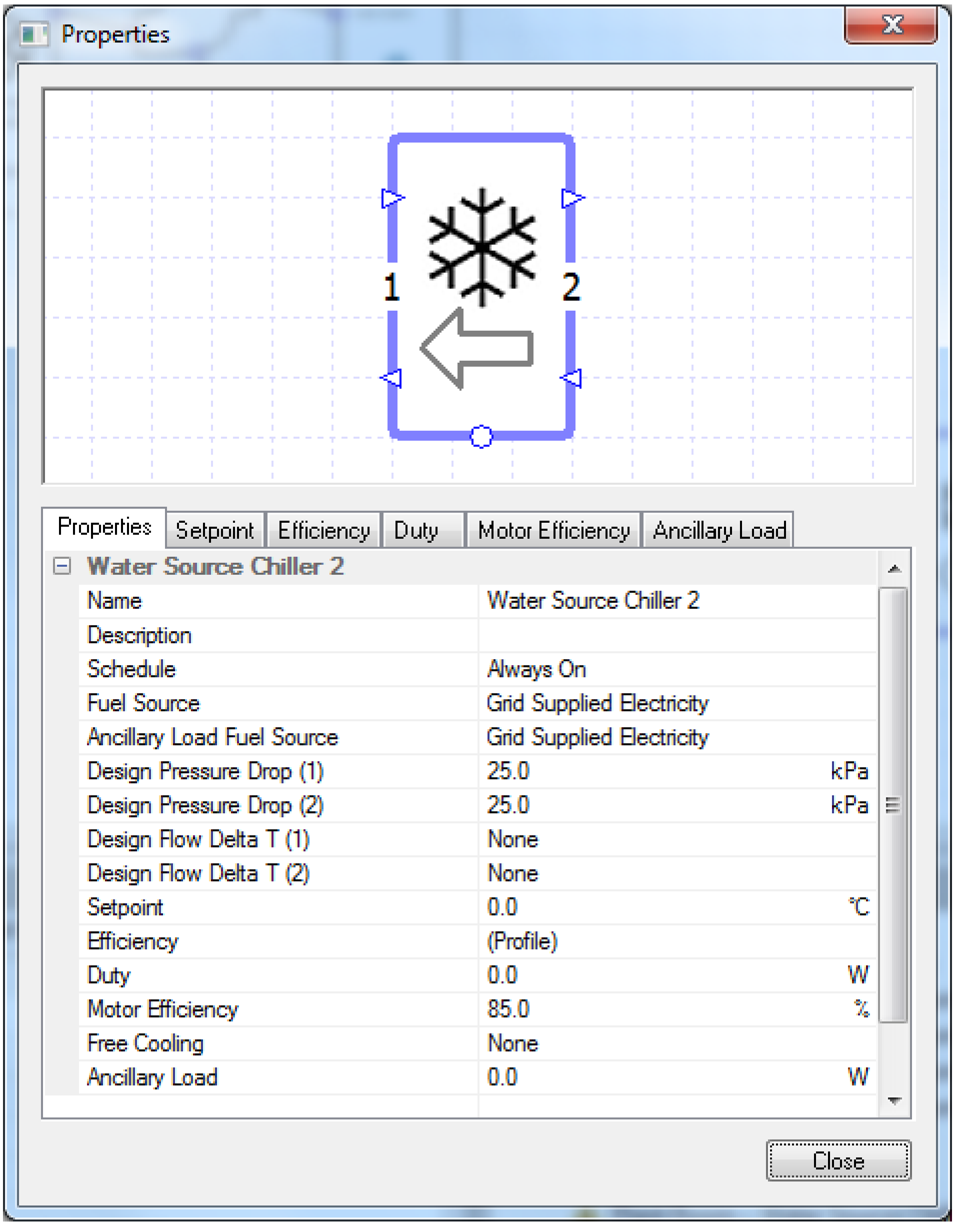
- Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
- Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
  - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
  - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

  Please note that to size the duty the user will need to have design conditions in their systems file.
- Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

- **Extra Fan Load –** This field allows the user to model the load used by the fan in the chiller. The user will be taken to the modifier profile in the Fan Load tab, where you can edit the profile to reflect the partload profile of the fan in the chiller.

4.8 Water Source Chiller



The Water Source Chiller component in TAS Systems allows users to model water source chillers which transfer their load to another fluid loop. Due to this the water source chiller requires connection to two circuit loops, one for providing the cooling to the collection and another for the rejected load. The component has an arrow on it to indicate to what loop the load is being rejected. Also when you click on the component, the numbers 1 and 2 appear next to the ports to denote the loops. The cooling collection loop should be connected to the ports on the side numbered 2 (the side of the arrow's tail) while the heat rejection loop should be connected to the ports on the side numbered 1 (the side of the arrow's point). You can use a controller with the chiller to control how the chiller behaves. For a chiller, the controller controls it by informing it of the amount of power the chiller should use to cool down the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it on the cooling loop. While if it received a signal of 1 the chiller will cool down the fluid flowing through it on the cooling loop using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor for determining the temperature of the fluid for the Setpoint field is assumed to be directly after the exit port on the cooling loop (side denoted with a 2).

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the water source chiller, this will mean that the fluid will flow through the cooling loop of the chiller uncooled and no heat would be rejected to the heat rejection loop, even if there is a controller sending a non-zero signal to the chiller. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source –** With this field the user can choose the fuel source of the water source chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder, so the user will be able to choose a non-electrical fuel source but this is not advised. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the chiller.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid on the heat rejection loop when flowing through the component (This is the side marked with a 1 when you click on the component). The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the heat rejection loop. If the “Not Used” option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) -** This field allows the user to enter the pressure drop of the fluid on the cooling loop when flowing through the component (This is the side marked with a 2 when you click on the component). The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the cooling loop. If the “Not Used” option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Flow Delta T (1) –** The Design Flow Delta T (1) field allows the user to size the flow rate of the heat rejection loop of the component such that the fluid flowing through the component is kept within a certain temperature band (Please note that this is the loop marked with a 1 when you click on the component). The user can choose not to size the flow rate using this band, by choosing the “None” option. If the user decides to use this option, by choosing the “Value” option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump’s Design Flow Rate field set to “Sized”. The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of x and a Design Flow Delta T of y, then the fluid will always be kept within the temperature range $x - y$ to $x + y$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.

- **Design Flow Delta T (2) -** The Design Flow Delta T (2) field allows the user to size the flow rate of the cooling loop of the component such that the fluid flowing through the component is kept within a certain temperature band (Please note that this is the loop marked with a 2 when you click on the component). Apart from this difference, it works in the same way as the Design Flow Delta T (1) field.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the Water Source Chiller, it will cool the fluid so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency -** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency. Normally the modifier chosen here would be a table modifier with a partload profile. The efficiency input for a chiller should be the EER.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a water source chiller, it means it wouldn’t be able to cool the fluid to the setpoint, it would fall short.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

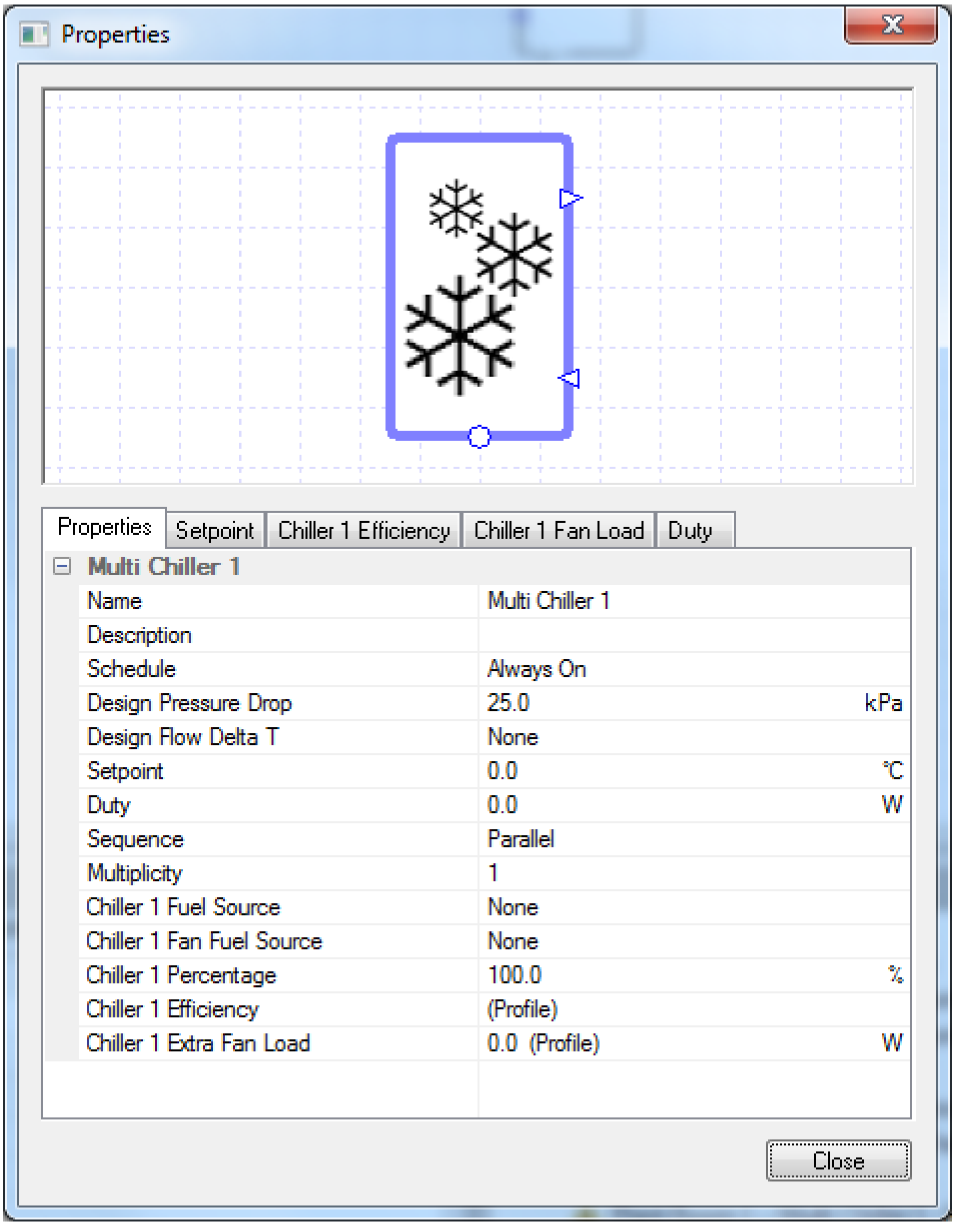
  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Motor Efficiency** – The Motor Efficiency field allows the user to set the chiller’s motor efficiency. This efficiency of the motor details how efficient the chiller is at transferring the heat created due to the consumption over to the heat rejection loop. The user enters in a percentage into this field and this percentage of the heat generated due to the consumption is transferred over to the heat rejection loop. If the efficiency is less than 100%, some of the heat created due to the consumption will be emitted outside of the chiller rather than being transferred to the heat rejection loop.
- **Free Cooling –** With this field, the user can decide if they want the Water Source Chiller component to model a heat exchanger exchanging heat between the heat rejection and cooling loops situated before the water source chiller (akin to a water side economiser setup). There are three options to choose from:
    - None – Selecting this option means that no heat exchanger will be modelled.

  - On/Off – The heat exchanger will only exchange heat between the heat rejection and cooling loops when the entire cooling demand can be met by the heat exchange. Otherwise, no heat will be exchanged and the chiller will solely meet the cooling demand.
  - Variable - The heat exchanger will exchange heat between the heat rejection and cooling loops when part of the cooling demand can be met by the heat exchange. The remaining demand will be met by the chiller.
- **Calculation Method –** This option allows the user to set up how the heat exchanger efficiency will be calculated when the Chiller is in free cooling mode. The user has the following two options:
  - Efficiency – The first calculation method is efficiency. Upon choosing this option the user is asked to enter the efficiency of the heat transfer. The efficiency of the heat transfer is calculated by the following formula:

$$\boldsymbol{Efficiency} = \frac{\boldsymbol{Cooling_{Out}} - \boldsymbol{Cooling_{in}}}{\boldsymbol{Heat_{in}} - \boldsymbol{Cooling_{in}}},$$

    where: $\boldsymbol{Cooling_{out}}$ is the temperature of the fluid leaving the chiller on the Cooling Collection loop, $\boldsymbol{Cooling_{in}}$ is the temperature of the fluid entering the chiller on the Cooling Collection loop and $\boldsymbol{Heat_{in}}$ is the temperature of the fluid entering the chiller on the Heat Rejection loop. Please note that in the efficiency Tab, you can add a modifier to the Efficiency.
  - NTU Method – The second calculation method is the NTU method. Upon choosing this method, the user will need to input: The heat transfer surface area, the heat transfer coefficient and the exchanger type. After entering these details, TAS will then work out the rate of heat transfer of the Exchanger.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the chiller. Any value entered here will be the associated load during any hour the chiller operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.9 Multi Chiller



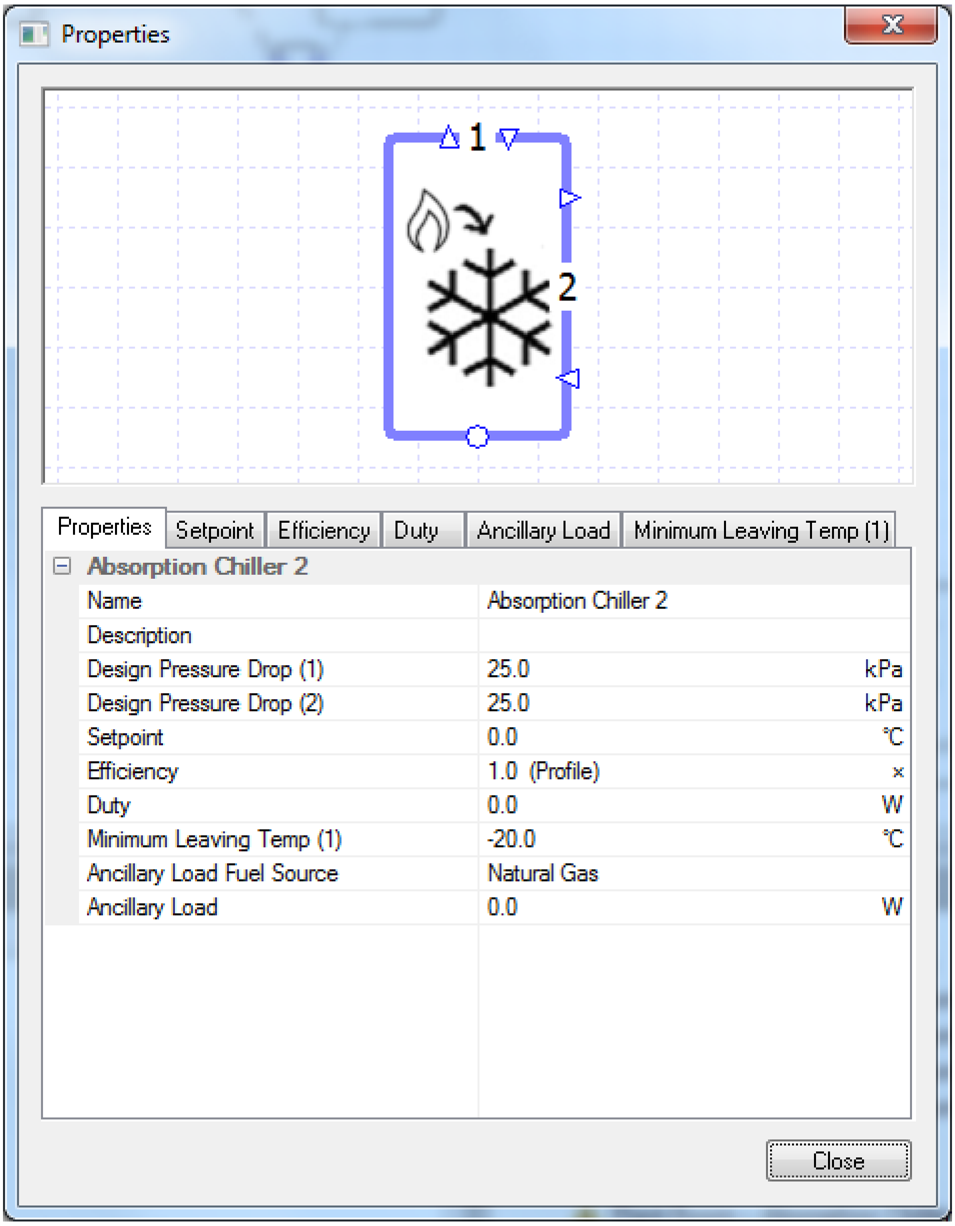
The Multi Chiller component allows the user to model multiple air source chillers using only one component. Please note that any field beginning with "Chiller x …", where x is a positive integer, is unique to the x^{th} air source chiller. The other fields (apart from: duty, sequence and multiplicity) are shared by all the chillers. So if the Multi Chiller is modelling 3 air source chillers, then each chiller will have the same setpoint while their fuel sources will be dependent on the field "Chiller x fuel source ". As air source chillers have been discussed previously, only the new properties will be discussed.

New Properties:

- **Multiplicity –** The number in the Multiplicity field represents how many chillers the Multi Chiller is representing.
- **Chiller x Percentage –** This option allows the user to set the duty of the x^{th} chiller. The user will enter a percentage in this field which will set the chillers duty as this percentage of the total duty in the duty field. Please note that the sum of all the "Chiller x Percentage" fields must add up to 100%. Not doing so will give you a warning and incorrect results.
- **Sequence –** The Sequence field tells the component how the multiple chillers will be set up. The user has three options to choose from:

- Parallel – The load is split up between the chillers. The load will be split evenly until one of the chillers reaches its duty.
- Serial – The first chiller covers the load until it reaches its duty. At that point the second chiller would start to cover the load and so on.
- Staged – With a staged setup only the first chiller runs at part load. As the first chiller normally has a bigger duty than the other chillers, once one of the other chillers can run at full load the load will transition to that chiller.

4.10 Absorption Chiller



The absorption chiller is an air source chiller that uses a heat source to provide the energy to drive the cooling system. Please note that as the heat source is providing the energy to the chiller, the demand on the heat source will match the consumption of the chiller. If the heat source cannot meet this demand then the absorption chiller will not have enough energy to meet the cooling demand. The Absorption Chiller component has two loops, which are denoted 1 and 2 when the component is selected. The loop denoted 1, called Loop 1, should be the loop containing the heat source. The loop denoted 2, called Loop 2, should be the loop containing the Cooling collection. You can use a controller with the chiller to control how the chiller behaves. For a absorption chiller, the controller controls it by informing it of the amount of power the absorption chiller should use to cool down the fluid flowing through it on Loop 2 (the cooling loop). The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it. While if it received a signal of 1 the chiller will cool down the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after Loop 2's return port.

Properties:

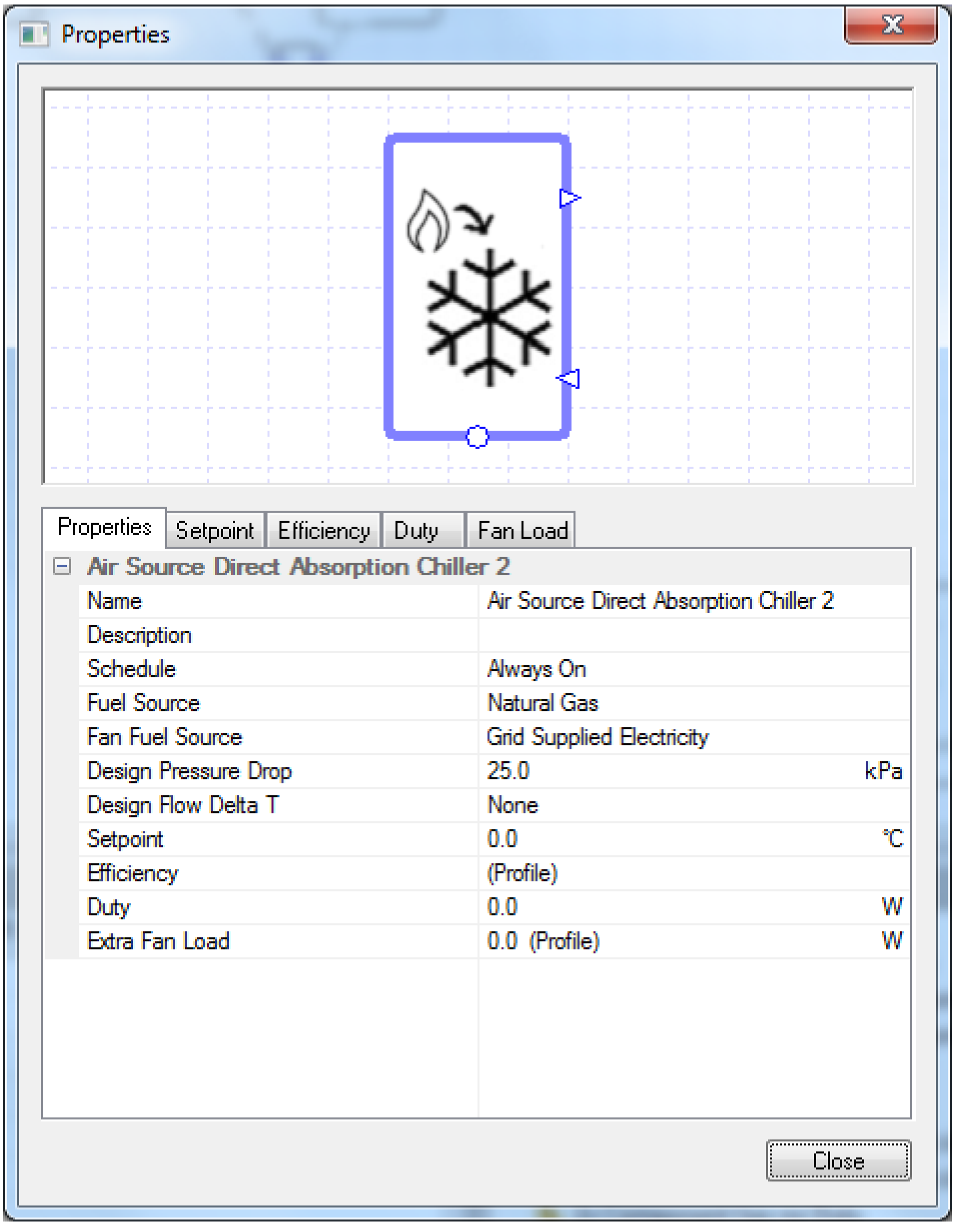
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid on Loop 1 (The loop containing the heat source providing the energy for the absorption chiller) when passing through the absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the left loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) -** This field allows the user to enter the pressure drop of the fluid on Loop 2 (The loop containing the Cooling Collection on the side of the absorption chiller) when passing through the absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the right loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Setpoint –** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the absorption chiller, it will cool the fluid on Loop 2 (the cooling loop) so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency -** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency. Normally the modifier chosen here would be a table modifier with a partload profile. The efficiency input for a chiller should be the EER.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For an absorption chiller, it means it wouldn't be able to cool the fluid to the setpoint, it would fall short.). In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

- **Minimum Leaving Temperature (1)** – This field sets a minimum leaving temperature for the fluid flowing through the absorption chiller on Loop 1 (i.e. the loop containing the heat source). Please note that setting this minimum return temperature too high can lead to the absorption chiller not receiving enough heat to meet the cooling demand. Modifiers can be added to this field by selecting the Minimum Leaving Temperature (1) tab.
- **Ancillary Load Fuel Source -** With this field the user can choose the fuel source of the ancillary loads associated with the absorption chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and any entered ancillary loads of the component will be discarded.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the chiller. Any value entered here will be the associated load during any hour the chiller operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.11 Air Source Direct Absorption Chiller



The Air Source Direct Absorption Chiller component acts like an absorption chiller, but does not require a separate loop containing the absorption chiller's heat source to be modelled. Instead, the Efficiency field of the chiller would account for the absorption chiller creating the heat and then using it to power the cooling. The two ports on the chiller represent where the fluid ducts connect to the chiller. You can use a controller with the chiller to control how the chiller behaves. For a chiller, the controller controls it by informing it of the amount of power the chiller should use to cool down the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it. While if it received a signal of 1 the chiller will cool down the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after the return port of the chiller.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the air source direct absorption chiller, this will mean that the fluid will flow through the chiller uncooled, even if there is a controller sending a non-zero signal to the chiller. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source –** With this field the user can choose the fuel source of the air source direct absorption chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and the loads of the component will be discarded.
- **Fan Fuel Source –** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the air source direct absorption chiller’s fan.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid flowing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the chillers circuit. If the “Not Used” option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the air source direct absorption chiller, it will cool the fluid so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the “None” option. If the user decides to use this option, by choosing the “Value” option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump’s Design Flow Rate field set to “Sized”. The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x-y}$ to $\boldsymbol{x+y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Efficiency –** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency. Normally the modifier chosen here would be a table modifier with a partload profile. Please note that for the air source

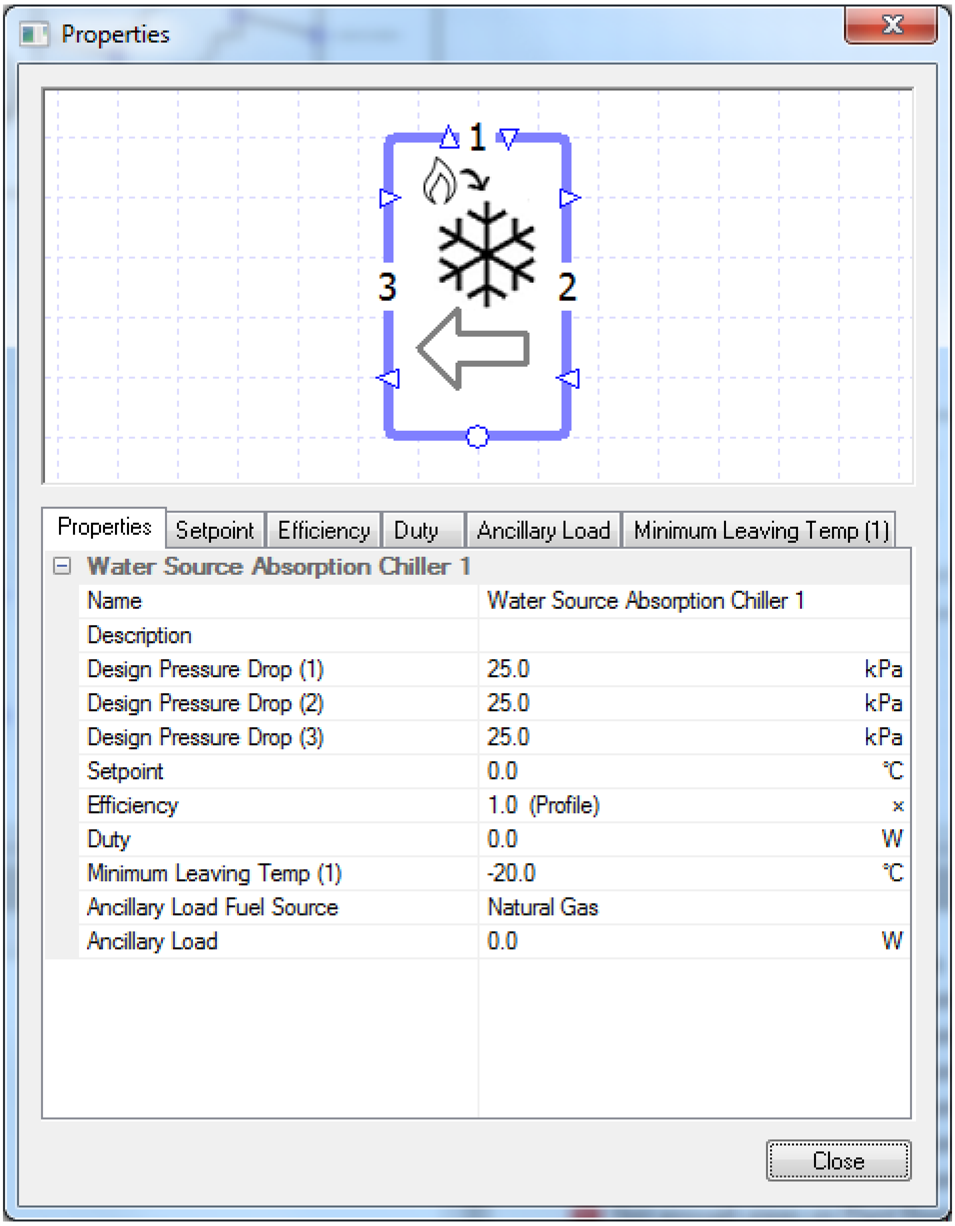
direct absorption chiller, the Efficiency of the chiller would account for the absorption chiller creating the heat and then using it to power the cooling.

- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For an air source direct absorption chiller, it means it wouldn't be able to cool the fluid to the setpoint, it would fall short.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
  - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
  - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
    - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
    - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

    Please note that to size the duty the user will need to have design conditions in their systems file.
  - Value – With this option the user will type in the duty of the component.

    In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Extra Fan Load –** This field allows the user to model the load used by the fan in the chiller. The user will be taken to the modifier profile in the Fan Load tab, where you can edit the profile to reflect the partload profile of the fan in the chiller.

4.12 Water – Source Absorption Chiller



The water source absorption chiller acts in much the same way as the absorption chiller but instead of rejecting the heat to the air, it rejects it to a heat rejection fluid loop. Due to the additional heat rejection loop, the Water Source Absorption Chiller component has three loops, which are denoted 1, 2 and 3 when the component is selected. The loop denoted 1, called Loop 1, should be the loop containing the heat source. The loop denoted 2, called Loop 2, should be the loop containing the Cooling collection, while the loop denoted 3, called Loop 3, should be the heat rejection loop. You can use a controller with the chiller to control how the chiller behaves. For a water source absorption chiller, the controller controls it by informing it of the amount of power the absorption chiller should use to cool down the fluid flowing through it on Loop 2 (the cooling loop). The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it. While if it received a signal of 1 the chiller will cool down the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after Loop 2's return port.

Properties:

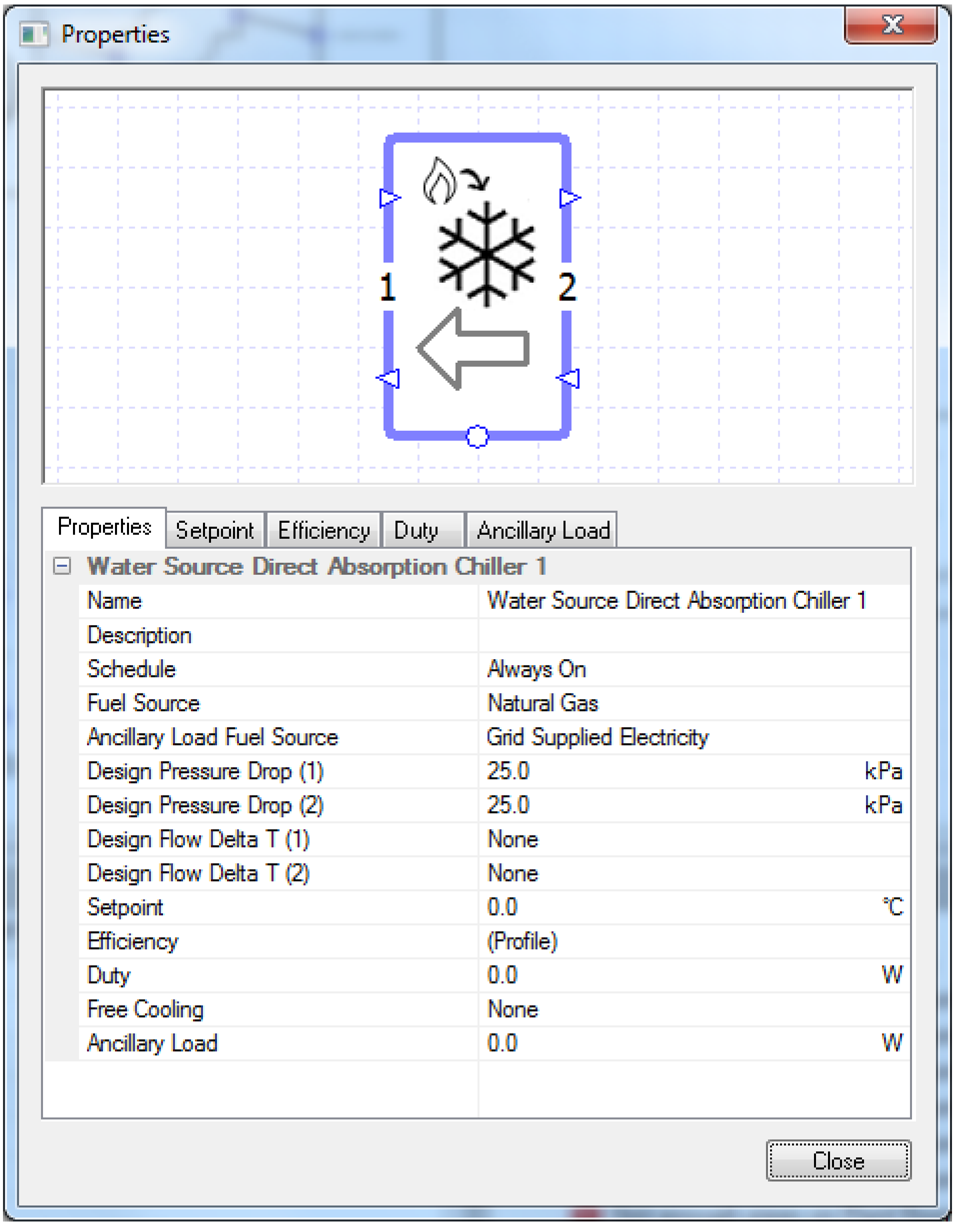
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid on Loop 1 (the loop containing the heat source providing the energy for the water source absorption chiller) when passing through the water source absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the left loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) -** This field allows the user to enter the pressure drop of the fluid on Loop 2 (the loop containing the Cooling Collection on the side of the water source absorption chiller) when passing through the water source absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the right loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (3) -** This field allows the user to enter the pressure drop of the fluid on Loop 3 (the heat rejection loop of the water source absorption chiller) when passing through the water source absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the right loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Setpoint –** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the water source absorption chiller, it will cool the fluid on Loop 2 (the cooling loop) so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency -** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency. Normally the modifier chosen here would be a table modifier with a partload profile. The efficiency input for a chiller should be the EER.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a water source absorption chiller, it means it wouldn't be able to cool the fluid to the setpoint, it would fall short.). In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:

        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Minimum Leaving Temperature (1)** – This field sets a minimum leaving temperature for the fluid flowing through the water source absorption chiller on Loop 1 (i.e. the loop containing the heat source). Please note that setting this minimum return temperature too high can lead to the absorption chiller not receiving enough heat to meet the cooling demand. Modifiers can be added to this field by selecting the Minimum Leaving Temperature (1) tab.
- **Ancillary Load Fuel Source -** With this field the user can choose the fuel source of the ancillary loads associated with the water source absorption chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and any entered ancillary loads of the component will be discarded.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the chiller. Any value entered here will be the associated load during any hour the chiller operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.13 Water Source Direct Absorption Chiller



The water source direct absorption chiller acts in much the same way as the air source direct absorption chiller but instead of rejecting the heat to the air; it rejects it to a heat rejection fluid loop. Due to the additional heat rejection loop, the Water Source Direct Absorption Chiller component has two loops, which are denoted 1 and 2 when the component is selected. The loop denoted 1, called Loop 1, is the heat rejection loop. The loop denoted 2, called Loop 2, should be the loop containing the Cooling collection. You can use a controller with the chiller to control how the chiller behaves. For a chiller, the controller controls it by informing it of the amount of power the chiller should use to cool down the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it on the cooling loop. While if it received a signal of 1 the chiller will cool down the fluid flowing through it on the cooling loop using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor for determining the temperature of the fluid for the Setpoint field is assumed to be directly after the exit port on the cooling loop (side denoted with a 2).

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the water source direct absorption chiller, this will mean that the fluid will flow through the cooling loop of the chiller uncooled and no heat would be rejected to the heat rejection loop, even if there is a controller sending a non-zero signal to the chiller. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source –** With this field the user can choose the fuel source of the water source direct absorption chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the chiller.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid on the heat rejection loop when flowing through the component (This is the side marked with a 1 when you click on the component). The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the heat rejection loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) -** This field allows the user to enter the pressure drop of the fluid on the cooling loop when flowing through the component (This is the side marked with a 2 when you click on the component). The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the cooling loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Flow Delta T (1) –** The Design Flow Delta T (1) field allows the user to size the flow rate of the heat rejection loop of the component such that the fluid flowing through the component is kept within a certain temperature band (Please note that this is the loop marked with a 1 when you click on the component). The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of x and a Design Flow Delta T of y, then the fluid will always be kept within the temperature range $x - y$ to $x + y$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Design Flow Delta T (2) -** The Design Flow Delta T (2) field allows the user to size the flow rate of the cooling loop of the component such that the fluid flowing through the component is kept within a certain temperature band (Please note that this is the loop

marked with a 2 when you click on the component). Apart from this difference, it works in the same way as the Design Flow Delta T (1) field.

- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the water source direct absorption chiller, it will cool the fluid so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency -** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency. Normally the modifier chosen here would be a table modifier with a partload profile. Please note that for the water source direct absorption chiller, the Efficiency of the chiller would account for the absorption chiller creating the heat and then using it to power the cooling.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a water source direct absorption chiller, it means it wouldn't be able to cool the fluid to the setpoint, it would fall short.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

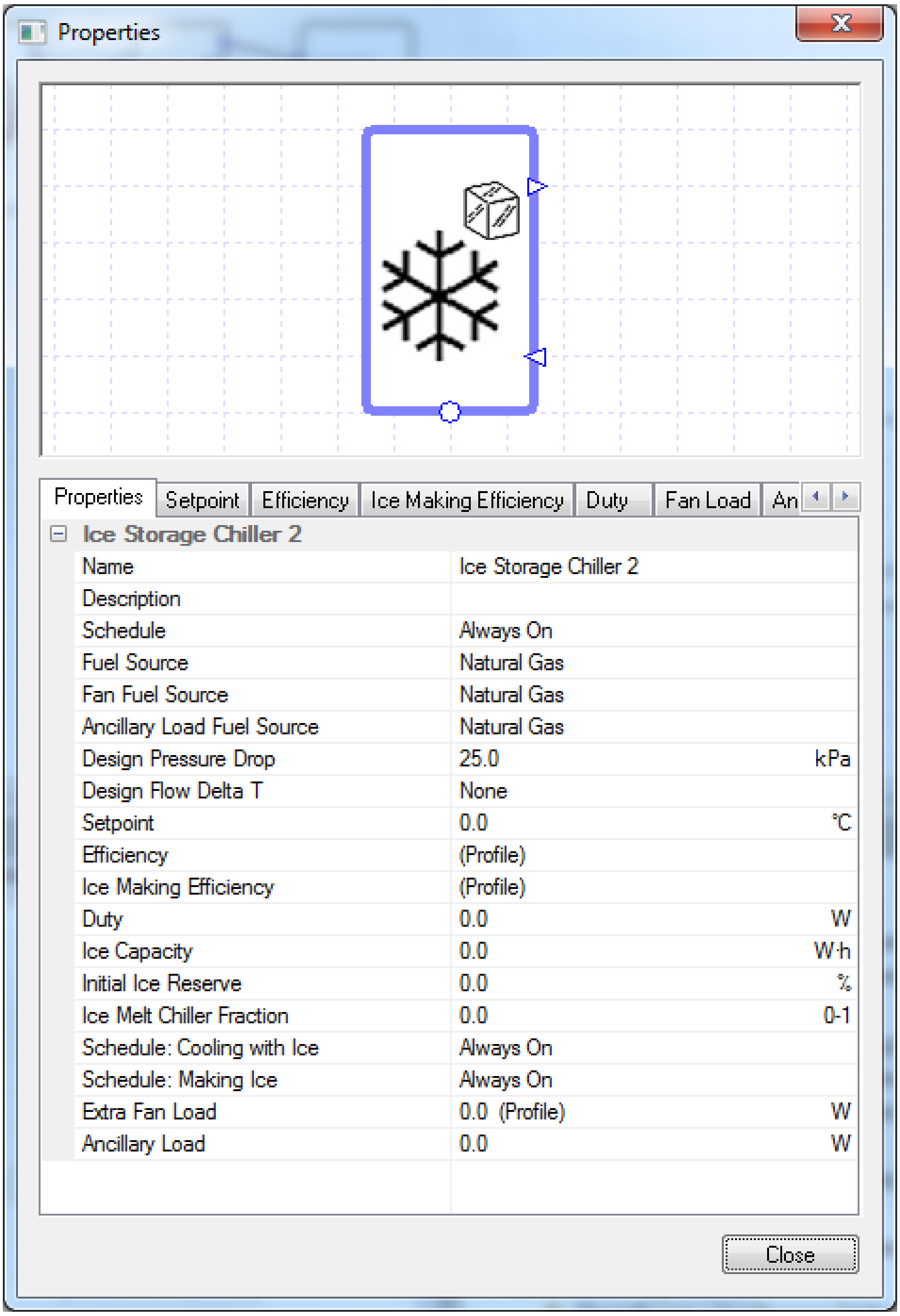
  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Free Cooling –** With this field, the user can decide if they want the Water Source Direct Absorption Chiller component to model a heat exchanger exchanging heat between the heat rejection and cooling loops situated before the water source direct absorption chiller (akin to a water side economiser setup). There are three options to choose from:
    - None – Selecting this option means that no heat exchanger will be modelled.
    - On/Off – The heat exchanger will only exchange heat between the heat rejection and cooling loops when the entire cooling demand can be met by the heat exchange. Otherwise, no heat will be exchanged and the chiller will solely meet the cooling demand.
    - Variable - The heat exchanger will exchange heat between the heat rejection and cooling loops when part of the cooling demand can be met by the heat exchange. The remaining demand will be met by the chiller.

- **Calculation Method –** This option allows the user to set up how the heat exchanger efficiency will be calculated when the Chiller is in free cooling mode. The user has the following two options:
  - Efficiency – The first calculation method is efficiency. Upon choosing this option the user is asked to enter the efficiency of the heat transfer. The efficiency of the heat transfer is calculated by the following formula:

    $$\boldsymbol{Efficiency} = \frac{\boldsymbol{Cooling_{Out}} - \boldsymbol{Cooling_{in}}}{\boldsymbol{Heat_{in}} - \boldsymbol{Cooling_{in}}},$$

    where: $\boldsymbol{Cooling_{out}}$ is the temperature of the fluid leaving the chiller on the Cooling Collection loop, $\boldsymbol{Cooling_{in}}$ is the temperature of the fluid entering the chiller on the Cooling Collection loop and $\boldsymbol{Heat_{in}}$ is the temperature of the fluid entering the chiller on the Heat Rejection loop. Please note that in the efficiency Tab, you can add a modifier to the Efficiency.
  - NTU Method – The second calculation method is the NTU method. Upon choosing this method, the user will need to input: The heat transfer surface area, the heat transfer coefficient and the exchanger type. After entering these details, TAS will then work out the rate of heat transfer of the Exchanger.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the chiller. Any value entered here will be the associated load during any hour the chiller operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.14 Ice Storage Chiller



The ice storage chiller is an air source chiller but with the added ability to store excess cooling capacity as ice. You can use a controller with the chiller to control how the chiller behaves. For a chiller, the controller controls it by informing it of the amount of power the chiller should use to cool down the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it. While if it received a signal of 1 the chiller will cool down the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after the return port of the chiller.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the ice storage chiller, this will mean that the fluid will flow through the chiller uncooled, even if there is ice in the chiller. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source -** With this field the user can choose the fuel source of the ice storage chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Fan Fuel Source –** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ice storage chiller's fan.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the chiller.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the chillers circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of x and a Design Flow Delta T of y, then the fluid will always be kept within the temperature range $x - y$ to $x + y$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the ice storage chiller, it will cool the fluid so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency –** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency of the chiller. Normally the

modifier chosen here would be a table modifier with a partload profile. The efficiency input for a chiller should be the EER.

- **Ice Making Efficiency –** Like the Efficiency field, the Ice Making Efficiency field uses a modifier to create a Partload profile. The efficiency entered here is the efficiency of the ice storage chiller when making ice.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For an ice storage chiller, it means that the cooling by duty will not be able to cover the demand. If any ice is being stored, the ice will be used to attempt to meet the demand.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
  - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
  - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
    - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
    - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

    Please note that to size the duty the user will need to have design conditions in their systems file.
  - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Ice Capacity –** The Ice Capacity field allows the user to input the maximum amount of ice the ice storage chiller can store. The user can determine how to set the capacity of the chiller from the following two options:
  - Sized – Allows the user to size the ice storage chiller during the simulation. An Hours at Full Load field will appear, allowing the user to enter how many hours of the chiller running at full load would it take to fully replenish the ice.
  - Value – With this option the user would type in the ice capacity into the software.
- **Initial Ice Reserve –** The Initial Ice Reserve field allows the user to set the amount of stored ice the ice storage chiller has at the start of the simulation. The user should enter the amount of ice stored as a percentage of the ice capacity.
- **Ice Melt Chiller Fraction –** This field allows the user to enter in a minimum demand always met by the chiller's duty, even when the chiller is set to prioritise cooling with ice. This minimum demand is entered as a fraction of the duty, where zero would denote there is no minimum demand always met by the duty of the chiller.
- **Schedule: Cooling With Ice -** This field allows the user to set a schedule dictating the hours where the chiller will prioritise cooling with Ice. Taking into account the Ice Melt Chiller Fraction, the priority for cooling in these hours will be:

  $$\textit{cooling with duty up to Ice Melt Chiller Fraction} \rightarrow \textit{cooling with ice} \rightarrow \textit{cooling with remaining duty}$$

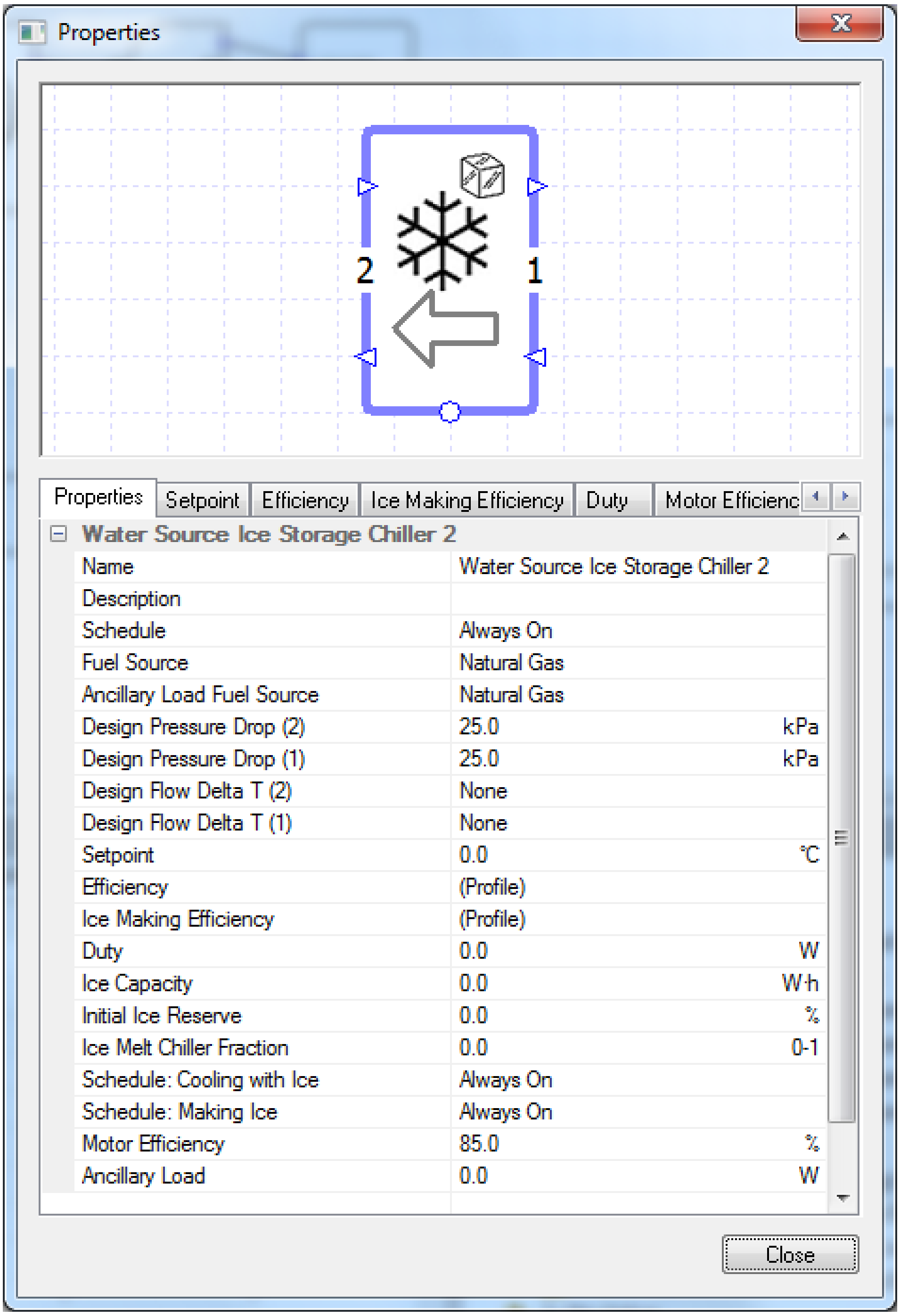
  Outside of these hours, or in hours where the Cooling with Ice schedule clashes with the Making Ice schedule, the priority will be:

  $$\textit{cooling with duty} \rightarrow \textit{cooling with ice} \rightarrow \textit{Making Ice.}$$
- **Schedule: Making Ice –** This field allows the user to set a schedule on when the chiller will make ice. Please be aware that outside of scheduled hours, the chiller will not cool using this

functionality. Also if all of the chillers duty is used up meeting the Cooling demand, then the chiller will not be able to create ice.

- **Extra Fan Load -** This field allows the user to model the load used by the fan in the chiller. The user will be taken to the modifier profile in the Fan Load tab, where you can edit the profile to reflect the partload profile of the fan in the chiller.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the chiller. Any value entered here will be the associated load during any hour the chiller operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.15 Water Source Ice Storage Chiller



The water source ice storage chiller works in nearly the same way as the ice storage chiller does apart from the fact that the water source ice storage chiller rejects its load to a heat rejection fluid loop, rather than to the air. This means the Water Source Ice Storage Chiller component has two loops, denoted 1 and 2 when the component is selected. The loop denoted 1, called Loop 1, is the loop the chiller will cool (i.e. the cooling loop) ; while the loop denoted 2, called Loop 2, is the loop the chiller will reject heat to (i.e. the heat rejection loop). You can use a controller with the chiller to control how the chiller behaves. For a chiller, the controller controls it by informing it of the amount of power the chiller should use to cool down the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the chiller dictating the proportion of the chiller's duty it should use to cool down the fluid. So, for example, if the chiller received a signal of zero the chiller would not cool down the fluid flowing through it. While if it received a signal of 1 the chiller will cool down the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after Loop 1's return port.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the water source ice storage chiller, this will mean that the fluid will flow through the chiller uncooled, even if there is ice in the chiller. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source -** With this field the user can choose the fuel source of the water source ice storage chiller. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the chiller.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid on Loop 1 (i.e. the cooling loop) when passing through the absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the left loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Pressure Drop (2) -** This field allows the user to enter the pressure drop of the fluid on Loop 2 (i.e. the heat rejection loop) when passing through the absorption chiller. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the right loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity of the fluid loop through this component.
- **Design Flow Delta T (1) –** The Design Flow Delta T (1) field allows the user to size the flow rate of the cooling loop of the component such that the fluid flowing through the component is kept within a certain temperature band (Please note that this is the loop marked with a 1 when you click on the component). The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of x and a Design Flow Delta T of y, then the fluid will always be kept within the temperature range $x - y$ to $x + y$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Design Flow Delta T (2) -** The Design Flow Delta T (2) field allows the user to size the flow rate of the heat rejection loop of the component such that the fluid flowing through the

component is kept within a certain temperature band (Please note that this is the loop marked with a 2 when you click on the component). Apart from this difference, it works in the same way as the Design Flow Delta T (1) field.

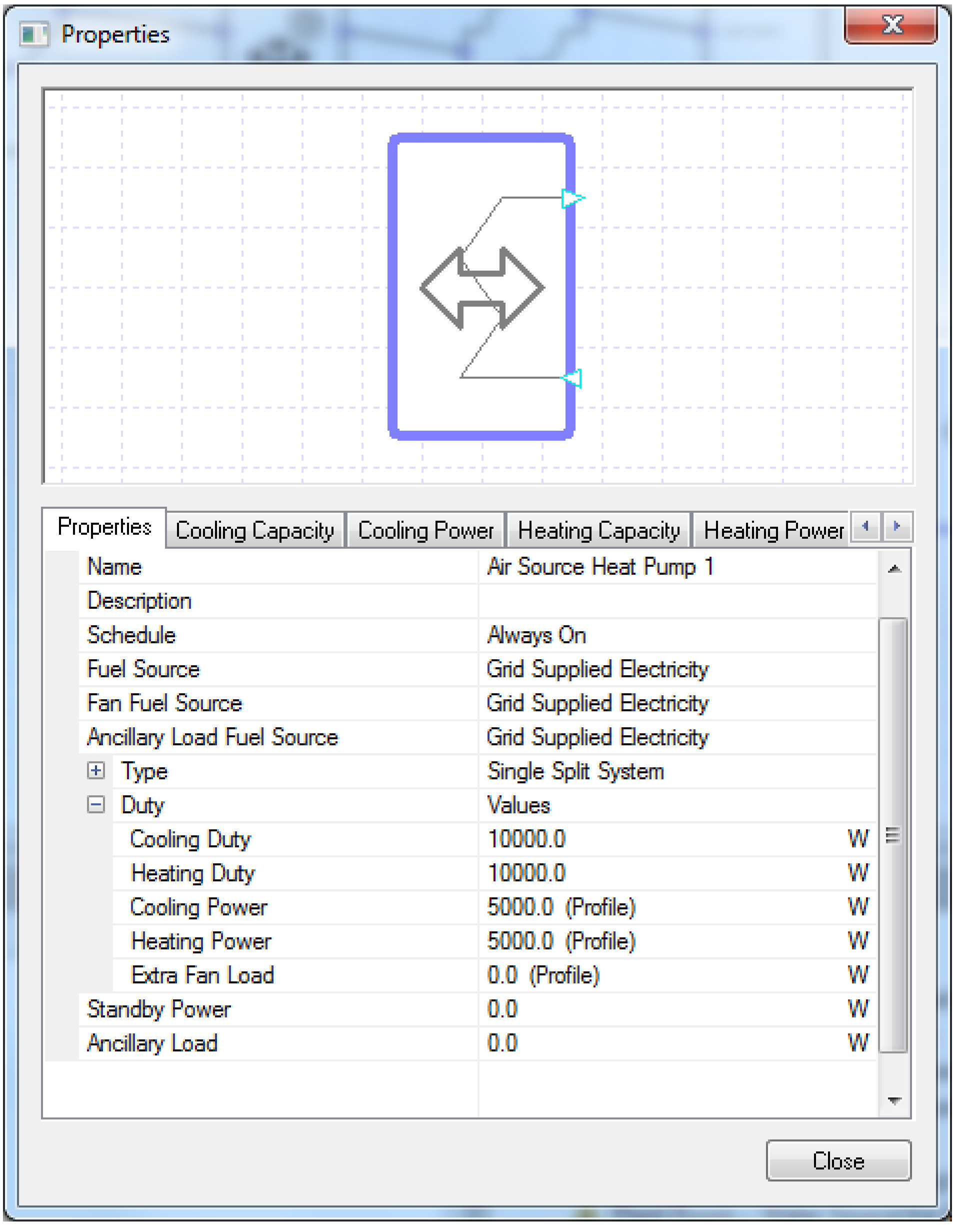
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the water source ice storage chiller, it will cool the fluid so it reaches the setpoint, but it will not be able to warm up the fluid to reach this setpoint. To heat the fluid you would need another component, for instance a boiler. Please note that when a controller is used in conjunction with the chiller, the Setpoint field will disappear from the properties. This is done because the chiller is being controlled by a controller and will cool down the fluid when the controller sends a signal informing the chiller to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency –** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency of the chiller. Normally the modifier chosen here would be a table modifier with a partload profile. The efficiency input for a chiller should be the EER.
- **Ice Making Efficiency –** Like the Efficiency field, the Ice Making Efficiency field uses a modifier to create a Partload profile. The efficiency entered here is the efficiency of the water source ice storage chiller when making ice.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a water source ice storage chiller, it means that the cooling by duty will not be able to cover the demand. If any ice is being stored, the ice will be used to attempt to meet the demand.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Ice Capacity –** The Ice Capacity field allows the user to input the maximum amount of ice the water source ice storage chiller can store. The user can determine how to set the capacity of the chiller from the following two options:
    - Sized – Allows the user to size the ice storage chiller during the simulation. An Hours at Full Load field will appear, allowing the user to enter how many hours of the chiller running at full load would it take to fully replenish the ice.
    - Value – With this option the user would type in the ice capacity into the software.
- **Initial Ice Reserve –** The Initial Ice Reserve field allows the user to set the amount of stored ice the water source ice storage chiller has at the start of the simulation. The user should enter the amount of ice stored as a percentage of the ice capacity.

- **Ice Melt Chiller Fraction –** This field allows the user to enter in a minimum demand always met by the chiller's duty, even when the chiller is set to prioritise cooling with ice. This minimum demand is entered as a fraction of the duty, where zero would denote there is no minimum demand always met by the duty of the chiller.
- **Schedule: Cooling With Ice -** This field allows the user to set a schedule dictating the hours where the chiller will prioritise cooling with Ice. Taking into account the Ice Melt Chiller Fraction, the priority for cooling in these hours will be:
$$cooling\ with\ duty\ up\ to\ Ice\ Melt\ Chiller\ Fraction\ \rightarrow\ cooling\ with\ ice\ \rightarrow\ cooling\ with\ remaining\ duty$$
Outside of these hours, or in hours where the Cooling with Ice schedule clashes with the Making Ice schedule, the priority will be:
$$cooling\ with\ duty\ \rightarrow\ cooling\ with\ ice\ \rightarrow\ Making\ Ice.$$
- **Schedule: Making Ice –** This field allows the user to set a schedule on when the chiller will make ice. Please be aware that outside of scheduled hours, the chiller will not cool using this functionality. Also if all of the chillers duty is used up meeting the Cooling demand, then the chiller will not be able to create ice.
- **Motor Efficiency** – The Motor Efficiency field allows the user to set the chiller's motor efficiency. This efficiency of the motor details how efficient the chiller is at transferring the heat created due to the consumption over to the heat rejection loop. The user enters in a percentage into this field and this percentage of the heat generated due to the consumption is transferred over to the heat rejection loop. If the efficiency is less than 100%, some of the heat created due to the consumption will be emitted outside of the chiller rather than being transferred to the heat rejection loop.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the chiller. Any value entered here will be the associated load during any hour the chiller operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.16 Air Source Heat Pump



The Air Source Heat Pump component in TAS Systems allows the user to model heat pumps which transfer the load with the air. The two ports on the side of the heat pump, which are light blue, indicate that the air source heat pump can only be connected to refrigerant circuits. Please note that controllers cannot be used with this component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the air source heat pump, no heating or cooling demand from a refrigerant group would be met. The default schedule option is always on, meaning that the component will operate 24/7.

- **Fuel Source -** With this field the user can choose the fuel source of the air source heat pump. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and the loads of the component will be discarded.
- **Fan Fuel Source –** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the air source heat pump‘s fan.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the air source heat pump.
- **Type –** This field allows the user to set how the air source heat pump works. There are 5 options to choose from, but only 3 should be used by the user:
    - Single Split – TAS models one air source heat pump for each component in the Refrigerant group. The demand and consumption figures are then worked out for each heat pump separately before being summed and reported by the Air Source Heat Pump component. Please note that each individual heat pump has identical parameters and the parameters are the ones entered into the component. So, for instance, if you enter the cooling duty as 1000W into the heat pump component then each individual heat pump’s cooling duty will be 1000W.
    - Multi Split – TAS models one air source heat pump for all components in the Refrigerant group. With a multi – split heat pump, heating and cooling cannot be provided at the same time. This means that for a hour where there is both a heating and cooling demand, the heat pump will cater for the larger demand while not being able to meet the smaller one (TAS will inform you, with a warning, of the hours this occurs).
    - VRF - Like the Multi-Split option, the VRF option models one air source heat pump dealing with the load from all the components in the Refrigerant group. The difference between the two is that the VRF option allows for both heating and cooling to be provided in a certain hour. The VRF option keeps the refrigerant flow for heating and cooling separate. The heat pump will then meet the larger of the heating / cooling demands and heat/cool the associated refrigerant to meet it. To meet the smaller of the two demands, the heat pump will use the rejected air from the heat pump to heat/cool the refrigerant on the other refrigerant path. To set up the VRF option correctly, TAS will ask the user to fill in the following fields:
        - Exchange Demand Factor (Heat) – When the heat pump has both a heating and cooling demand to meet with the heating demand being the bigger demand, the heat pump can attempt to meet the cooling demand by using the rejected cool air created while meeting the heating demand. The Exchange Demand Factor (Heat) represents the extra work required to do this. The following equation states how the Exchange Demand Factor (heat) is used to work out the total demand on the heat pump, which is then used to calculate the consumption of the heat pump:

$$\boldsymbol{Total\ Demand\ = Ref\ Demand + (EDF * Cooling\ Demand),}$$

          Where: $\boldsymbol{Total\ Demand}$ is the total demand on the heat pump, $\boldsymbol{Ref\ Demand}$ is the absolute value of the refrigerant demand reported by the refrigerant collection (The refrigerant demand reported here is just the $\boldsymbol{Heating\ demand - Cooling\ Demand}$), $\boldsymbol{EDF}$ is the Exchange Demand Factor and $\boldsymbol{Cooling\ Demand}$ is the cooling demand reported at the refrigerant collection.
        - Exchange Demand Factor (Cool) – When the heat pump has both a heating and cooling demand to meet with the Cooling demand being the bigger demand, the heat pump can attempt to meet the heating demand by using

the rejected hot air created while meeting the cooling demand. The Exchange Demand Factor (Cool) represents the extra work required to do this. The following equation states how the Exchange Demand Factor (cool) is used to work out the total demand on the heat pump, which is then used to calculate the consumption of the heat pump:

$$\boldsymbol{Total\ Demand} = \boldsymbol{Ref\ Demand} + (\boldsymbol{EDF} * \boldsymbol{Heating\ Demand})$$

Where: $\boldsymbol{Total\ Demand}$ is the total demand on the heat pump, $\boldsymbol{Ref\ Demand}$ is the absolute value of the refrigerant demand reported by the refrigerant collection (The refrigerant demand reported here is just the $\boldsymbol{Heating\ demand} - \boldsymbol{Cooling\ Demand}$), $\boldsymbol{EDF}$ is the Exchange Demand Factor and $\boldsymbol{Heating\ Demand}$ is the cooling demand reported at the refrigerant collection.

- Consumption per Port (Heat) – This field allows the user to model additional consumption per component in the Refrigerant group connected to the heat pump which requires heating in a given hour. The user enters a value in watts and it is multiplied by the number of components requiring heating in a given hour. The amount is then included in the reported consumption of the chiller.
- Consumption per Port (Cool) – Works in the same way as the Consumption per Port (Heat) but instead allows for additional consumption per component in the Refrigerant group connected to the heat pump which requires cooling in a given hour.

- VRF with BC box – This option should only be used with Mitsubishi Heat Pumps. As Mitsubishi components are available to be imported from the manufacturer database, there is no need to use this option manually.
- HVRF with BC box - This option should only be used with Mitsubishi Heat Pumps. As Mitsubishi components are available to be imported from the manufacturer database, there is no need to use this option manually.

- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand. In TAS Systems, the demand met by a component is reported for each hour in the results section. The user has two methods of setting the duty of the heat pump, with the first option being “Values”. Upon choosing this option, the user is presented with the following fields:
  - Cooling Duty – The Cooling Duty field allows the user to enter in the cooling duty. This value is the upper limit on the amount of power the heat pump can provide for cooling. Please note that the user will be able to use modifiers with this field by going to the “Cooling Capacity” tab.
  - Heating Duty – The Heating Duty field allows the user to enter in the heating duty. This value is the upper limit on the amount of power the heat pump can provide for heating. Please note that the user will be able to use modifiers with this field by going to the “Heating Capacity” tab.
  - Cooling Power – The Cooling Power property is used by TAS to dictate the rate of energy consumption by the heat pump when cooling. By default, this field has an efficiency partload profile set up (by using a modifier) which requires the user to enter the rate of energy consumption at full load as a value. If the user deletes the modifier and doesn’t create a new one, the heat pump will always use the value entered as the consumption when the heat pump is cooling.
  - Heating Power – The Heating Power property is used by TAS to dictate the rate of energy consumption by the heat pump when heating. By default, this field has an efficiency partload profile set up (by using a modifier) which requires the user to

enter the rate of energy consumption at full load as a value. If the user deletes the modifier and doesn't create a new one, the heat pump will always use the value entered as the consumption when the heat pump is heating.

- Extra Fan load – The Extra Fan load property allows the user to set the consumption of the fan in the air source heat pump. By default, this field has an efficiency partload profile set up (by using a modifier) which requires the user to enter the rate of energy consumption when the fan is at full load. If the user deletes the modifier and doesn't create a new one, the heat pump will always use the value entered as the consumption of the fan whenever the heat pump is heating or cooling. Please note that consumption by the fan is included in the overall consumption of the heat pump.

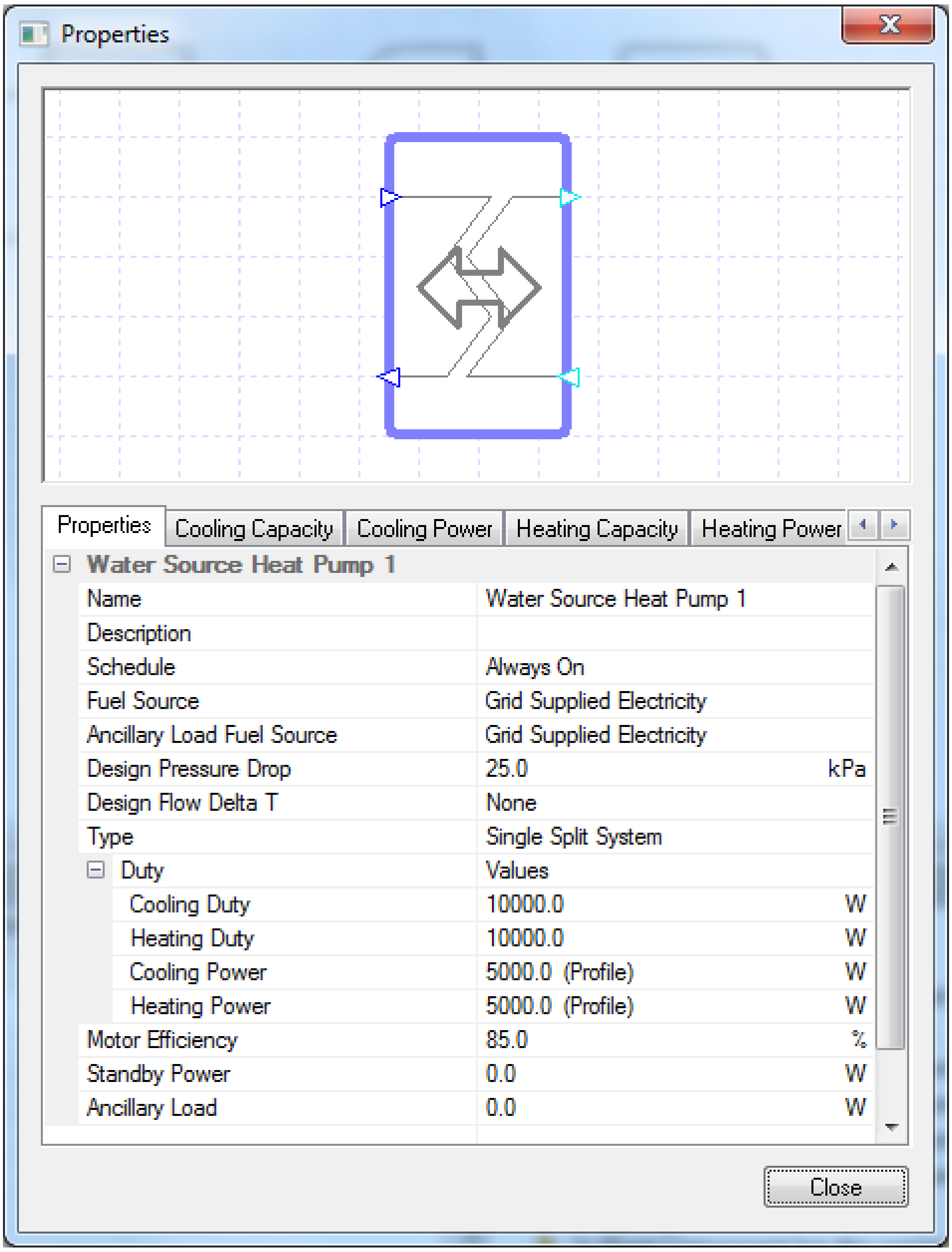
The second option available for setting duty of the heat pump is "sized". Upon choosing this option, the user is presented with the following fields:

- Heating: Cooling duty ratio – This field allows the user to set up a ratio between the heating duty and the cooling duty. The user will have four options to choose from:
    - Value – Upon choosing this option the user enters in the ratio as a value. The value the user enters will be the heating part of the ratio when the cooling side is one (So if the ratio was 3:1, the user would enter a value of 3). After sizing the heating and cooling duty's during a simulation, TAS will modify one of the duties so that the ratio is maintained. TAS will do this so that neither duty is less than the original sized value.
    - Heating only – This option should only be used if the heat pump is only being used to provide heat. This option sets the cooling duty to zero, meaning the heat pump cannot provide any cooling.
    - Cooling only – This option should only be used if the heat pump is only being used for cooling purposes. This option sets the heating duty to zero, meaning the heat pump cannot provide any heating.
    - Not Used – The Not Used option will not change the heating and cooling duties from the sized values taken from the simulation.
- Cooling Energy Input Ratio – This ratio is used to set the rate of energy consumption by the heat pump when it is cooling at its maximum cooling duty. The ratio used is cooling power: cooling duty and the value entered is the cooling power value when the cooling duty is one (So if the ratio was 3:1, the user would enter a value of 3 in this field). Please note that any modifier set in the Cooling Power Profile will be applied to the cooling power worked out by this ratio
- Heating Energy Input Ratio - This ratio is used to set the rate of energy consumption by the heat pump when it is heating at its maximum heating duty. The ratio used is heating power: heating duty and the value entered is the heating power value when the heating duty is one (So if the ratio was 3:1, the user would enter a value of 3 in this field). Please note that any modifier set in the Heating Power Profile will be applied to the Heating power worked out by this ratio.
- Fan Load : Duty Ratio – This ratio is used to set the rate of energy consumption by the fan in the heat pump when the fan is at full load. The duty used in the ratio is the maximum of the Heating and Cooling duties. The value entered in this field will be the fan load part of the ratio when the duty part is one (So if the ratio was 0.3:1, the user would enter a value of 0.3 in this field). Please note that any modifier set in the Condenser Fan Load tab will be applied to the Extra Fan Load worked out by this ratio.

When using the sizing options, please note that the options available when you choose "Value" are still visible but greyed out. This will be populated after a simulation so you can see the duties etc.

- **Standby Power –** This field sets the amount of power consumed in an hour by the heat pump when it is not being used during scheduled hours.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the air source heat pump. Any value entered here will be the associated load during any hour the heat pump operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.17 Water Source Heat Pump



The Water Source Heat Pump component in TAS systems transfers the load from the refrigerant group to the connected fluid circuit loop. Please note that the light blue ports are refrigerant ports and should connect up to the refrigerant group while the dark blue ports are fluid ports and should connect up to the fluid circuit loop. Please note that controllers cannot be used with this component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the water source heat pump, no heating or cooling demand from a

refrigerant group would be met. The default schedule option is always on, meaning that the component will operate 24/7.

- **Fuel Source -** With this field the user can choose the fuel source of the water source heat pump. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the water source heat pump.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid (not the refrigerant) through the component. Please note that this is for the fluid circuit loop only, the refrigerant circuit is unaffected by this value. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the chillers circuit.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band (please note this is for the fluid loop, not the refrigerant loop). The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x-y}$ to $\boldsymbol{x+y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Type –** This field allows the user to set how the water source heat pump works. There are 4 options to choose from, but only 3 should be used by the user:
    - Single Split – TAS models one water source heat pump for each component in the Refrigerant group. The demand and consumption figures are then worked out for each heat pump separately before being summed and reported by the Water Source Heat Pump component. Please note that each individual heat pump has identical parameters and the parameters are the ones entered into the component. So, for instance, if you enter the cooling duty as 1000W into the heat pump component then each individual heat pump's cooling duty will be 1000W.
    - Multi Split – TAS models one air source heat pump for all components in the Refrigerant group. With a multi – split heat pump, heating and cooling cannot be provided at the same time. This means that for a hour where there is both a heating and cooling demand, the heat pump will cater for the larger demand while not being able to meet the smaller one (TAS will inform you, with a warning, of the hours this occurs).
    - VRF - Like the Multi-Split option, the VRF option models one water source heat pump dealing with the load from all the components in the Refrigerant group. The difference between the two is that the VRF option allows for both heating and cooling to be provided in a certain hour. The VRF option keeps the refrigerant flow for heating and cooling separate. The heat pump will then meet the larger of the heating / cooling demands and heat/cool the associated refrigerant to meet it. To

meet the smaller of the two demands, the heat pump will use the rejected load from the heat pump to heat/cool the refrigerant on the other refrigerant path. To set up the VRF option correctly, TAS will ask the user to fill in the following fields:

- Exchange Demand Factor (Heat) – When the heat pump has both a heating and cooling demand to meet with the heating demand being the bigger demand, the heat pump can attempt to meet the cooling demand by using the rejected cool air created while meeting the heating demand. The Exchange Demand Factor (Heat) represents the extra work required to do this. The following equation states how the Exchange Demand Factor (Heat) is used to work out the total demand on the heat pump, which is then used to calculate the consumption of the heat pump:

$$\boldsymbol{Total\ Demand\ = Ref\ Demand + (EDF * Cooling\ Demand),}$$

  Where: $\boldsymbol{Total\ Demand}$ is the total demand on the heat pump, $\boldsymbol{Ref\ Demand}$ is the absolute value of the refrigerant demand reported by the refrigerant collection (The refrigerant demand reported here is just the $\boldsymbol{Heating\ demand - Cooling\ Demand}$), $\boldsymbol{EDF}$ is the Exchange Demand Factor and $\boldsymbol{Cooling\ Demand}$ is the cooling demand reported at the refrigerant collection.

- Exchange Demand Factor (Cool) – When the heat pump has both a heating and cooling demand to meet with the Cooling demand being the bigger demand, the heat pump can attempt to meet the heating demand by using the rejected hot air created while meeting the cooling demand. The Exchange Demand Factor (Cool) represents the extra work required to do this. The following equation states how the Exchange Demand Factor (cool) is used to work out the total demand on the heat pump, which is then used to calculate the consumption of the heat pump:

$$\boldsymbol{Total\ Demand\ = Ref\ Demand + (EDF * Heating\ Demand),}$$

  Where: $\boldsymbol{Total\ Demand}$ is the total demand on the heat pump, $\boldsymbol{Ref\ Demand}$ is the absolute value of the refrigerant demand reported by the refrigerant collection (The refrigerant demand reported here is just the $\boldsymbol{Heating\ demand - Cooling\ Demand}$), $\boldsymbol{EDF}$ is the Exchange Demand Factor and $\boldsymbol{Heating\ Demand}$ is the cooling demand reported at the refrigerant collection.

- Consumption per Port (Heat) – This field allows the user to model additional consumption per component in the Refrigerant group connected to the heat pump which requires heating in a given hour. The user enters a value in watts and it is multiplied by the number of components requiring heating in a given hour. The amount is then included in the reported consumption of the chiller.
- Consumption per Port (Cool) – Works in the same way as the Consumption per Port (Heat) but instead allows for additional consumption per component in the Refrigerant group connected to the heat pump which requires cooling in a given hour.

  - VRF with BC box – This option should only be used with Mitsubishi Heat Pumps. As Mitsubishi components are available to be imported from the manufacturer database, there is no need to use this option manually.

- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand. In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. The user has two methods of setting the duty of the heat pump, with the first option being "Values". Upon choosing this option, the user is presented with the following fields:

- Cooling Duty – The Cooling Duty field allows the user to enter in the cooling duty. This value is the upper limit on the amount of power the heat pump can provide for cooling. Please note that the user will be able to use modifiers with this field by going to the "Cooling Capacity" tab.
- Heating Duty – The Heating Duty field allows the user to enter in the heating duty. This value is the upper limit on the amount of power the heat pump can provide for heating. Please note that the user will be able to use modifiers with this field by going to the "Heating Capacity" tab.
- Cooling Power – The Cooling Power property is used by TAS to dictate the rate of energy consumption by the heat pump when cooling. By default, this field has an efficiency partload profile set up (by using a modifier) which requires the user to enter the rate of energy consumption at full load as a value. If the user deletes the modifier and doesn't create a new one, the heat pump will always use the value entered as the consumption when the heat pump is cooling.
- Heating Power – The Heating Power property is used by TAS to dictate the rate of energy consumption by the heat pump when heating. By default, this field has an efficiency partload profile set up (by using a modifier) which requires the user to enter the rate of energy consumption at full load as a value. If the user deletes the modifier and doesn't create a new one, the heat pump will always use the value entered as the consumption when the heat pump is heating.
- Extra Fan load – The Extra Fan load property allows the user to set the consumption of the Fan in the water source heat pump. By default, this field has an efficiency partload profile set up (by using a modifier) which requires the user to enter the rate of energy consumption when the fan is at full load. If the user deletes the modifier and doesn't create a new one, the heat pump will always use the value entered as the consumption of the fan whenever the heat pump is heating or cooling. Please note that consumption by the fan is included in the overall consumption of the heat pump.

The second option available for setting duty of the heat pump is "sized". Upon choosing this option, the user is presented with the following fields:

- Heating: Cooling duty ratio – This field allows the user to set up a ratio between the heating duty and the cooling duty. The user will have four options to choose from:
  - Value – Upon choosing this option the user enters in the ratio as a value. The value the user enters will be the heating part of the ratio when the cooling side is one (So if the ratio was 3:1, the user would enter a value of 3). After sizing the heating and cooling duty's during a simulation, TAS will modify one of the duties so that the ratio is maintained. TAS will do this so that neither duty is less than the original sized value.
  - Heating only – This option should only be used if the heat pump is only being used to provide heat. This option sets the cooling duty to zero, meaning the heat pump cannot provide any cooling.
  - Cooling only – This option should only be used if the heat pump is only being used for cooling purposes. This option sets the heating duty to zero, meaning the heat pump cannot provide any heating.
  - Not Used – The Not Used option will not change the heating and cooling duties from the sized values taken from the simulation.
- Cooling Energy Input Ratio – This ratio is used to set the rate of energy consumption by the heat pump when it is cooling at its maximum cooling duty. The ratio used is cooling power: cooling duty and the value entered is the cooling power value when

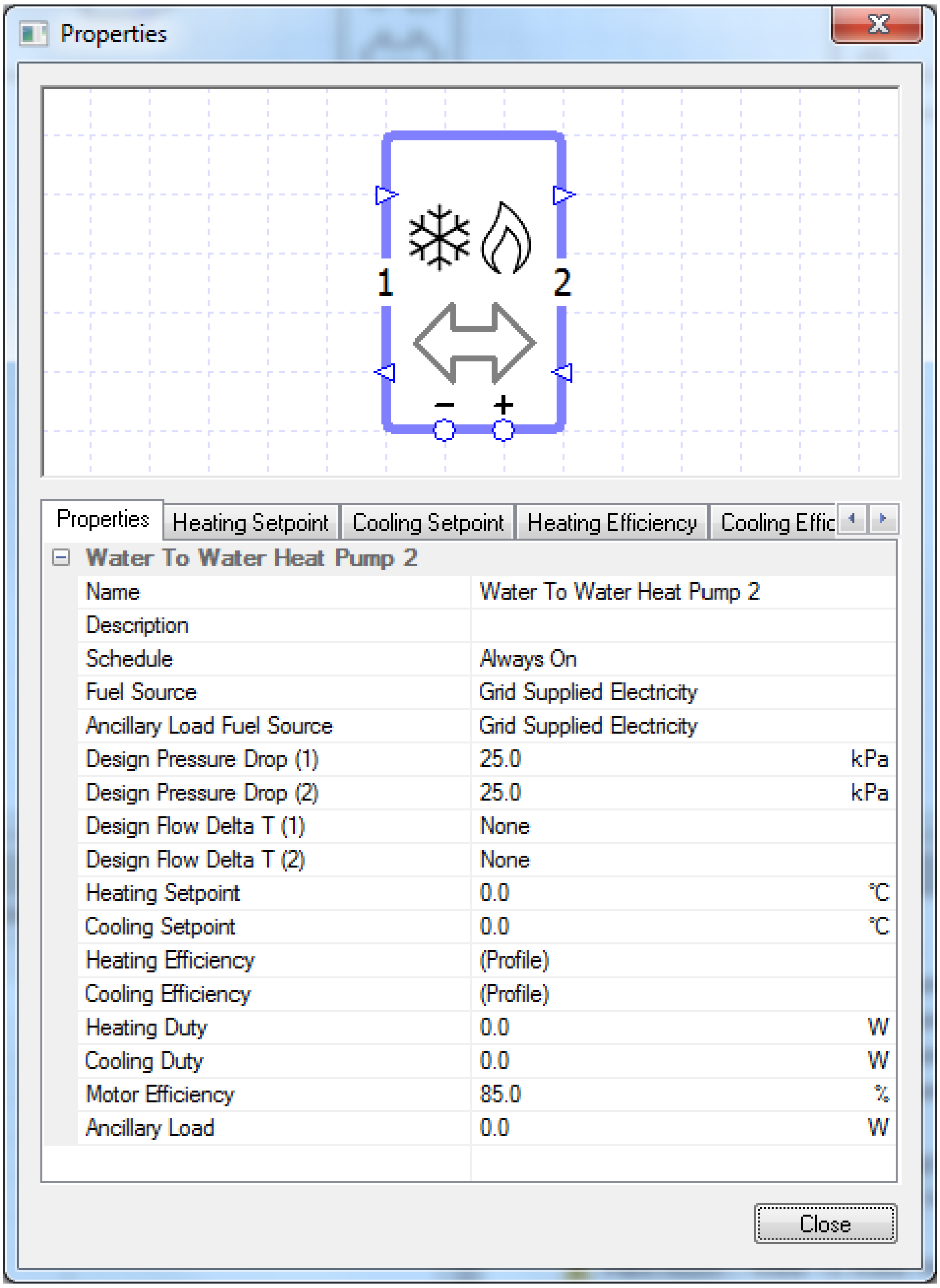
the cooling duty is one (So if the ratio was 3:1, the user would enter a value of 3 in this field). Please note that any modifier set in the Cooling Power Profile will be applied to the cooling power worked out by this ratio.

- Heating Energy Input Ratio - This ratio is used to set the rate of energy consumption by the heat pump when it is heating at its maximum heating duty. The ratio used is heating power: heating duty and the value entered is the heating power value when the heating duty is one, i.e. if the ratio was 3:1, the user would enter a value of 3 in this field. Please note that any modifier set in the Heating Power Profile will be applied to the Heating power worked out by this ratio.
- Fan Load : Duty Ratio – This ratio is used to set the rate of energy consumption by the fan in the heat pump when the fan is at full load. The duty used in the ratio is the maximum of the heating and cooling duties. The value entered in this field will be the fan load part of the ratio when the duty part is one, i.e. if the ratio was 0.3:1, the user would enter a value of 0.3 in this field. Please note that any modifier set in the Condenser Fan Load tab will be applied to the extra fan load worked out by this ratio.

When using the sizing options, please note that the options available when you choose “Value” are still visible but greyed out. This will be populated after a simulation so you can see the duties etc.

- **Motor Efficiency** – The Motor Efficiency field allows the user to set the heat pump’s motor efficiency. This efficiency of the motor details how efficient the heat pump is at transferring the heat created due to the consumption over to the heat rejection loop. The user enters in a percentage into this field and this percentage of the heat generated due to the consumption is transferred over to the heat rejection loop. If the efficiency is less than 100%, some of heat created due to the consumption will be emitted outside of the chiller rather than being transferred to the heat rejection loop.
- **Standby Power –** This field sets the amount of power consumed in an hour by the heat pump when it is not being used during scheduled hours.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the water source heat pump. Any value entered here will be the associated load during any hour the heat pump operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.18 Water to Water Heat Pump



The water to water heat pump allows the user to connect a heat pump up to fluid collections, like heating and cooling collections, rather than to refrigerant collections. Please note that the water to water heat pump should only be connected to either a heating collection or a cooling collection, as connecting it to both may cause issues when there is a heating and cooling demand in the same hour. The water to water heat pump connects up to two fluid loops, denoted 1 and 2 when the component is selected. The loop denoted 1, called Loop 1, is the loop heat will be rejected to. The loop denoted 2, called Loop 2, is the loop that will be heated or cooled by the heat pump.

The water to water heat pump can be used in conjunction with a controller, to control the heat pump. If a controller is connected to the plus signal port, then the signal the controller sends details the proportion of the heat pump's heating duty it should use to heat the fluid. If a controller is connected to the minus signal port, then the signal the controller sends details the proportion of the heat pumps's cooling duty it should use to cool the fluid. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint fields is assumed to be directly after the return port on Loop 2.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the water to water heat pump, no heating or cooling demand would be met. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source -** With this field the user can choose the fuel source of the water to water heat pump. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the water source heat pump.
- **Design Pressure Drop (1) –** This field allows the user to enter the pressure drop of the fluid on Loop 1 (i.e. the heat rejection loop) when flowing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the left loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Design Pressure Drop (2) –** This field allows the user to enter the pressure drop of the fluid on Loop 1 (i.e. the heating / cooling loop) when flowing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the left loop's circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Design Flow Delta T (1) –** This field allows the user to size the flow rate of the heat rejection loop of the component such that the fluid flowing through the component is kept within a certain temperature band (Please note that this is the loop marked with a 1 when you click on the component). The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of x and a Design Flow Delta T of y, then the fluid will always be kept within the temperature range $x - y$ to $x + y$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Design Flow Delta T (2) –** The Design Flow Delta T (2) field works in the same way as the Design Flow Delta T (1) field, which is described above, but affects the heating/cooling loop instead of Loop 1. I.e. the loop marked with a 2 when you click on the component. Please read the description of the Design Flow Delta T (1) field for how this works.

- **Heating Setpoint –** When a temperature is entered into the Heating Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the water to water heat pump, it will heat the fluid so it reaches the heating setpoint. Please note that when a controller is used in conjunction with the water to water heat pump, the Heating Setpoint field will disappear from the properties. This is done because the water to water heat pump is being controlled by a controller and will heat the fluid when the controller sends a signal informing the water to water heat pump to do so. When the Heating Setpoint field is visible, modifiers can be added to the setpoint using the Heating Setpoint tab.
- **Cooling Setpoint –** When a temperature is entered into the Cooling Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the water to water heat pump, it will cool the fluid so it reaches the cooling setpoint. Please note that when a controller is used in conjunction with the water to water heat pump, the Cooling Setpoint field will disappear from the properties. This is done because the water to water heat pump is being controlled by a controller and will cool the fluid when the controller sends a signal informing the water to water heat pump to do so. When the Cooling Setpoint field is visible, modifiers can be added to the setpoint using the Cooling Setpoint tab.
- **Heating Efficiency –** Upon clicking on the Heating Efficiency field, you will be transferred over to the Heating Efficiency tab where you can create a profile for the heating efficiency of the water to water heat pump. Normally the modifier chosen here would be a table modifier with a partload profile.
- **Cooling Efficiency –** Upon clicking on the Cooling Efficiency field, you will be transferred over to the Cooling Efficiency tab where you can create a profile for the cooling efficiency of the water to water heat pump. Normally the modifier chosen here would be a table modifier with a partload profile. The cooling efficiency input for a heat pump should be the EER.
- **Heating Duty –** The heating duty of a component is the upper limit on the amount of power a component can provide for heating. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a water to water heat pump, if the heating demand is bigger than the heating duty it means that the heat pump will not be able to cover the heating demand.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the heating duty of the component.

  In the Heating Duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Cooling Duty –** The cooling duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this

demand (For a water to water heat pump, if the cooling demand is bigger than the cooling duty it means that the heat pump will not be able to cover the cooling demand.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:

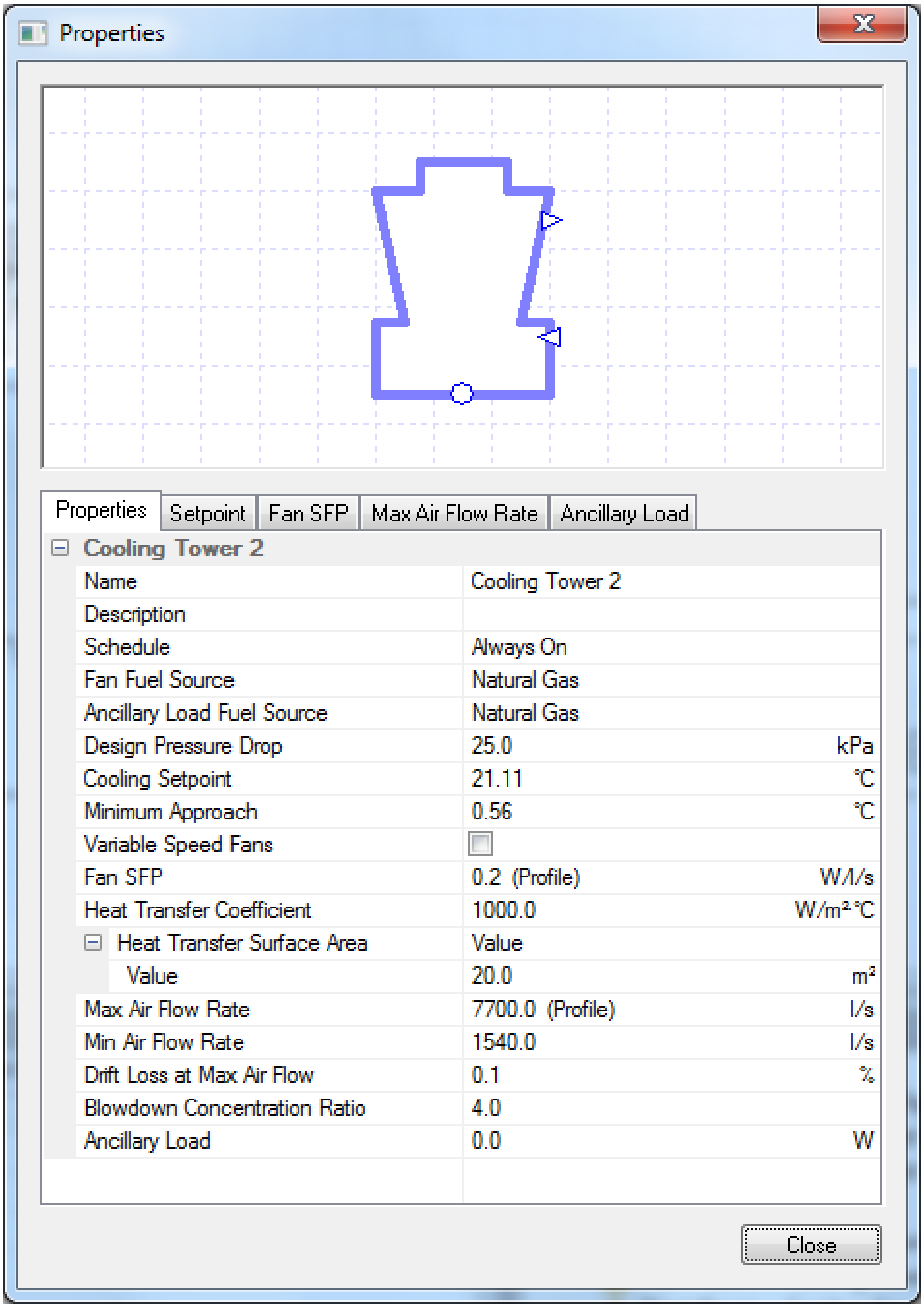
  - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
  - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
    - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
    - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.

    Please note that to size the duty the user will need to have design conditions in their systems file.
  - Value – With this option the user will type in the cooling duty of the component.

  In the Cooling Duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

- **Motor Efficiency** – The Motor Efficiency field allows the user to set the chiller's motor efficiency. This efficiency of the motor details how efficient the chiller is at transferring the heat created due to the consumption over to the heat rejection loop. The user enters in a percentage into this field and this percentage of the heat generated due to the consumption is transferred over to the heat rejection loop. If the efficiency is less than 100%, some of heat created due to the consumption will be emitted outside of the chiller rather than being transferred to the heat rejection loop.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the water to water heat pump. Any value entered here will be the associated load during any hour the heat pump operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.19 Cooling Tower



The Cooling Tower component in TAS Systems allows the user to model a fan driven cooling tower in TAS Systems. The cooling tower rejects heat by passing the fluid through a moving air stream. Controllers can be used with the cooling tower to control it by informing it of the amount of air flow the fans should provide to facilitate the heat exchange. The controller does this by sending a signal, between zero and one, to the cooling tower dictating the proportion of the cooling tower's max air flow rate it should use to cool down the fluid. So, for example, if the cooling tower received a signal of zero then the fans will be switched off for the whole hour. While if it received a signal of 1 the cooling tower's fans would provide the maximum flow rate specified in the Max Air Flow Rate field. If no controller is used, the sensor used to determine the fluid's temperature for the Cooling Setpoint field is assumed to be directly after the return port of the cooling tower.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the cooling tower, this will mean that the fluid will flow through the cooling tower uncooled, as the cooling tower fans will not operate. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fan Fuel Source –** With this field the user can choose the fuel source of the cooling tower's fans. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the cooling tower.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Cooling Setpoint -** When a temperature is entered into the Cooling Setpoint field, the component will attempt to regulate the temperature of the fluid going through it by cooling the fluid to reach this setpoint. It should be noted that the cooling tower will not always be able to meet this setpoint. For instance if the cooling setpoint of the cooling tower is set to lower than the external wet bulb temperature for a given hour then the cooling tower will only be able to cool the fluid down to the external wetbulb temperature in this hour. Please note that when a controller is used in conjunction with the cooling tower, the Cooling Setpoint field will disappear from the properties. This is done because the cooling tower is being controlled by a controller rather than being controlled using this setpoint field. When the Setpoint field is visible, modifiers can be added to the setpoint using the Cooling Setpoint tab.
- **Minimum Approach –** When the temperature difference between the fluid the cooling tower is cooling and the external wet bulb temperature is very small, the cooling tower will become very inefficient at rejecting heat. To get around this problem, the user can enter a minimum approach, which creates a buffer zone to prevent inefficient operation. The minimum approach works in the following way. If the user enters the value x into this field, then when the air's wet-bulb temperature is within x of the cooling setpoint of the cooling tower or when the cooling setpoint of the cooling tower is above the external wetbulb temperature, TAS will set the cooling setpoint to:

  $$\boldsymbol{Cooling\ Setpoint = Air's\ wetbulb\ temperature + x.}$$

  Please note that depending on the properties of the cooling tower, it may not always be able to meet this revised setpoint.
- **Variable Speed Fans –** This tick box field allows the user to decide on how the fans in the cooling tower work. If this field is left un-ticked, the air flow rates entered in the Max Air Flow Rate field and the Min Air Flow Rate field are the only air flow rates the fans can provide. As the fans can't provide any other flow rate, they alternate between the two air flow rates to cover partial loads. If this field is ticked, the Fans can provide any air flow rate between the max air flow rate (which will be the maximum allowed air flow) and the min air

flow rate (which will be the minimum allowed air flow). Allowing the fans to be variable provides better performances at part loads. Please note that for both options, if the cooling tower has a partial load which requires an air flow rate below the minimum air flow rate, the cooling tower will meet this by turning the fan on and off. Also note that TAS Systems reports the average air flow rate for the hour in its result section which means, for instance, the reported air flow rate for the hour can be less than the minimum air flow rate.

- **Fan SFP –** The Fan SFP field allows the user to enter the SFP of the cooling tower's fans. By default the fan has a modifier profile set up in the Fan SFP tab and the user only needs to enter the SFP at the fan's full load.
- **Heat Transfer Coefficient –** This field allows the user to input the heat transfer coefficient between the air and water streams.
- **Heat Transfer Surface Area –** This field allows the user to input the surface area where heat transfer can occur between the air and water streams. This service area can be set using two different methods. If the "Value" option is selected, then the user will be asked to enter in the heat transfer surface area. If the "Sized" option is selected, then the heat transfer surface area will be sized based on design conditions entered into the cooling tower properties. These design conditions will be discussed in their own property fields as they can also be used for sizing the max air flow rate. Please note that a size fraction also appears, allowing the area to be oversized or undersized.
- **Max Air Flow Rate –** The Max Air Flow Rate field allows the user to specify the maximum permitted air flow through the cooling tower. Please note that how this maximum is used varies depending on the choice made for the Variable Speed Fans field. Depending on other properties, there are three options to choose from when setting the max air flow rate:
  - Value – Upon Choosing the Value option the user is asked to enter the max air flow rate. This option is the default option and is always available.
  - Fan Load Ratio – This option is only available when the Heat Transfer Surface Area field is sized, as it will make use of the design condition options that appear when that field is sized. Upon choosing the Fan Load Ratio option, a Fan Load Ratio field will appear asking the user to enter in the ratio of the maximum fan load to the maximum cooling load.
  - Air Flow / Water Flow Ratio - This option is only available when the Heat Transfer Surface Area field is sized, as it will make use of the design condition options that appear when that field is sized. Upon choosing this option, an Air Flow / Water Flow Ratio field will appear asking the user to enter the ratio of the maximum air flow rate to the maximum fluid flow rate.

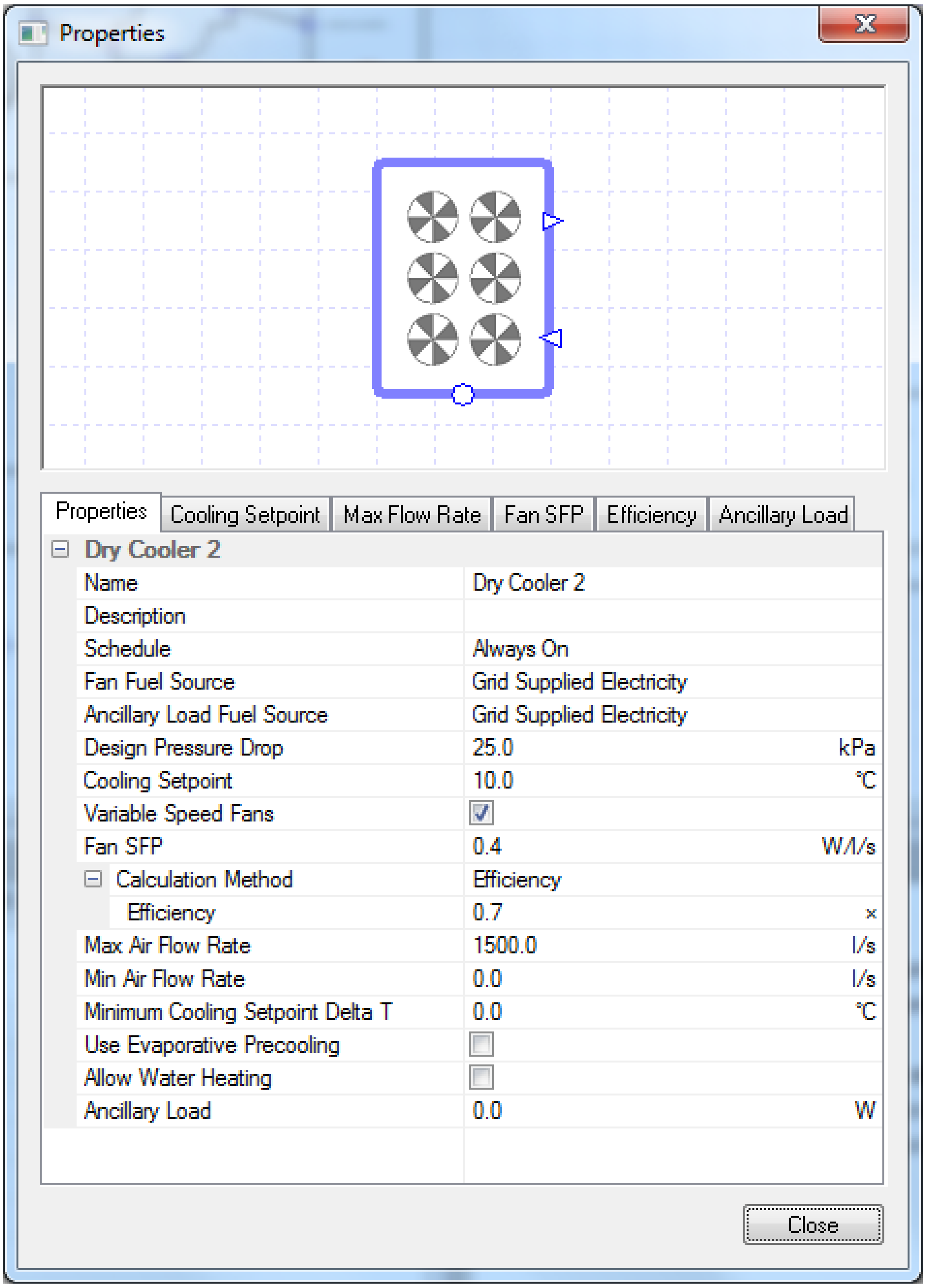
  Please note that modifiers can be used with this field by going to the Max Air Flow Rate field. By default, a modifier is already applied to this field.
- **Min Air Flow Rate –** The Min Air Flow Rate field allows the user to specify the minimum permitted air flow the cooling tower fans can provide. Please note that how this minimum is used varies depending on the choice made for the Variable Speed Fans field. How the minimum air flow rate is set depends on how the max air flow rate is calculated. If the Max Air Flow Rate field is set to "Value" then the user will be required to enter in the minimum air flow rate into this field. However if the Max Air Flow Rate field is not set to "Value" then the user will be required to enter in the minimum air flow sizing ratio into the Min Air Flow Sizing Ratio field. This ratio would be the ratio between the minimum air flow rate and the maximum air flow rate.
- **Design Water Flow Rate** – This field is only available when the Heat Transfer Surface Area field is sized and sets the flow rate of the water entering the cooling tower on the design condition. There are two options to choose from when using this field:
  - Value – With this option the user must enter in the water flow rate to be used on the sizing condition.

    - As Pump Design Flow Rate – With this option the design flow rate from the pump on the heat rejection loop will be used in the design conditions.
- **Design External Wetbulb –** This field is only available when the Heat Transfer Surface Area field is sized and sets the external wetbulb temperature used on the design condition. There are three options to choose from with this field:
    - Value – Upon choosing this option, the user would enter the design external wetbulb temperature as a value.
    - Peak External – Upon choosing this option, TAS will take the peak external wet bulb temperature from the weather data as the design external wetbulb for the sizing run.
    - Max Operating – Upon choosing this option, the user is asked to enter a limiting wetbulb temperature. TAS will then compare this temperature with the peak external wetbulb temperature from the weather data and use the minimum value as the design external wetbulb temperature.
- **Design Approach -** This field is only available when the Heat Transfer Surface Area field is sized and sets the minimum approach used on the design conditions.
- **Design Range -** This field is only available when the Heat Transfer Surface Area field is sized and sets the water temperature drop of the fluid flowing through the cooling tower used on the design conditions.
- **Drift Loss at Max Air Flow –** The Drift Loss at Max Air Flow field allows the user to enter the proportion of the water flow lost due to water droplets being carried out with the exhaust air. The percentage entered in this field will be the percentage loss when the air flow is at its maximum rate. As the air flow rate decreases, the drift loss percentage will decrease linearly as well.
- **Blowdown Concentration Ratio –** The Blowdown Concentration Ratio field is used to calculate the amount of blowdown. As some of the water in the cooling tower evaporates, the concentration of impurities in the cooling tower water will increase. To maintain the concentration within acceptable levels, some of the cooling tower water is rejected (which is called blowdown) and replaced with fresh water. In this field the user is required to enter the ratio

$$\textbf{\textit{Impurity concentration in cooling tower water}} \\ : \textbf{\textit{Impurity concentration in fresh water}}$$

  To enter the ratio into this field, the user will enter the value x, where the ratio is 1:x . TAS will then use this ratio to maintain the impurity concentration of the cooling tower water, any water added to the cooling tower will be reported in the results under the make-up water column.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the cooling tower. Any value entered here will be the associated load during any hour the heat pump operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.20 Dry Cooler



A dry cooler is a heat rejection component which rejects heat from the fluid entering it to the outside air. Controllers can be used with the dry cooler to control it by informing it of the amount of air flow the fans should provide to facilitate the heat exchange. The controller does this by sending a signal, between zero and one, to the dry cooler dictating the proportion of the dry cooler's max air flow rate it should use to cool down the fluid. So, for example, if the dry cooler received a signal of zero then the fans will be switched off for the whole hour. While if it received a signal of 1 the dry cooler's fans would provide the maximum flow rate specified in the Max Air Flow Rate field. If no controller is used, the sensor used to determine the fluid's temperature for the Cooling Setpoint field is assumed to be directly after the return port of the dry cooler.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Fan Fuel Source –** With this field the user can choose the fuel source of the dry cooler's fans. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the dry cooler.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Cooling Setpoint -** When a temperature is entered into the Cooling Setpoint field, the component will attempt to regulate the temperature of the fluid going through it by cooling the fluid to reach this setpoint. It should be noted that the dry cooler will not always be able to meet this setpoint. For instance if the cooling setpoint of the dry cooler is set to lower than the external drybulb temperature for a given hour then the dry cooler will only be able to cool the fluid down to the external drybulb temperature in this hour. Please note that when a controller is used in conjunction with the dry cooler, the Cooling Setpoint field will disappear from the properties. This is done because the dry cooler is being controlled by a controller rather than being controlled using this setpoint field. When the Setpoint field is visible, modifiers can be added to the setpoint using the Cooling Setpoint tab.
- **Variable Speed Fans –** This tick box field allows the user to decide on how the fans in the dry cooler work. If this field is left un-ticked, the air flow rates entered in the Max Air Flow Rate field and the Min Air Flow Rate field are the only air flow rates the fans can provide. As the fans can't provide any other flow rate, they alternate between the two air flow rates to cover partial loads. If this field is ticked, the fans can provide any air flow rate between the max air flow rate (which will be the maximum allowed air flow) and the min air flow rate (which will be the minimum allowed air flow). Allowing the fans to be variable provides better performances at part loads. Please note that for both options, if the dry cooler has a partial load which requires an air flow rate below the minimum air flow rate, the dry cooler will meet this by turning the fan on and off. Also note that TAS Systems reports the average air flow rate for the hour in its result section which means, for instance, the reported air flow rate for the hour can be less than the minimum air flow rate.
- **Fan SFP –** The Fan SFP field allows the user to enter the SFP of the dry cooler's fans. Modifiers can be applied to this field by going to the Fan SFP tab.
- **Calculation Method –** In TAS systems, there are two options to calculate the maximum rate of heat rejection for the dry cooler:
  - Efficiency – The first calculation method is efficiency. Upon choosing this option the user is asked to enter the efficiency of the dry cooler. The efficiency of the dry cooler is calculated by the following formula:

$$\boldsymbol{Efficency} = \frac{\textbf{temp of fluid leaving dry cooler} - \textbf{temp of fluid entering dry cooler}}{\textbf{external drybulb temp} - \textbf{temp of fluid entering dry cooler}}.$$

    Please note that in the Efficiency Tab, you can add a modifier to the Efficiency.
  - NTU Method – The second calculation method is the NTU method. Upon choosing this method, the user will need to input: The heat transfer surface area, the heat transfer coefficient and the Exchanger type. After entering these details, TAS will then work out the rate of heat transfer of the Exchanger.

- **Max Air Flow Rate –** The Max Air Flow Rate field allows the user to set the maximum air flow rate for the dry cooler. The value entered here is only an upper limit on how big the air flow can be. If the water flow means that a reduced air flow rate can provide the max amount of heat rejection then this reduced value will be used as the max. After obtaining the max air flow, a linear profile is then used to work out the air flow at partial loads. The user is given two methods to set the value:
  - Value – If the user chooses this option then the user will have to enter the max air flow rate as a value.
  - Sized – Upon choosing this option, the max air flow rate will be sized on a design conditions that will appear in the dry cooler's properties. These design conditions will be discussed in their own fields.

  Please note that the user can set up a modifier for this field, using the Max Flow Rate tab.
- **Min Air Flow Rate –** The Min Air Flow Rate field allows the user to specify the minimum permitted air flow the dry cooler fans can provide. Please note that how this minimum is used varies depending on the choice made for the Variable Speed Fans field. How the minimum air flow rate is set depends on how the max air flow rate is calculated. If the Max Air Flow Rate field is set to "Value" then the user will be required to enter in the minimum air flow rate into this field. However if the Max Air Flow Rate field is set to "Sized" then the user will be required to enter in the minimum air flow sizing ratio into the Min Air Flow Sizing Ratio field. This ratio would be the ratio between the minimum air flow rate and the maximum air flow rate.
- **Minimum Cooling Setpoint Delta T –** When the temperature difference between the fluid the dry cooler is cooling and the external drybulb temperature is very small, the dry cooler will become very inefficient at rejecting heat. To get around this problem, the user can enter a minimum cooling setpoint delta T, which creates a buffer zone to prevent inefficient operation. The minimum cooling setpoint delta T works in the following way. If the user enters the value x into this field, then when the air's drybulb temperature is within x of the cooling setpoint of the dry cooler, or when the cooling setpoint of the dry cooler is above the external drybulb temperature, TAS will set the cooling setpoint to:

  $$\boldsymbol{Cooling\ Setpoint = Air's\ drybulb\ temperature + x.}$$

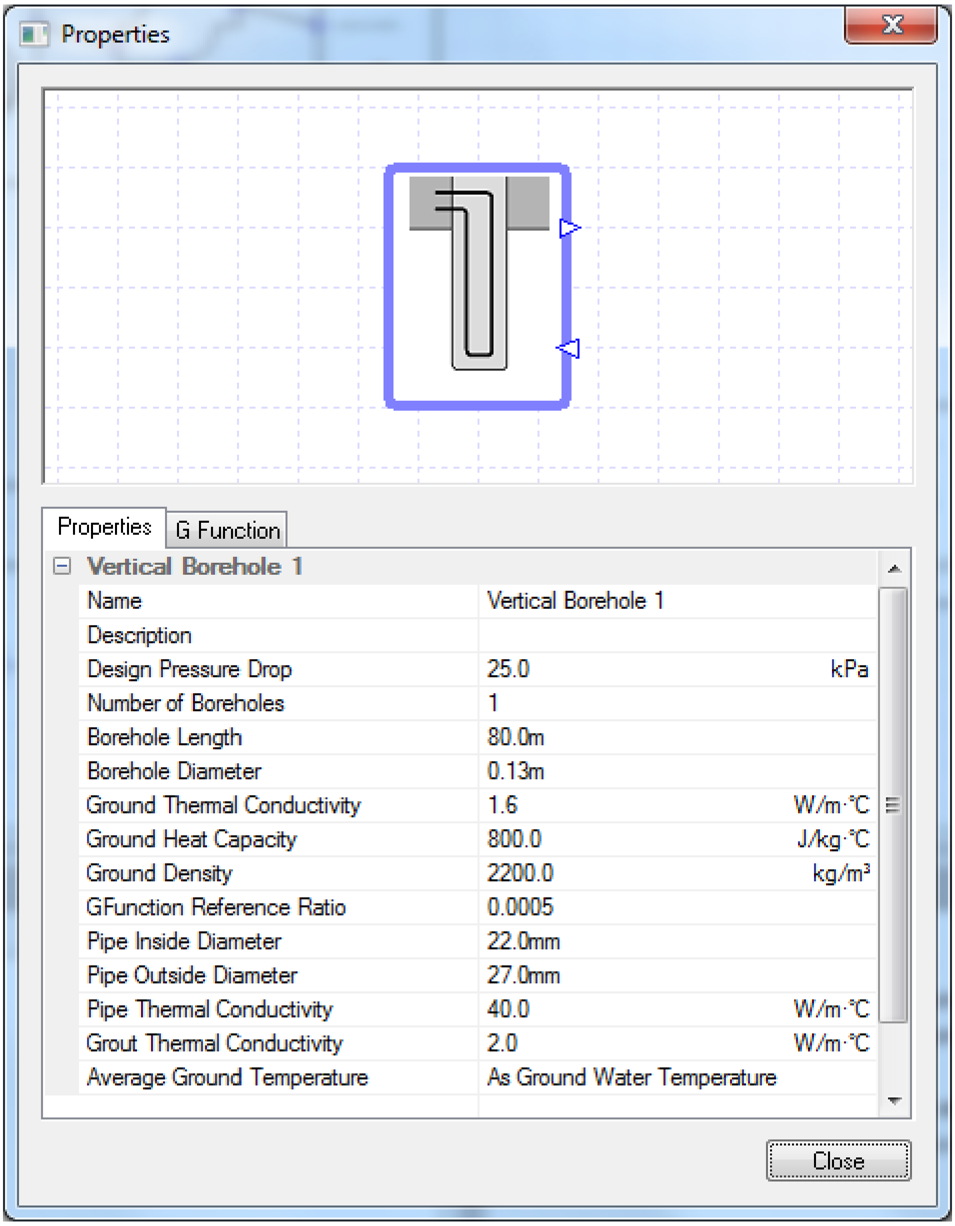
  Please note that depending on the properties of the cooling tower, it may not always be able to meet this setpoint.
- **Design Water Flow Rate** – This field is only available when the Max Air Flow Rate field is sized and sets the flow rate of the water entering the dry cooler on the design condition. There are two options to choose from when using this field:
  - Value – With this option the user must enter in the water flow rate to be used on the sizing condition.
  - As Pump Design Flow Rate – With this option the design flow rate from the pump on the heat rejection loop will be used in the design conditions.
- **Design External Drybulb –** This field is only available when the Max Air Flow Rate field is sized and sets the external drybulb temperature used on the design condition. There are three options to choose from with this field:
  - Value – Upon choosing this option, the user would enter the design external drybulb temperature as a value.
  - Peak External – Upon choosing this option, TAS will take the peak external drybulb temperature from the weather data as the design external drybulb for the sizing run.
  - Max Operating – Upon choosing this option, the user is asked to enter a limiting drybulb temperature. TAS will then compare this temperature with the peak external drybulb temperature from the weather data and use the minimum value as the design external drybulb temperature.

- **Design Range -** This field is only available when the Max Air Flow Rate field is sized and sets the water temperature drop of the fluid flowing through the dry cooler used on the design conditions.
- **Use Evaporative Precooling** – This field allows the user to determine if water is sprayed into the air entering the dry cooler to cool it down. If ticked, the following field appears:
  - Effectiveness – This option allows the user to input how effective the evaporative precooling is. The effectiveness needs to be entered as a factor between 0 and 1. A modifier can be applied to this field by going to the Effectiveness tab.
- **Allow Water Heating –** Upon ticking this box, the dry cooler will be able to heat the fluid passing through it when the fluid's temperature is below the heating setpoint of the dry cooler and the external air's drybulb temperature. As most of the time, the component will be used for dry cooling only; this option should be used occasionally. If this option is used, the user will have to enter the following information:
  - Heating Setpoint – When a temperature is entered into the Heating Setpoint field, the component will attempt to regulate the temperature of the fluid going through it by heating the fluid to reach this setpoint. With the dry cooler, if the heating setpoint is set to be higher than the external dry bulb temperature then the dry cooler will only be able to heat the fluid up to the external drybulb temperature. When the Heating Setpoint field is visible, modifiers can be added to the setpoint using the Heating Setpoint tab.
  - Minimum Heating Setpoint Delta T - When the temperature difference between the fluid the dry cooler is heating and the external drybulb temperature is very small, the dry cooler will become very inefficient at heating the fluid. To get around this problem, the user can enter a minimum heating setpoint delta T, which creates a buffer zone to prevent inefficient operation. The minimum heating setpoint delta T works in the following way. If the user enters the value x ^{o}C into this field, then when the air's drybulb temperature is within x ^{o}C of the heating setpoint of the dry cooler, or when the heating setpoint of the dry cooler is above the external drybulb temperature, TAS will set the heating setpoint to:

    $$\boldsymbol{Heating\ setpoint = Air's\ drybulb\ temperature - x.}$$

    Please note that depending on the properties of the cooling tower, it may not always be able to meet this setpoint.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the dry cooler. Any value entered here will be the associated load during any hour the heat pump operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.21 Vertical Borehole



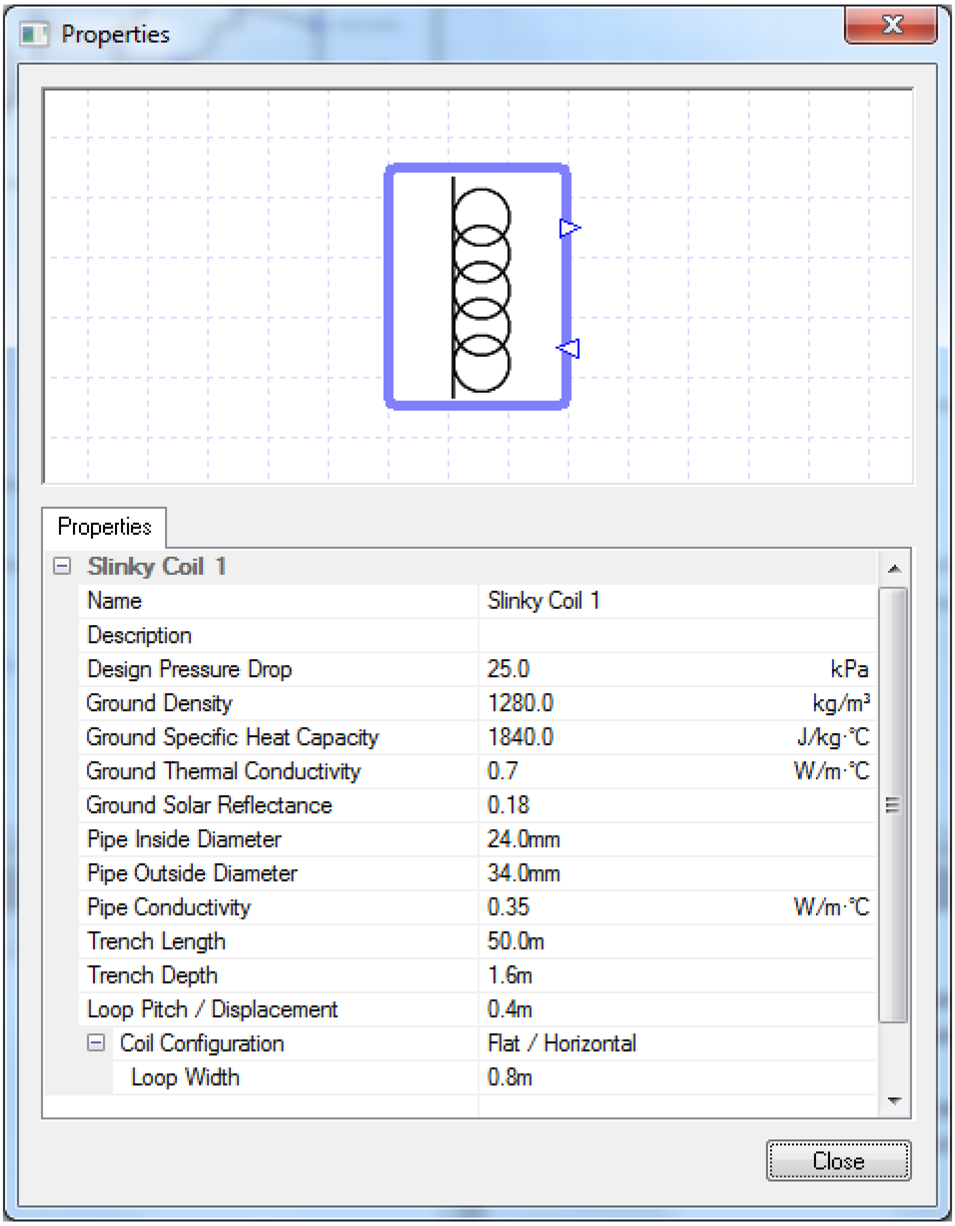
A vertical borehole is a type of ground source heat exchanger that can heat or cool the fluid going through it depending on the temperatures of the fluid and the ground. The Vertical Borehole component in TAS comes with a G-function importer, which allows the user to import a pre-defined G – function, which will dictate the thermal response of the borehole. Please note that the ground temperatures will depend on the results of the previous hours and the thermal properties entered for the ground in the Borehole's properties. Also in the results section, a positive load indicates the borehole is transferring heat out of the water to the ground while a negative load indicates the borehole is transferring heat from the ground to the water. Controllers cannot be used with this component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Number of Boreholes –** This field allows the user to enter how many boreholes they want the component to model. These boreholes will be identical and use the same G-function. If you import a G-function, this field will change depending on the G-function you import.
- **Borehole Length –** In this field the user must enter the length of the borehole from top to bottom.
- **Borehole Diameter –** In this field the user must enter the diameter of the borehole.
- **Ground Thermal Conductivity –** This field requires the user to enter the thermal conductivity of the ground surrounding the borehole. If the soil type varies in the ground around the borehole, then please take a weighted average for the thermal conductivity.
- **Ground Heat Capacity –** This field requires the user to enter the heat capacity of the ground surrounding the borehole. If the soil type varies in the ground around the borehole, then please take a weighted average for the heat capacity.
- **Ground Density –** This field requires the user to enter the density of the ground that the borehole is placed into. If the soil type is different around the borehole then please take a weighted average for the density.
- **G-Function Reference Ratio –** The user should enter here the ratio of borehole radius to borehole length. This ratio should be entered into the software as the value x, where the ratio is 1:x. This ratio is used to adjust the g-values when the geometry of the borehole isn't an exact match to the g-function
- **Pipe Inside Diameter –** Enter the internal diameter of the pipes in the vertical borehole.
- **Pipe Outside Diameter –** Enter here the external diameter of the pipes in the vertical borehole.
- **Pipe Thermal Conductivity –** In this field, please enter the thermal conductivity of the pipes running through the vertical borehole.
- **Grout Thermal Conductivity –** In this field, please enter the thermal conductivity of the grout in the vertical borehole.
- **Average Ground Temp –** In this field, the user will need to specify the average ground temperature of the ground the borehole is in. The user has two options when setting this, they can enter in the temperature as a value (useful for deep boreholes as the further down you go the warmer the ground gets) or they can set it to be the same as the ground water temperature. This value is set in the simulation dialog box.
- **G – Function Importer –** A G-function is a function which dictates the thermal response of a vertical borehole. To enter the vertical boreholes G-function into the component, the user must go to the G-Function tab. From here the user can create their own G-function for the borehole or import one from the G-Function database by pressing on the import button.

4.22 Slinky Coil



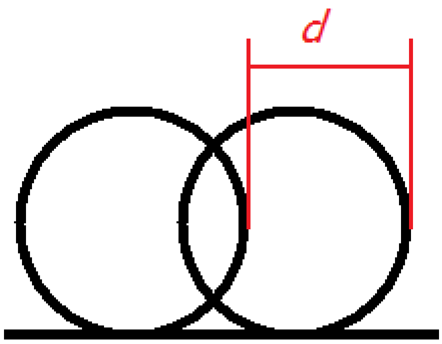
The Slinky Coil component in TAS systems allows the user to model vertical or horizontal slinky coils. The slinky coil can provide heating or cooling depending on the temperature of the fluid going through it and the temperature of the ground. Ground temperatures depend on: weather data, the thermal properties of the ground and the results of previous hours. Please note that a negative load in the results section indicates heat transfer from the ground to the fluid while a positive load indicates heat transfer from the fluid to the ground. Controllers cannot be used with this component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid when passing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this

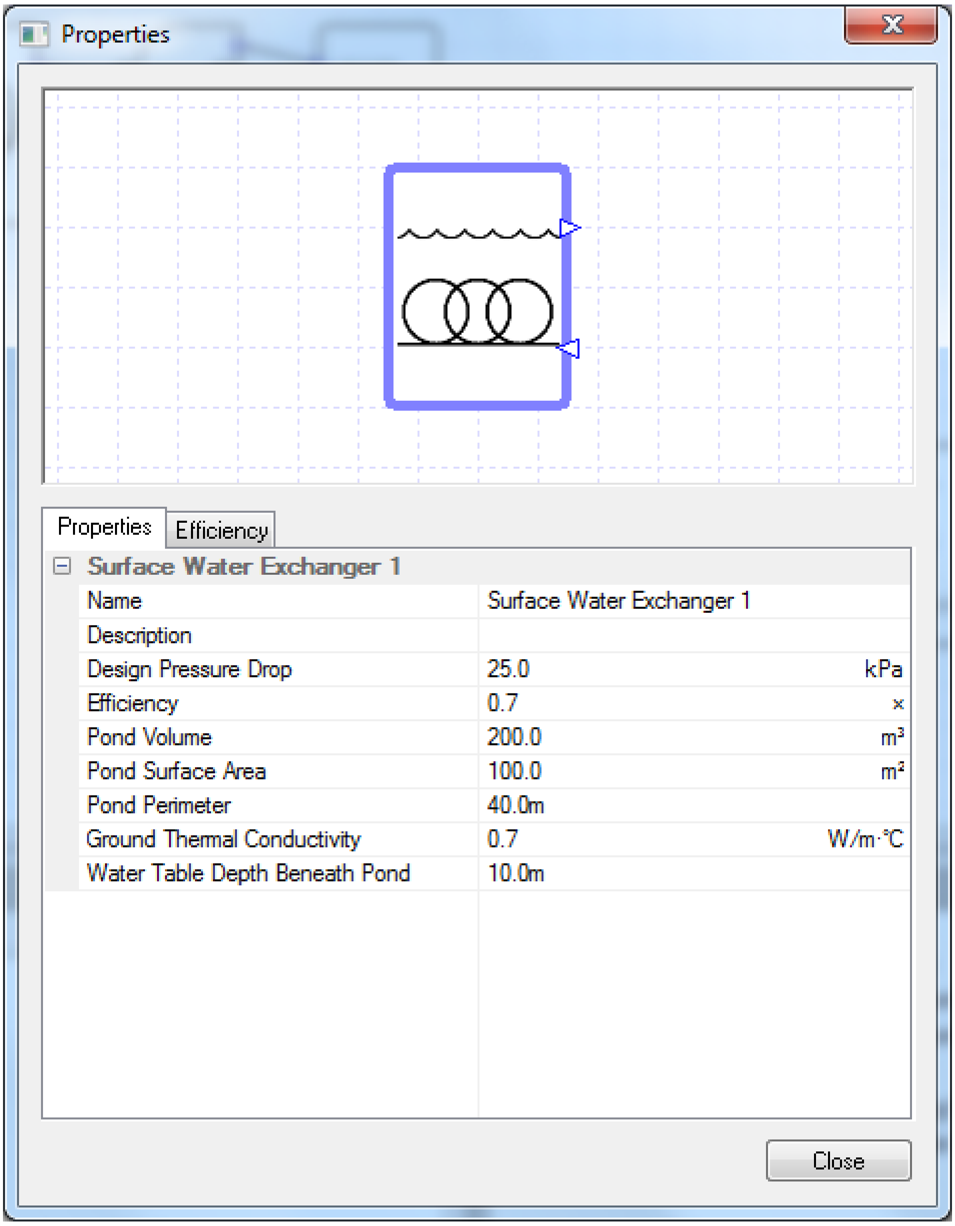
circuit. If the “Not Used” option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.

- **Ground Density –** This field requires the user to enter the density of the ground that the slinky coil is placed. If the soil type is different around the slinky coil then please take a weighted average for the density.
- **Ground Specific Heat Capacity –** This field requires the user to enter the specific heat capacity of the ground surrounding the slinky coil. If the soil type varies in the ground around the slinky coil, then please take a weighted average for the specific heat capacity.
- **Ground Thermal Conductivity –** This field requires the user to enter the thermal conductivity of the ground surrounding the slinky coil. If the soil type varies in the ground around the slinky coil, then please take a weighted average for the thermal conductivity.
- **Ground Solar Reflectance –** This field requires the user to enter in the solar reflectance of the top surface of the ground. It is used to calculate the heat gain of the soil from solar energy.
- **Pipe Inside Diameter –** Enter the internal diameter of the pipes in the slinky coil. Please note that this is for a cross section of pipe, not the diameter of a loop of pipe.
- **Pipe Outside Diameter -** Enter here the external diameter of the pipes in the slinky coil. Please note that this is for a cross section of pipe, not the diameter of a loop of pipe.
- **Pipe Thermal Conductivity –** In this field, please enter the thermal conductivity of the pipes running through the slinky coil.
- **Trench Length –** In this field, please enter the total length of the trench the slinky coil is sitting in. Please note that this is the length of the trench and not the length of the pipes.
- **Trench Depth –** In this field, please enter the depth of the trench. The depth of the trench is the distance from the underside of the slinky coil to the top surface of the ground.
- **Loop Pitch / Displacement –** The Loop Pitch / Displacement field requires the user to enter the interval between coil loops. The interval the user should enter is shown in the diagram below as d.



- **Coil Configuration –** This field allows the user to set how the Slinky Coil is placed in the ground. The user gets the following options to choose from:
    - Flat / Horizontal – This option places the coils horizontally in the ditch. Upon choosing this option the user is presented with the Loop Width field, where they should enter the diameter of the loops. If the diameter of the slinky coil loops varies with each loop, then the user should enter the average here.
    - Upright / Vertical – This option places the coils vertically in the ditch. Upon choosing this option the user is presented with the Loop Height field, where they should enter the diameter of the loops. If the diameter of the slinky coil loops varies with each loop, then the user should enter the average here.

4.23 Surface Water Exchanger



The Surface Water Exchanger component is a water source heat exchanger which can be used to provide heating and cooling depending on the water temperatures. The temperature of the body of water the exchanger is placed in depends on the results of previous hours, weather conditions and the geometry of the body of water. Please note that the surface water exchanger component should not be used to model heat transfer with moving bodies of water (for example rivers) or for deep bodies of water (as it assumes perfect temperature mixing throughout the body of water). To emphasise this point the component will refer to the body of water as a pond in the properties. Please note that controllers cannot be used with this component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid when passing through the component. The drop in pressure is caused by resistance to the

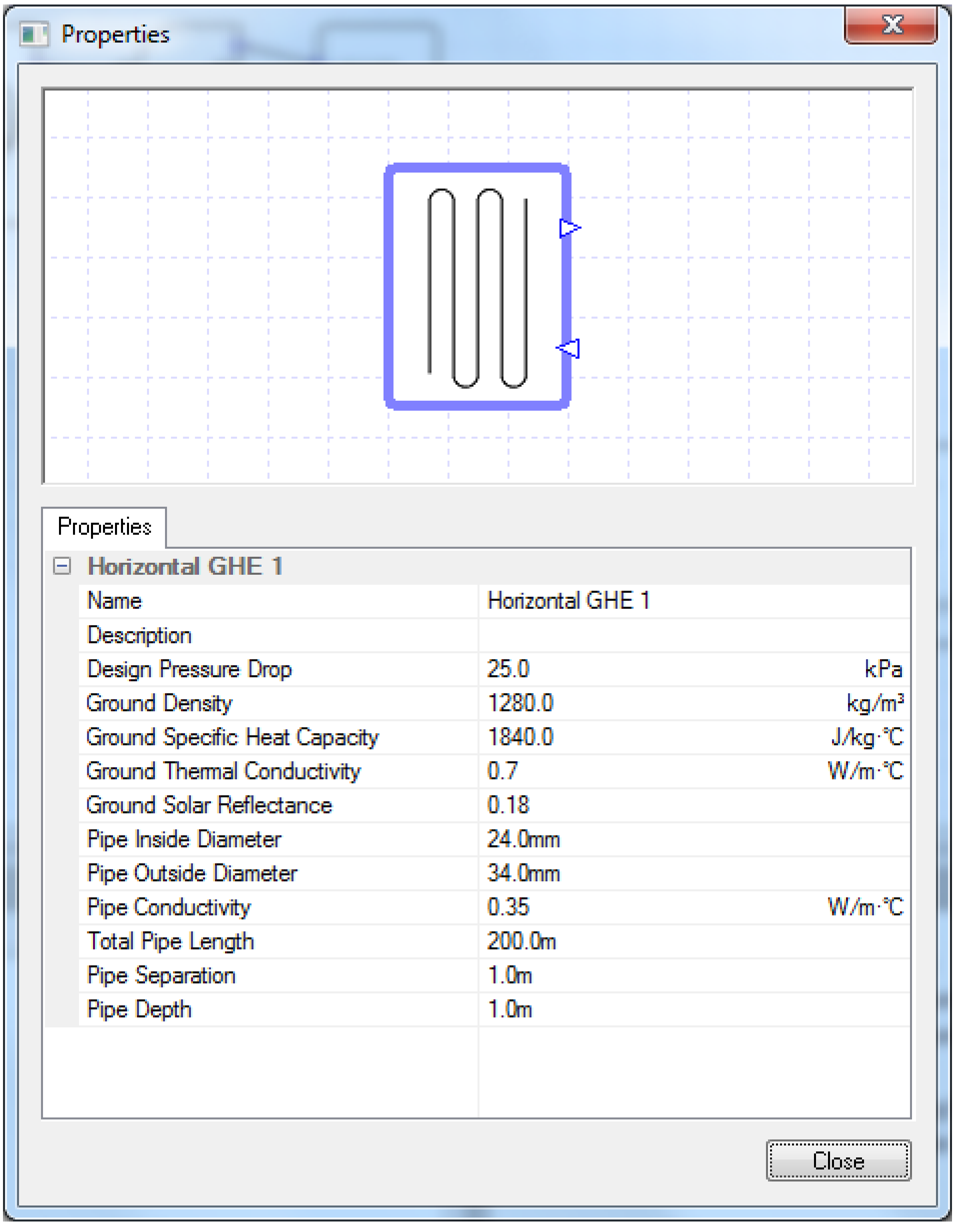
flow and the value entered will affect the amount of energy used by the pump on this circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.

- **Efficiency -**– The Efficiency field allows the user to set the efficiency of the heat transfer between the body of water and the fluid circuit. The efficiency is set by the following formula:

$$\boldsymbol{Efficiency = \frac{Pond_{return} - Pond_{Supply}}{Pond_{Mean} - Pond_{supply}}},$$

  Where: $\boldsymbol{Pond_{Return}}$ is the temperature of the fluid leaving the surface water exchanger on the fluid circuit, $\boldsymbol{Pond_{Supply}}$ is the temperature of the fluid entering the surface water exchanger on the fluid circuit and $\boldsymbol{Pond_{Mean}}$ is the mean temperature of the body of water for the hour which can be found by taking the average of the current hours pond temperature and the previous hours pond temperature.
- **Pond Volume –** In this field the user is required to enter the volume of the body of water.
- **Pond Surface Area –** In this field the user is required to enter the surface area of the top surface of the pond (i.e. the surface that is not in contact with the ground).
- **Pond Perimeter –** In this field the user is required to enter the perimeter of the body of water.
- **Ground Thermal Conductivity –** This field requires the user to enter the thermal conductivity of the ground under and around the body of water. If the soil type changes around the body of water then the user should take a weighted average for the thermal conductivity. This field, along with the Pond Perimeter and Water Table Depth Beneath Pond field will be used to calculate heat transfer between the ground surrounding the body of water and the body of water.
- **Water Table Depth Beneath Pond –** The user should enter here the depth of the water table beneath the pond. To disregard the effects of ground heat transfer, the user should have a deep water table.

4.24 Horizontal Ground Heat Exchanger



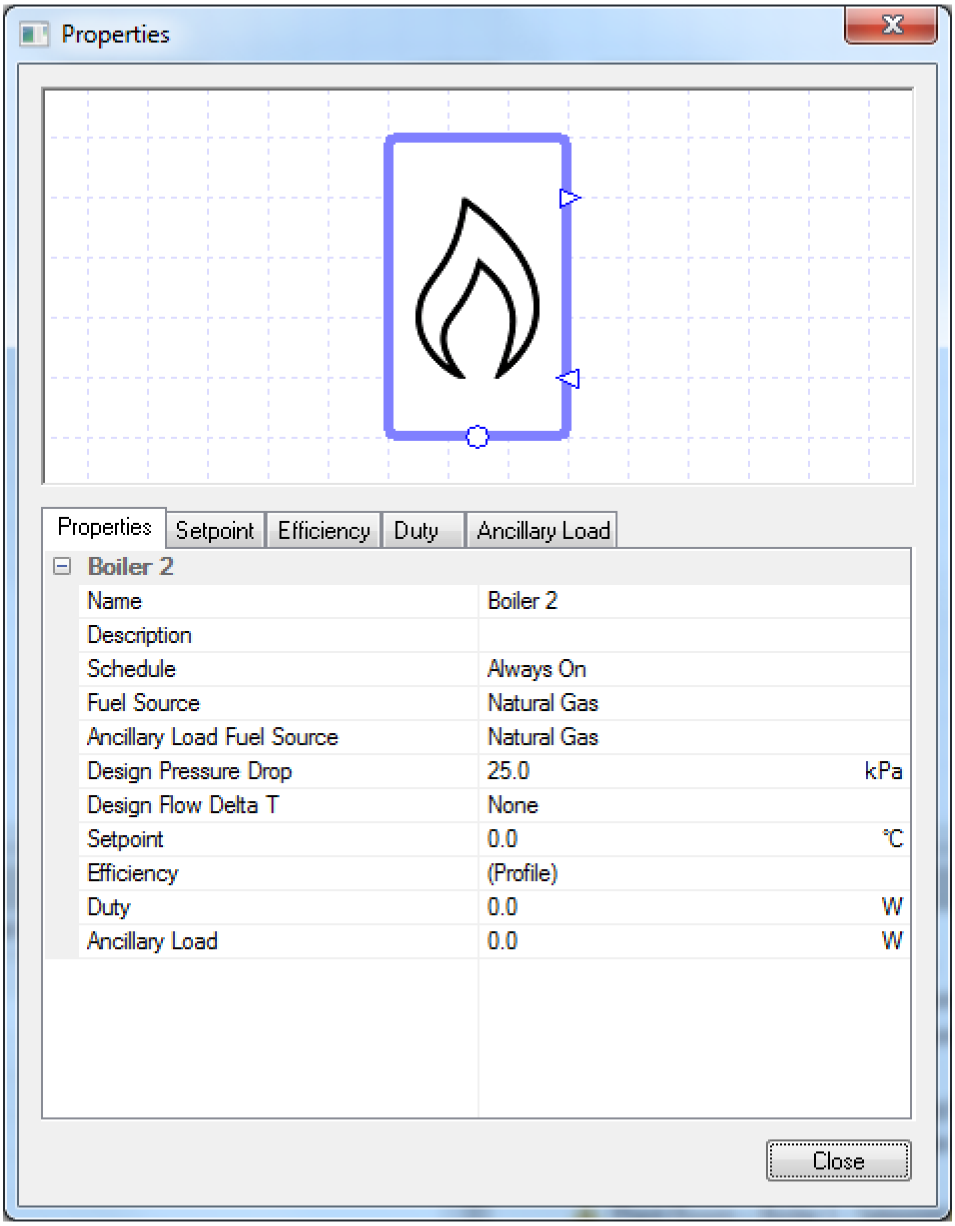
The Horizontal Ground Heat Exchanger (GHE) component in TAS systems allows the user to model a horizontal ground heat exchanger. The Horizontal GHE is tailored to model a horizontal field of parallel pipes. This tailoring means that it cannot be used to model vertical pipes. The Horizontal GHE can provide heating or cooling depending on the temperature of the fluid going through it and the temperature of the ground. Normally a horizontal GHE would connect up to a heat pump, as it doesn't have a setpoint to control the fluid temperatures. In the simulation the ground temperatures depend on: weather data, the thermal properties of the ground and the results of previous hours. Please note that a negative load in the results section indicates heat transfer from the ground to the fluid while a positive load indicates heat transfer from the fluid to the ground. Controllers cannot be used with this component.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.

- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on this circuit. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Ground Density –** This field requires the user to enter the density of the ground that the horizontal GHE is placed. If the soil type changes around the horizontal GHE then please take a weighted average for the density.
- **Ground Specific Heat Capacity –** This field requires the user to enter the specific heat capacity of the ground surrounding the horizontal GHE. If the soil type varies in the ground around the GHE, then please take a weighted average for the specific heat capacity.
- **Ground Thermal Conductivity –** This field requires the user to enter the thermal conductivity of the ground surrounding the horizontal GHE. If the soil type varies in the ground around the GHE, then please take a weighted average for the thermal conductivity.
- **Ground Solar Reflectance –** This field requires the user to enter in the solar reflectance of the top surface of the ground the horizontal GHE is placed in. It is used to calculate the heat gain of the soil from solar energy.
- **Pipe Inside Diameter –** Enter the internal diameter of the pipes in the horizontal GHE. Please note that this is for a cross section of pipe.
- **Pipe Outside Diameter -** Enter here the external diameter of the pipes in the horizontal GHE. Please note that this is for a cross section of pipe.
- **Pipe Thermal Conductivity –** In this field, please enter thermal conductivity of the pipes running through the Horizontal GHE.
- **Total Pipe Length –** In this field the user should enter the total length of the pipe running through the GHE.
- **Pipe Separation –** In this field, the user should enter the separation between the pipe as it snakes through the GHE. If the separation between the pipe varies throughout the GHE, the user should take the average separation. This separation distance will have an effect on how effective the heat transfer is. Please note that the maximum allowed separation is 2m.
- **Pipe Depth -** In this field, please enter the depth of the pipe in the horizontal GHE. The depth of the pipe is the distance from the underside of the pipe in the horizontal GHE to the top surface of the ground.

4.25 Boiler



The Boiler component in TAS allows users to model boilers in their plant room circuits. The boiler heats up the fluid passing through it to a setpoint by burning fuel. Please note that a boiler cannot provide any cooling. Boilers are normally connected up to heating and DHW collections but can be used in other circuits. You can use a controller with the boiler to control it by informing it of the amount of power the boiler should use to heat up the fluid flowing through it. The controller does this by sending a signal, between zero and one, to the boiler dictating the proportion of the boilers's duty it should use to heat up the fluid. So, for example, if the boiler received a signal of zero the boiler would not heat up the fluid flowing through it. While if it received a signal of 1 the boiler will heat up the fluid flowing through it using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after the return port of the boiler.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.

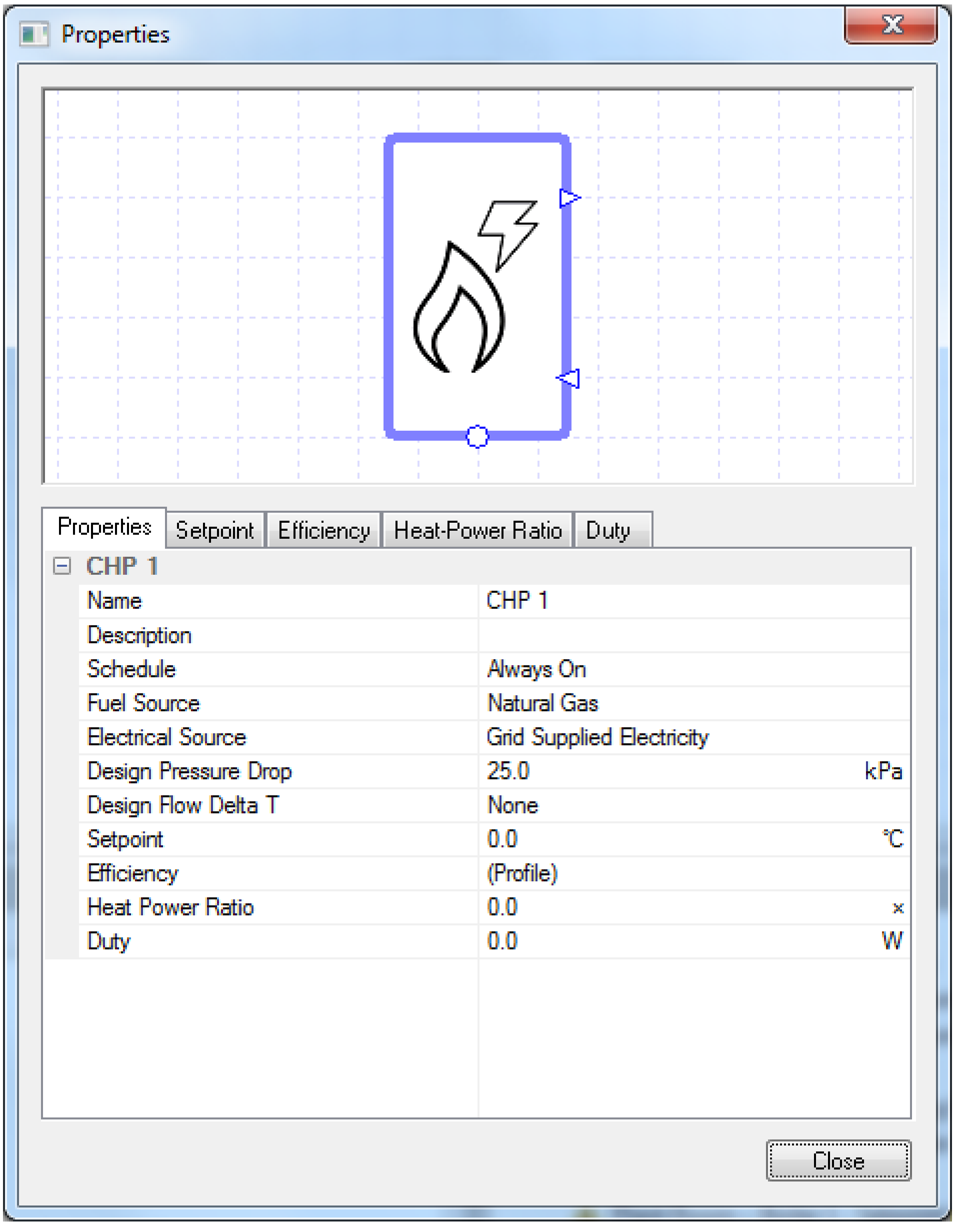
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the boiler, this will mean that the fluid will flow through the boiler unheated, even if there is a controller sending a non-zero signal to the boiler. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source -** With this field the user can choose the fuel source of the boiler. For the boiler, the fuel will be burned to produce the heat to warm the fluid up. The options provided in the drop down menu for the fuel source come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and the loads of the component will be discarded.
- **Ancillary Load Fuel Source -** This option works in the same way as the Fuel Source option but allows the user to set the fuel source for the ancillary loads of the boiler.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid when passing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the boiler’s circuit loop. If the “Not Used” option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the “None” option. If the user decides to use this option, by choosing the “Value” option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump’s Design Flow Rate field set to “Sized”. The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the boiler, it will heat the fluid so it reaches the setpoint, but it will not be able to cool down the fluid to reach this setpoint. To cool the fluid you would need another component, for instance an air source chiller. Please note that when a controller is used in conjunction with the boiler, the Setpoint field will disappear from the properties. This is done because the boiler is being controlled by a controller and will heat up the fluid when the controller sends a signal informing the boiler to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.
- **Efficiency –** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency using modifiers. By default, a partload modifier is set up on the boiler efficiency so the user can enter the boilers efficiency at different part loads. If the user wishes, they can delete the modifier and enter in a value for the efficiency of the boiler. This is equivalent to having a partload modifier with the same efficiency entered at each step.

- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a boiler, it means it wouldn't be able to heat up the fluid to the setpoint, it would fall short.). In TAS Systems, the demand met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
  - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
  - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
    - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
    - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.
    - Add load, all attached heating – TAS will size the duty of the component on the demand from all heating collections in the circuit.
    - Add load, all attached DHW – TAS will size the duty of the component on the demand from all DHW collections in the circuit.

    Please note that to size the duty the user will need to have design conditions in their systems file.
  - Value – With this option the user will type in the duty of the component.

  In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.
- **Ancillary Load –** This field allows the user to model the load used by any additional ancillary services associated with the boiler. Any value entered here will be the associated load during any hour the boiler operates, so it is advised to make use of the modifiers available with this field. Modifiers can be added to this field using the Ancillary Load tab.

4.26 Combined Heat and Power (CHP)



The CHP component in TAS systems allows users to model Combined Heat and Power (CHP) units. As the TAS Systems wizard comes with a set up CHP circuit, it is recommended that the user uses the wizard when setting up CHP's. The user will notice in the Wizard a priority option which is not present in the properties list. The priority field in the wizard sets which loop (heating or DHW) the fluid out of the CHP reaches first. CHPs normally cover the demand from heating and DHW collections but can also provide heat to absorption chillers. You can use a controller with the CHP to control it by informing it of the amount of power the CHP should use to operate. The controller does this by sending a signal, between zero and one, to the CHP dictating the proportion of the CHP's duty it should use to heat up the fluid and generate electricity. So, for example, if the CHP received a signal of zero the CHP would not heat up the fluid flowing through it and not generate any electricity. While if it received a signal of 1 the CHP will heat up the fluid flowing through it and generate electricity using the maximum amount of power allowed from the Duty field. If no controller is used, the sensor used to determine the fluid's temperature for the Setpoint field is assumed to be directly after the return port of the CHP.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Schedule –** The Schedule field allows the user to apply a schedule to their component to detail the operational hours of the component. If a schedule is applied by the user, then they should note that for all hours outside of the scheduled hours, the component will not operate. In the case of the CHP, this will mean that the fluid flowing through the CHP will remain unheated and no power will be generated. The default schedule option is always on, meaning that the component will operate 24/7.
- **Fuel Source -** With this field the user can choose the fuel source of the CHP. This fuel source will be the fuel the CHP unit burns to provide the heat and power. The options provided in the drop down menu for the fuel source come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.
- **Electrical Source –** In this field the user should assign a "fuel source" to assign to the electricity produced from the CHP. The reason for assigning a fuel source here is to set the tariff (for the cost report, normally the tariff would be negative for electricity you generate so the cost report will subtract it from the grid supplied electricity cost) and for the CO_2 report (As generated electricity may have a different CO_2 factor than electricity from the grid). Normally the user would choose Grid Displaced Electricity for this field from the default fuel options from the library.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid when passing through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the CHP's circuit loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Setpoint -** When a temperature is entered into the Setpoint field, the component will attempt to regulate the temperature of the fluid going through it to reach the setpoint. In the case of the CHP, it will heat the fluid so it reaches the setpoint, but it will not be able to cool down the fluid to reach this setpoint. To cool the fluid you would need another component, for instance an air source chiller. Please note that when a controller is used in conjunction with the CHP, the Setpoint field will disappear from the properties. This is done because the CHP is being controlled by a controller and will heat up the fluid when the

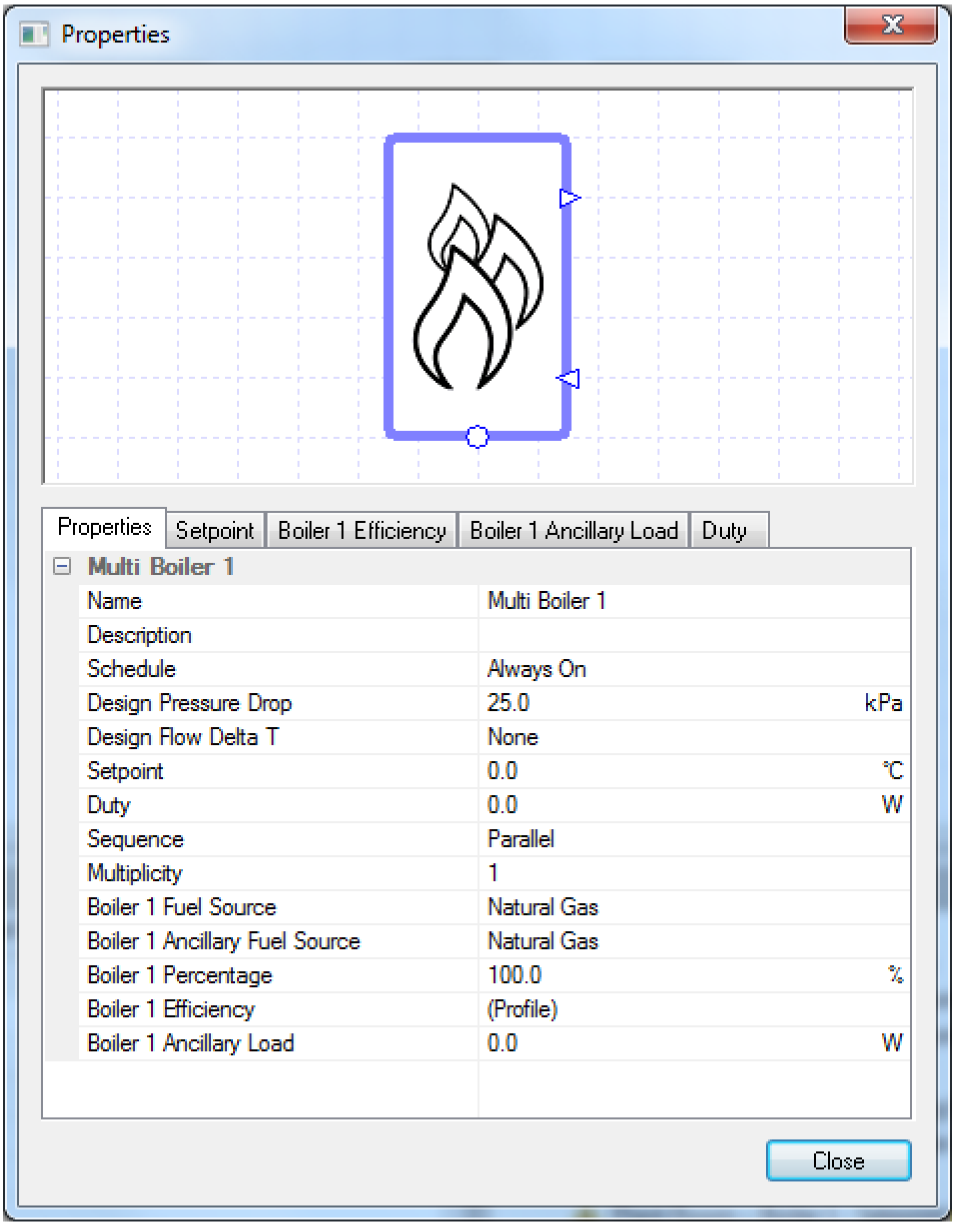
controller sends a signal informing the CHP to do so. When the Setpoint field is visible, modifiers can be added to the setpoint using the Setpoint tab.

- **Efficiency –** Upon clicking on the Efficiency field, you will be transferred over to the Efficiency tab where you can create a profile for the efficiency using modifiers. By default, a partload modifier is set up on the CHP efficiency so the user can enter the CHP's efficiency at different part loads. If the user wishes, they can delete the modifier and enter in a value for the efficiency of the CHP. This is equivalent to having a partload modifier with the same efficiency entered at partload. The user should note that the efficiency entered for the CHP is the combined heat and power efficiency.
- **Heat to Power Ratio -** This ratio is used to set the power generated by the CHP based on the heating demand. The ratio used is heating demand: power generated and the value entered in this field is the heating demand side of the ratio when the power generated side is one (So if the ratio was 3:1, the user would enter a value of 3 in this field). Please note that the user can add a modifier to this field using the Heat-Power Ratio tab.
- **Duty –** The duty of a component is the upper limit on the amount of power a component can provide. If, in a certain hour, the power demand on the component is greater than the duty of the component, the component will not be able to meet this demand (For a CHP, it means it wouldn't be able to heat up the fluid to the setpoint, it would fall short.). In TAS Systems, the demand (or load) met by a component is reported for each hour in the results section. Currently, there are 3 options for setting the duty:
    - Unlimited – Unlimited means the component is always able to meet the demand. Please note that this option cannot be used when a controller is attached to the component.
    - Sized – Allows the user to size the duty on a design condition. The user will also be asked for a size fraction and a method to size on. With the method option, you get to choose from the following options:
        - Add load, all attached – TAS Systems will size the duty of the component on the demand from all attached collections in the circuit.
        - Add load, local – TAS systems will size the duty of the component on the demand from all collections on the same loop as the component.
        - Add load, all attached heating – TAS will size the duty of the component on the demand from all heating collections in the circuit.
        - Add load, all attached DHW – TAS will size the duty of the component on the demand from all DHW collections in the circuit.
        - Add load, all attached Chillers – TAS will size the duty of the component on the demand from all Chillers on the circuit. This option could be used with absorption chillers, allowing the CHP to provide the heat necessary to drive the absorption chiller's cooling system.

      Please note that to size the duty the user will need to have design conditions in their systems file.
    - Value – With this option the user will type in the duty of the component.

In the duty tab, you will be able to choose these 3 options as well, but with the sized and value options you will be able to add a modifier.

4.27 MultiBoiler



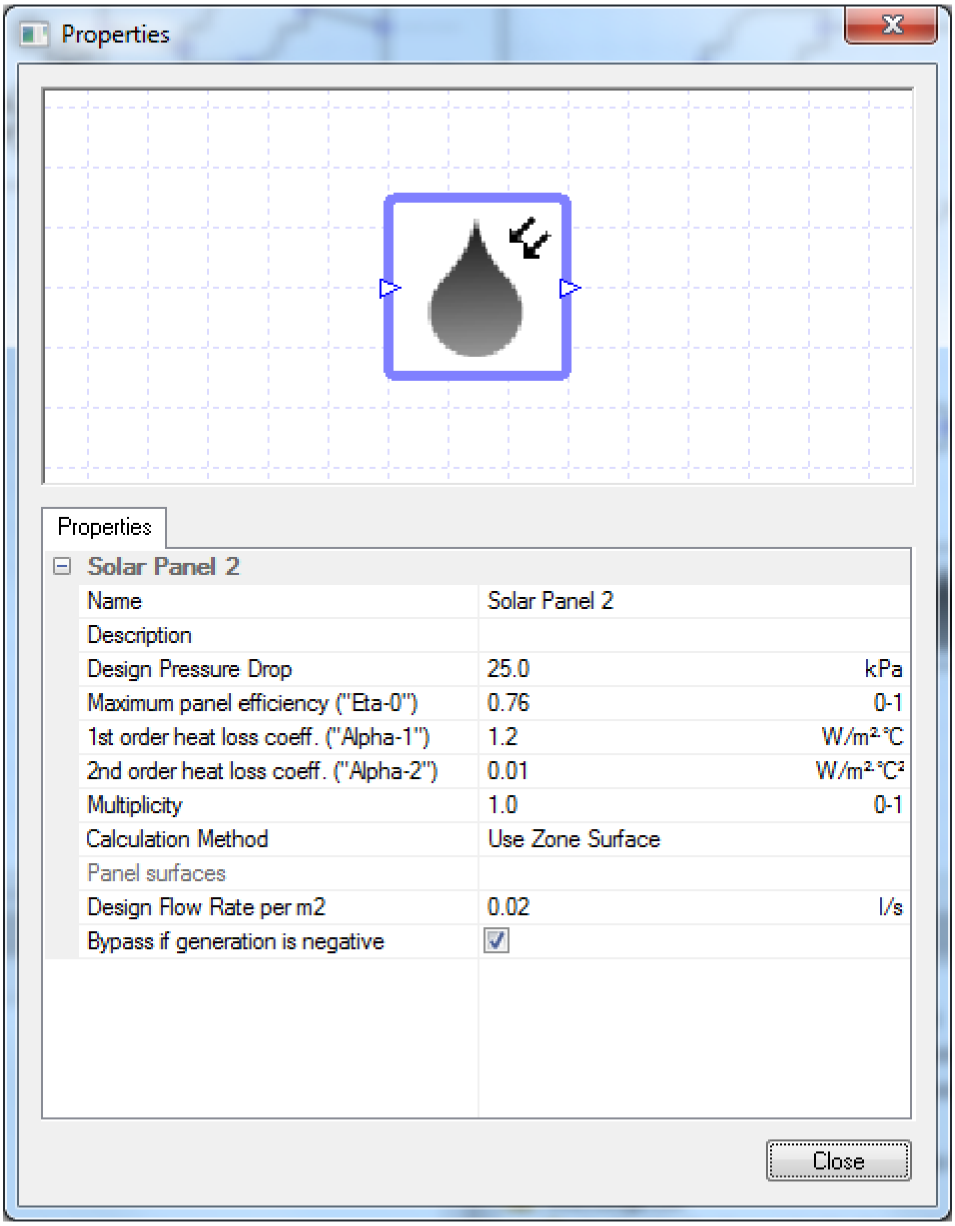
The Multi Boiler component allows the user to model multiple boilers using only one component. Please note that any field beginning with "Boiler x …", where x is a positive integer, is unique to the xth boiler. The other fields (apart from: duty, sequence and multiplicity) are the same for each boiler. Please note that the duty is the amount of demand all the boilers combined could meet, each boilers individual duty is set by the "boiler x percentage" field. So if the Multi Boiler is modelling 3 boilers, then each boiler will have the same setpoint while their fuel sources will be dependent on the field "Boiler x fuel source ". As boilers have been discussed previously, only the new properties will be discussed.

New Properties:

- **Multiplicity –** The number in the multiplicity field represents how many boilers the Multi boiler is representing.
- **Boiler x Percentage –** This option allows the user to set the duty of the xth boiler. The user will enter a percentage in this field which will set the boilers duty as this percentage of the total duty in the duty field. Please note that the sum of all the "Boiler x Percentage" fields must add up to 100%. Not doing so will give you a warning and incorrect results.

- **Sequence –** The Sequence field tells the component how the multiple Boilers will be set up. The user has three options to choose from:
    - Parallel – The load is split up between the boilers. The load will be split evenly until one of the boilers reaches its duty.
    - Serial – The first boiler covers the load until it reaches its duty. At that point the second boiler would start to cover the load and so on.
    - Staged – With a staged setup only the first boiler runs at part load. As the first boiler normally has a bigger duty than the other chiller; once one of the other boilers can run at full load the load will transition to that boiler.

4.28 Solar Panel



The Solar Panel component in TAS Systems allows the user to model solar hot water panels, which convert the solar energy absorbed by the panel to heat which warms up the fluid passing through the solar panel. Please note that the user can choose whether to use a surface from their model or create a surface by specifying the values directly into the properties of the component. Controllers cannot be used with this component.

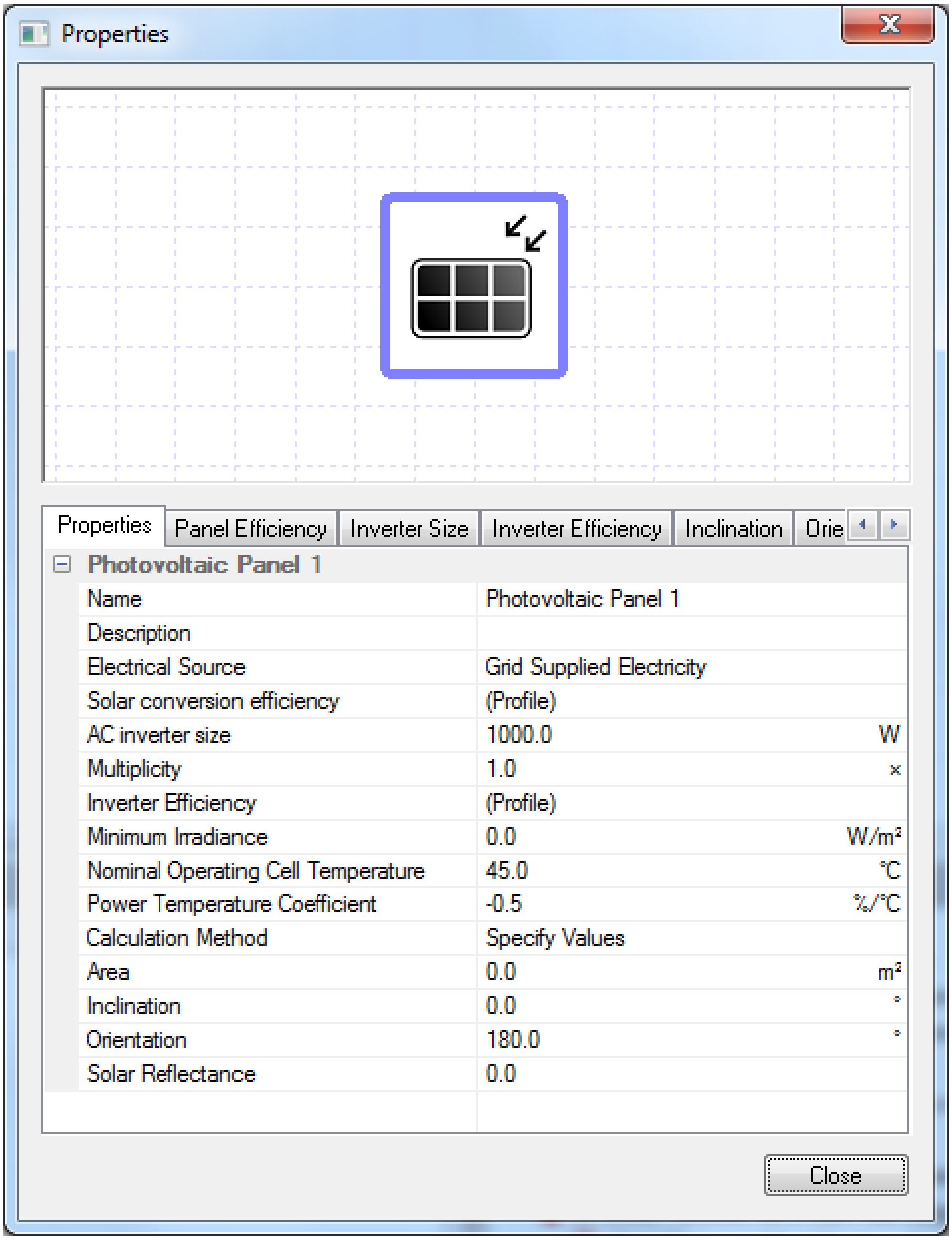
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the component. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the Solar Panel's circuit loop. If the "Not Used" option is selected for this field, the user will be asked to enter in a Capacity for the fluid loop through this component.

- **Maximum Panel Efficiency ("Eta -0") –** In this field the user will need to input the Eta- 0 value of their solar panel. The Eta – 0 value can be obtained from the solar panel's manufacturer and it details the efficiency of panel at transferring solar energy into heat.
- **1st order heat loss coefficient (Alpha – 1) –** In this field the user will need to input the Alpha – 1 value of their solar panel. The Alpha -1 value can be obtained from the solar panel's manufacturer and it details the first order heat loss from the solar panel.
- **2nd order heat loss coefficient (Alpha – 2) –** In this field the user will need to input the Alpha – 2 value of their solar panel. The Alpha -2 value can be obtained from the solar panel's manufacturer and it details the second order heat loss from the solar panel.
- **Multiplicity –** The Multiplicity field allows the user to enter in a factor which is then multiplied by the surface area of the panel so that the component can model a smaller or larger array of panels. This will be especially useful when using the component with a surface from a panel drawn in the 3D modeller, as this field allows you to modify the surface area of the panel from Systems rather than going back to the 3D modeller and making the changes there.
- **Calculation Method –** This option allows the user to decide if they wish to calculate the dimensions of their solar panel using a surface from the TBD or if they wish to set the properties of the solar panel within TAS Systems. If the user chooses to use the "Use zone surface" option they will need to drag a zone, which has a surface that has the solar panel box ticked, onto the component. TAS Systems will then use this surface to calculate the results. The user can also choose the "Specify Values" option, where they will need to fill in the following information:
    - Area – Enter the area of the panel.
    - Inclination – Enter the angle the panel is tilted at in degrees.
    - Orientation – Here the user has two options. If they choose the "Value" option, the user is required to enter the orientation the panel is facing in degrees where: 0 degrees is north, 90 degrees is East etc. The user can also choose the "Sun Tracking" option; with this option the panel will move its orientation to follow the sun's path during the day.
    - Solar Reflectance – In this field the user should enter the solar reflectance of the panel's surface as a factor between zero and one.

  Please note that when using the "Specify Values" option, the solar gain on the panel due to solar energy being reflected from the surrounding ground is not included in the solar gain figures. To include this, the user will have to model the panel in the 3d modeller.
- **Design Flow Rate per m^2 –** In this field the user enters the design flow rate of the fluid they require per m^2 (per ft^2 in U.S Customary units). If the pump on the circuit loop is set to sizing, it will set the design flow rate of the circuit loop by multiplying this field by the area of the panel.
- **Bypass if Generation is negative –** Upon ticking this field, the fluid will bypass the solar panel in the hours where there is not enough solar energy to warm the fluid up to counter the heat loss it incurs by passing through the panel.

4.29 PV Panel



The PV panel component in TAS systems allows the user to model PV panels. The PV panels do not connect up to any plant room circuit as they are only used to calculate the amount of electric they will generate. Please note that user can choose whether to use a surface from their model or create a surface by specifying the values directly into the properties of the component. Controllers cannot be used with this component.

Properties:

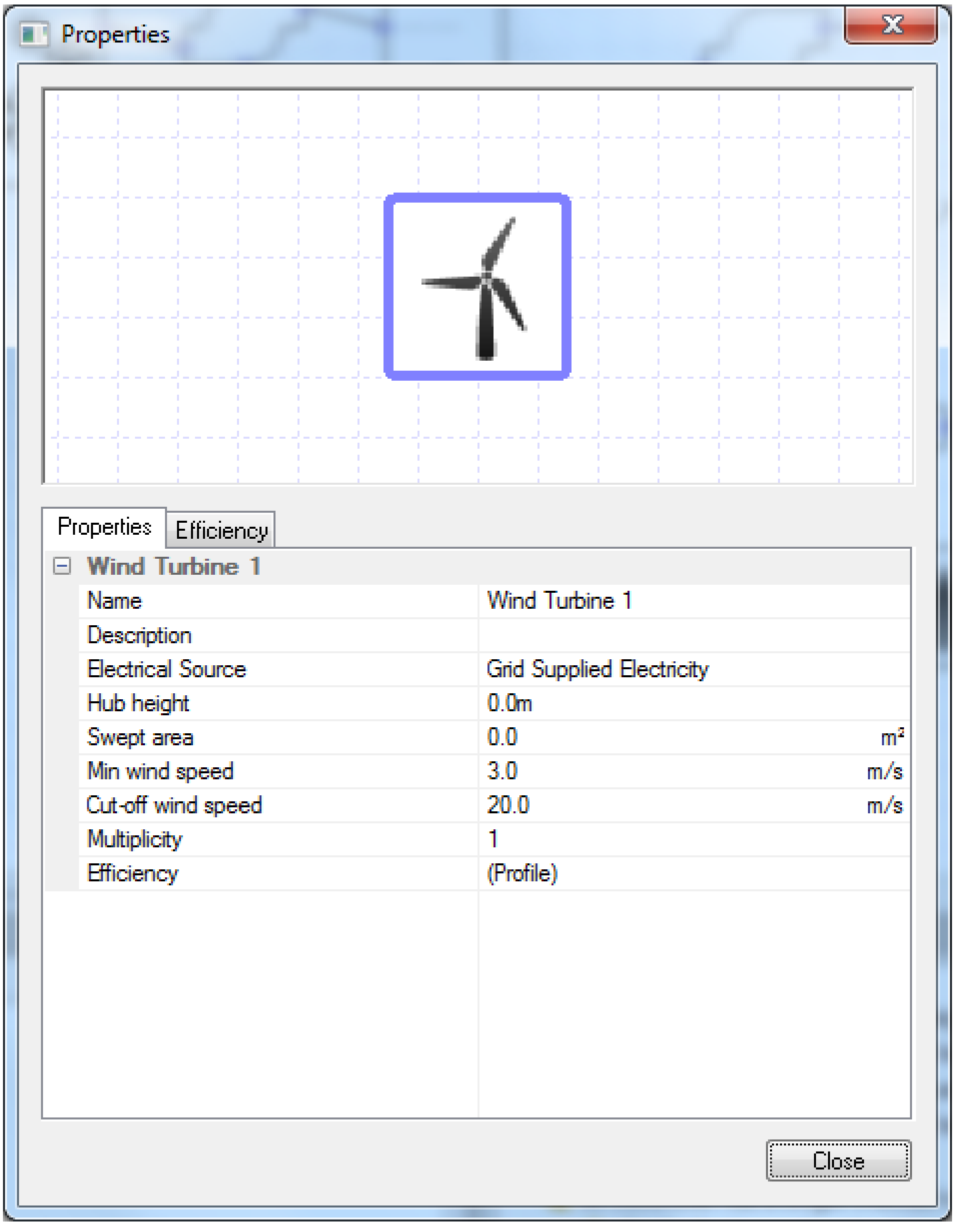
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Electrical Source –** In this field the user should assign a "fuel source" which the electricity generated from the panel will be assigned to. The reason for assigning a fuel source here is to set the tariff (for the cost report, normally the tariff would be negative for electricity you generate so the cost report will subtract it from the grid supplied electricity cost) and for the CO_2 report (As generated electricity may have a different CO_2 factor than electricity from the

grid). Normally the user would choose Grid Displaced Electricity for this field from the default options.

- **Solar Conversion Efficiency –** In this field the user should enter how efficient the PV panel is at converting the solar energy into electricity. By default, a profile based on Irradiance has been set up in the Panel efficiency tab. The user can modify this profile, create a new one or delete the profile and just enter a value for the efficiency in per cent. If a value is used, this efficiency is used for every hour of operation for the PV panel.
- **AC Inverter Size –** This field allows the user to set the size of the AC inverter. This sets the upper limit of how much electricity can be generated in one hour.
- **Multiplicity –** The Multiplicity field allows the user to enter in a factor which is then multiplied by the surface area of the panel so that the component can model a smaller or larger array of panels. This will be especially useful when using the component with a surface from a panel drawn in the 3D modeller, as this field allows you to modify the surface area of the panel from Systems rather than going back to the 3d modeller and making the changes there.
- **Inverter Efficiency –** The Inverter Efficiency field allows the user to input how efficient the inverter is at converting solar energy into electricity. By default a partload profile modifier is set to this field detailing the inverter's efficiency at different part loads. The user can edit or modify the modifiers or delete it and just enter a percentage value.
- **Minimum Irradiance –** This field set's a minimum irradiance level that must be met for the PV panel to generate electricity. If the irradiance in any hour is below this value, then no electricity will be generated.
- **Nominal Operating Cell Temperature –** This field requires the user to enter the panel's temperature when the efficiency was measured.
- **Power Temperature Coefficient –** This field requires the user to enter the power temperature coefficient of the panel. This is used to account for the decreased efficiency of the panel when it goes above the nominal operating cell temperature.
- **Calculation Method –** This option allows the user to decide if they wish to calculate the dimensions of their PV panel using a surface from the tbd or to set the properties of the PV panel within systems. If the user chooses to use the "Use zone surface" option, the user will need to drag a zone, which has a surface that has the PV panel box ticked, onto the component. TAS Systems will then use this surface to calculate the results. The user can also choose the "Specify Values" option, where they will need to fill in the following information:
    - Area – Enter the area of the panel.
    - Inclination – Enter the angle the panel is tilted at in degrees.
    - Orientation – Here the user has two options. If they choose the "Value", the user is required to enter the orientation the panel is facing in degrees where: 0 degrees is north, 90 degrees is East etc. The user can also choose the "Sun tracking" option; with this option the panel will move its orientation to follow the sun's path during the day.
    - Solar Reflectance – In this field the user should enter the solar reflectance of the panel's surface as a factor between zero and one.

  Please note that when using the "Specify Values" option, the solar gain on the panel due to solar energy being reflected from the surrounding ground is not included in the solar gain figures. To include this, the user will have to model the panel in the 3d modeller.

4.30 Wind Turbine



The Wind Turbine component in TAS systems allows the user to model vertical axis and horizontal axis wind turbines. The wind turbine component does not connect up to any plant room circuits as they are only used to calculate the amount of electric they will generate. As the turbine is only used to generate electricity and do not link with any other component, controllers cannot be used with them.

Properties:

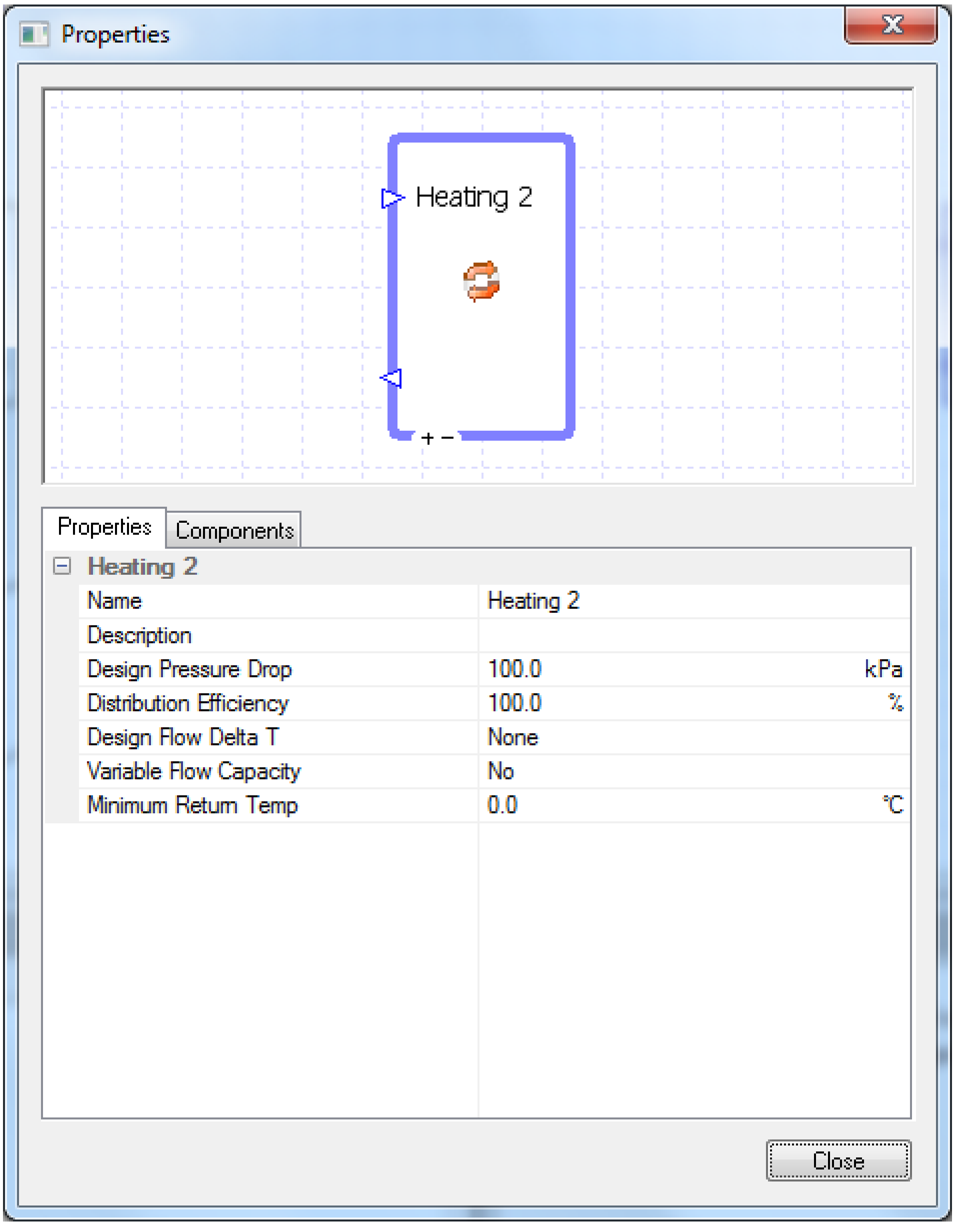
- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Electrical Source –** In this field the user should assign a "fuel source" which the electricity generated from the turbine will be assigned to. The reason for assigning a fuel source here is to set the tariff (for the cost report, normally the tariff would be negative for electricity you generate so the cost report will subtract it from the grid supplied electricity cost) and for the CO_2 report (As generated electricity may have a different CO_2 factor than electricity from the

grid). Normally the user would choose Grid Displaced Electricity for this field from the default options.

- **Hub Height –** This field requires the user to enter the height of the turbine's hub. The height should be measured from the ground to the hub. Please note that a higher hub will produce more electricity than a wind turbine with the same properties except from the hub being nearer the ground.
- **Swept Area –** This field requires the user to enter the area the blades sweep out during a rotation. The bigger the area they sweep out, the more electricity the turbine produces. For a vertical axis wind turbine the surface area is the side curved surface of the cylinder swept out by the blades.
- **Minimum Wind Speed –** This field requires the user to enter the minimum wind speed required for the turbines to produce electricity. Please note that if the wind speed is below this value for an hour then no electricity will be produced.
- **Cut – Off Wind Speed –** This field requires the user to enter the maximum wind speed that the turbine will operate in. If the wind speed goes above this cut off value in an hour, then the turbine will not produce any electricity in this hour.
- **Multiplicity -** Allows the user to set how many wind turbines the component is modelling. Please note that each wind turbine will have identical properties matching the values entered into the component.
- **Efficiency** - The Efficiency field allows the user to input how efficient the wind turbine is at converting wind power into electricity. By default a wind speed profile modifier is set to this field detailing the turbine's efficiency at different wind speeds. While the default modifier is based on wind speed, the user can edit to be based on a few different options, including wind direction. If they wish, the user can also delete the modifier and just enter a value into this field; this value will always be the efficiency of the wind turbine.

5 Collections

5.1 Heating Collection



A Heating Collection groups together a selection of heating coils / heating components that share a common source of heat. To add an air-side component to a collection requires the user to select the collection in the heating collection field of the air-side component's properties. While sensors from a controller can be attached to the collection, a controller cannot be used to control any of the properties of a collection.

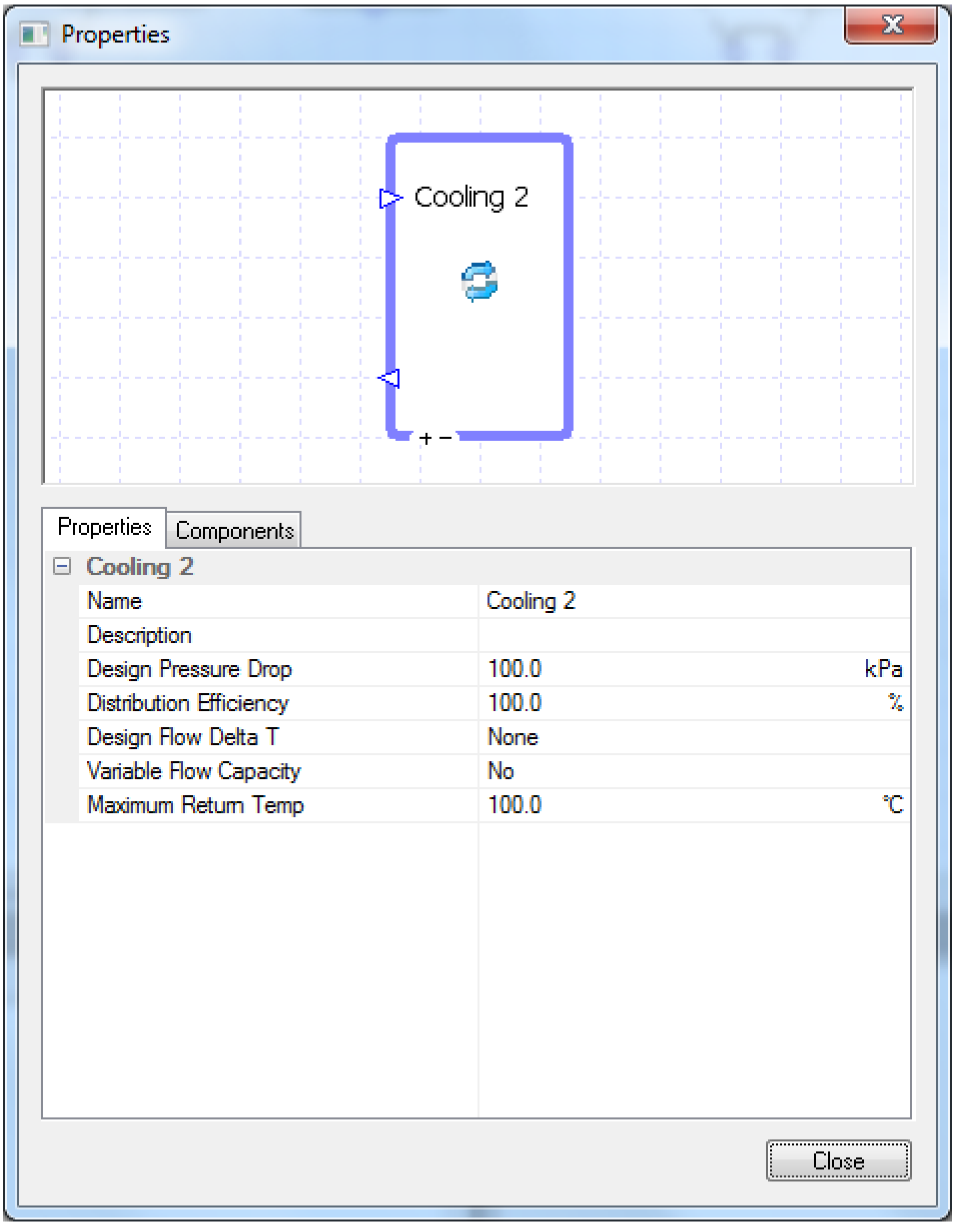
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the design pressure drop of the fluid through the collection. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the collections circuit

loop. If the "Not Used" option is selected then the user will need to enter in the capacity of the collection.

- **Distribution Efficiency –** This field requires the user to input how efficient the heating collection is at distributing the heat to the air-side components. Please note that in the results section, the demand reported by the heating collection is always the total demand from all the components in the collection summed together. The demand with the effects of the Distribution Efficiency included can be seen at boilers / CHP units on your circuit.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Variable Flow Capacity –** The Variable Flow Capacity option allows the user to decide if the fluid loop the collection is on will have a variable flow or a constant flow. If the user chooses yes the flow rate on the loop will vary while if the user chooses no then the flow rate will be constant.
- **Minimum Return Temperature –** The minimum return temperature sets a limit on the temperature of the fluid exiting the collection. This field will not impact the flow rate of the circuit but if it is set to high with respect to the flow rate you will receive warnings informing you that the demand is greater than the supply. An example of this field being set to high with respect to the flow rate can be easily seen when using the Design Flow Delta T field. Upon entering a Design Flow Delta T of $\boldsymbol{y}$ where the collection is being provided fluid at a temperature of $\boldsymbol{x}$ then the flow rate will be sized so that the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ and also so that the demand is met. If the minimum return temperature is set in between this range, then any heat source will not be able to meet the demand of the collection due to the chosen flow rate and the limit on the return temperature. If you do not wish to set a minimum return temperature then select the "None" option for this field.
- **Components Tab –** This tab allows the user to view all of the components that are part of this collection. The user can reorder this list if they wish, please note that this will have no impact on the results.

5.2 Cooling Collection



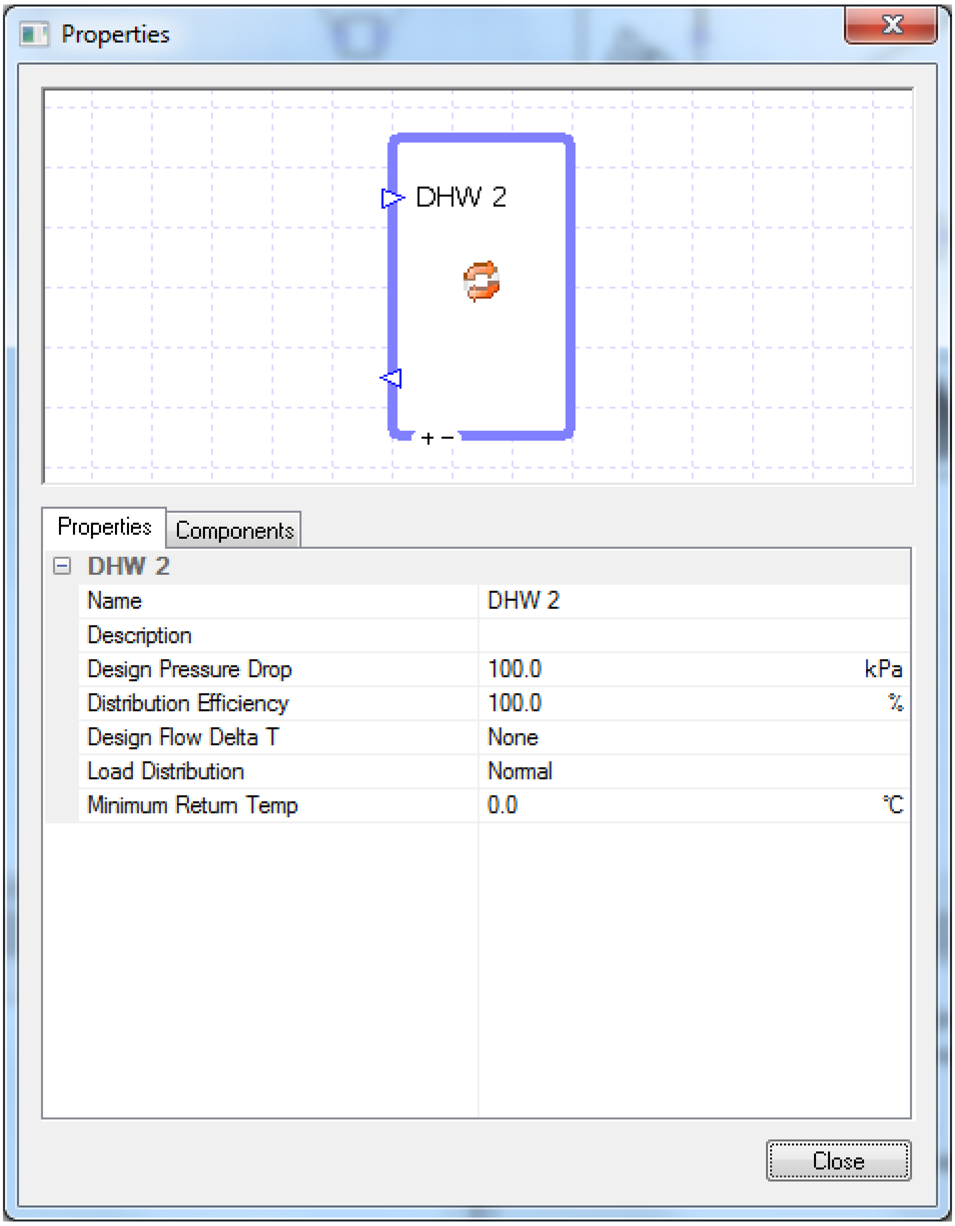
A Cooling Collection groups together a selection of cooling coils / cooling components that share a common energy source. To add an air-side component to a collection requires the user to select the collection in the cooling collection field of the air-side component's properties. While sensors from a controller can be attached to the collection, a controller cannot be used to control any of the properties of a collection.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the collection. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the collections circuit loop. If the "Not Used" option is selected then the user will need to enter in the capacity of the collection.

- **Distribution Efficiency –** This field requires the user to input how efficient the cooling collection is at distributing the cooled fluid to the air-side components. Please note that in the results section, the demand reported by the cooling collection is always the total demand from all the components in the collection summed together. The demand with the effects of the distribution efficiency included can be seen at chillers on your circuit.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x} - \boldsymbol{y}$ to $\boldsymbol{x} + \boldsymbol{y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Variable Flow Capacity –** The Variable Flow Capacity option allows the user to decide if the fluid loop the collection is on will have a variable flow or a constant flow. If the user chooses "Yes" the flow rate on the loop will vary while if the user chooses "No" then the flow rate will be constant.
- **Maximum Return Temperature –** The maximum return temperature sets an upper limit on the temperature of the fluid exiting the collection. This field will not impact the flow rate of the circuit but if it is set to low with respect to the flow rate you will receive warnings informing you that the demand is greater than the supply. An example of this field being set to low with respect to the flow rate can be easily seen when using the Design Flow Delta T field. Upon entering a Design Flow Delta T of $\boldsymbol{y}$ where the collection is being provided fluid at a temperature of $\boldsymbol{x}$ then the flow rate will be sized so that the fluid will always be kept within the temperature range $\boldsymbol{x} - \boldsymbol{y}$ to $\boldsymbol{x} + \boldsymbol{y}$ and also so that the demand is met. If the maximum return temperature is set in between this range, then any chiller may not be able to meet the demand of the collection due to the chosen flow rate and the limit on the return temperature. If you do not wish to set a minimum return temperature then select the "None" option for this field.
- **Components Tab –** This tab allows the user to view all of the components that are part of this collection. The user can reorder this list if they wish, please note that this will have no impact on the results.

5.3 DHW Collection



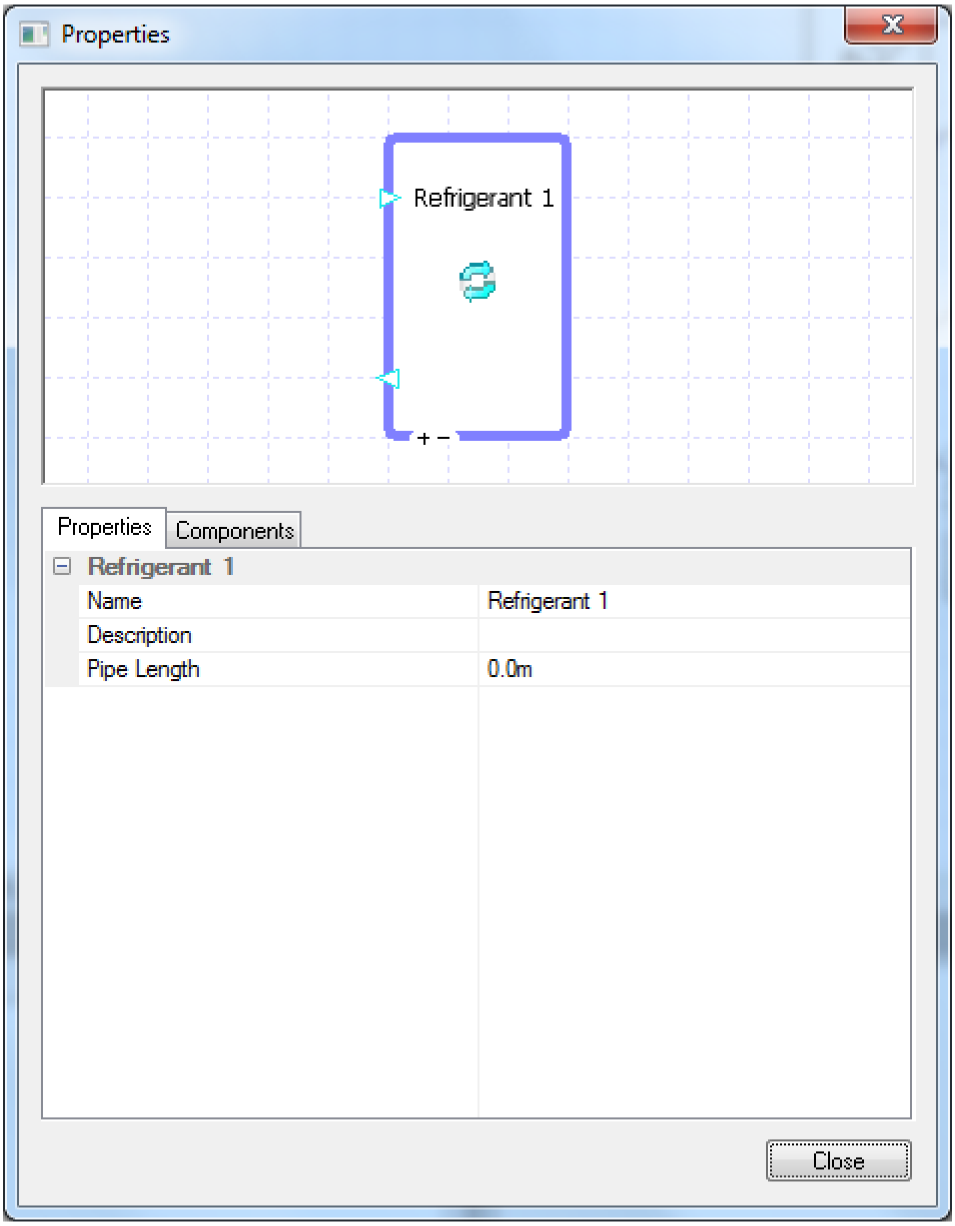
A DHW Collection groups together a selection of zones that share a common source for their DHW. To add a zone component to a collection requires the user to select the collection in the DHW collection field of the zones component's properties. While sensors from a controller can be attached to the collection, a controller cannot be used to control any of the properties of a collection.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Design Pressure Drop –** This field allows the user to enter the pressure drop of the fluid through the collection. The drop in pressure is caused by resistance to the flow and the value entered will affect the amount of energy used by the pump on the collections circuit loop. If the "Not Used" option is selected then the user will need to enter in the capacity of the collection.

- **Distribution Efficiency –** This field requires the user to input how efficient the DHW collection is at distributing the hot water to the zones. Please note that in the results section, the demand reported by the DHW collection is always the total demand from all the zones in the collection summed together. The demand with the effects of the distribution efficiency included can be seen at boilers / CHP units on your circuit.
- **Design Flow Delta T –** The Design Flow Delta T field allows the user to size the flow rate of the circuit loop the component is on such that the fluid flowing through the component is kept within a certain temperature band. The user can choose not to size the flow rate using this band, by choosing the "None" option. If the user decides to use this option, by choosing the "Value" option, they will need to make sure that they have a pump on the same circuit loop as the component with the pump's Design Flow Rate field set to "Sized". The value entered into the Design Flow Delta T field of the chiller affects the flow rate of the circuit loop in the following way. If the component has a setpoint of $\boldsymbol{x}$ and a Design Flow Delta T of $\boldsymbol{y}$, then the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ when flowing through the component by ensuring the design flow rate of the circuit is sized high enough. If multiple components / collections on the same circuit have a Design Flow Delta T value entered, TAS will take the results of the one which requires the highest design flow rate. Please note that for components, the Design Flow Delta T will also size the flow rate so that the demand placed on the component is also met.
- **Load Distribution –** The Load Distribution field allows the user to decide how their DHW demand will be spread out throughout the day. The DHW demand is set in the internal condition for each zone and this field allows the user to decide how TAS Systems distributes it, using the following three options:
  - Normal – For each zone TAS will split the DHW demand up evenly over the hours that zone is occupied (TAS decides if a zone is occupied by looking at the occupancy gains field from the zones internal condition.). Once the DHW demand is worked out for each hour for the zone, the DHW collection sums the hourly DHW for each zone to produce the hourly DHW demand for the collection.
  - Daily - TAS will work out the total DHW demand for the collection for each day by summing the DHW demand for each zone in the collection. TAS will then distribute it evenly over any hour where a zone in the DHW collection is occupied.
  - Even – TAS will work out the total DHW demand for the collection for the whole year by summing the DHW demand for each zone in the collection. TAS will then distribute it evenly over any hour where a zone in the DHW collection is occupied.
- **Minimum Return Temperature –** The minimum return temperature sets a limit on the temperature of the fluid exiting the collection. This field will not impact the flow rate of the circuit but if it is set to high with respect to the flow rate you will receive warnings informing you that the demand is greater than the supply. An example of this field being set to high with respect to the flow rate can be easily seen when using the Design Flow delta T field. Upon entering a Design Flow Delta T of $\boldsymbol{y}$ where the collection is being provided fluid at a temperature of $\boldsymbol{x}$ then the flow rate will be sized so that the fluid will always be kept within the temperature range $\boldsymbol{x}-\boldsymbol{y}$ to $\boldsymbol{x}+\boldsymbol{y}$ and also so that the demand is met. If the minimum return temperature is set in between this range, then any heat source will not be able to meet the demand of the collection due to the chosen flow rate and the limit on the return temperature. If you do not wish to set a minimum return temperature then select the "None" option for this field.
- **Components Tab –** This tab allows the user to view all of the components that are part of this collection. The user can reorder this list if they wish, please note that this will have no impact on the results.

5.4 Refrigerant Collection



A Refrigerant Collection groups together a selection of refrigerant coils that share a common energy source. Please note that in the plant room, Refrigerant Collections can only be attached to refrigerant components, like for example heat pumps. The refrigerant only connections are highlighted by the collection having light blue ports on its side, to indicate that only refrigerant components can be connected to it. To add an air-side component to a collection requires the user to select the collection in the Refrigerant Collection field of the air-sides component's properties. While sensors from a controller can be attached to the collection, a controller cannot be used to control any of the properties of a collection.

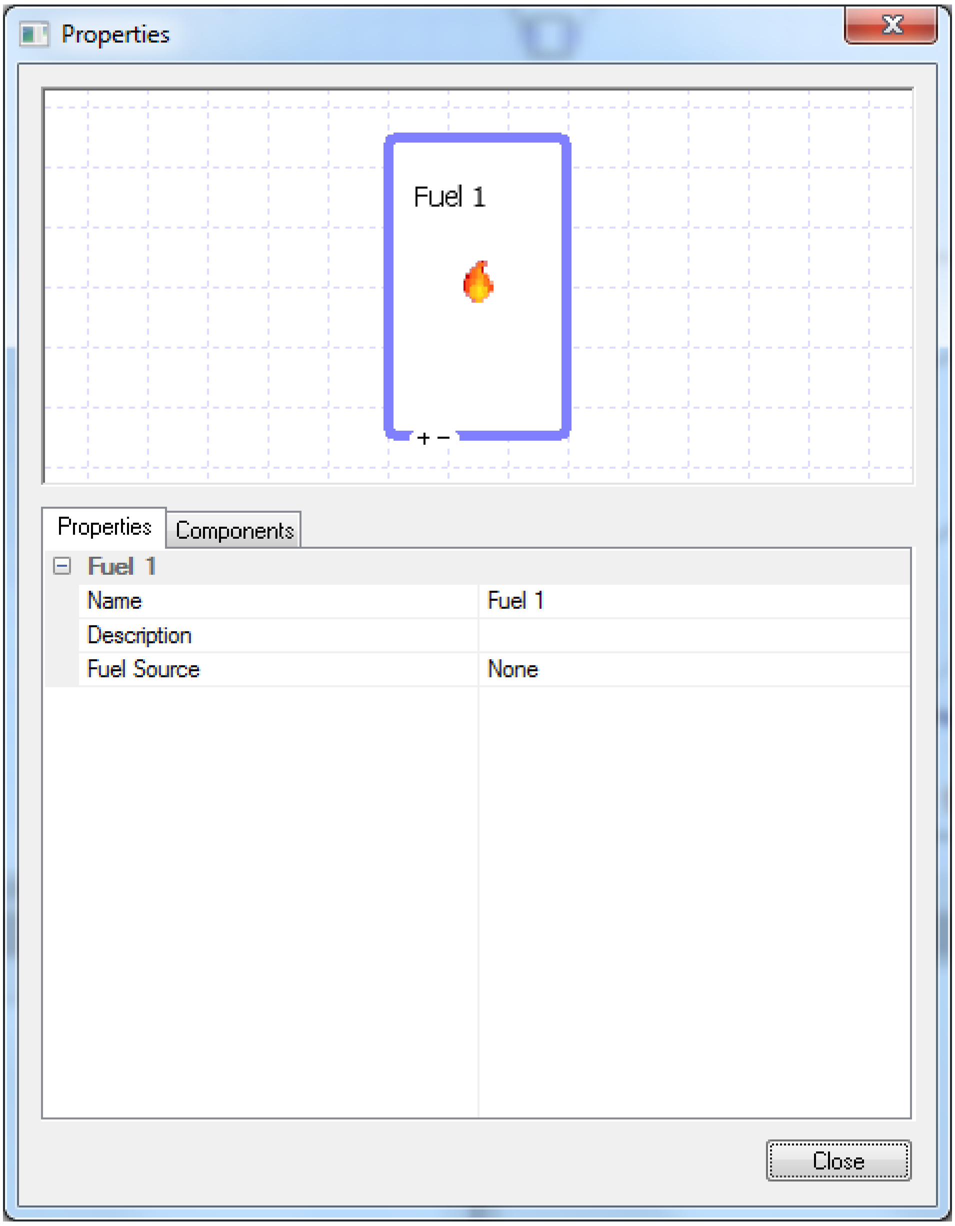
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Pipe Length –** The Pipe Length field allows the user to enter in the length of the refrigerant pipes that run through the refrigerant collection. While this field is not used directly by the

calculations it can be used by modifiers on the heat pump, for instance when the user imports a Mitsubishi heat pump into their file the capacities will have a pipe length modifier applied to them.

- **Components Tab –** This tab allows the user to view all of the components that are part of this collection. The user can reorder this list if they wish, please note that this will have no impact on the results.

5.5 Fuel Collection



A Fuel Collection can be used instead of a heating collection to group together a selection of heating coils / heating components that directly burn fuel at the air-side component to produce heat, instead of being connected up to boilers and other plant room components. As the fuel collection is used to model the fuel being used at the air-side component, it does not have any ducts to connect up to plant room components. To add an air-side component to a fuel collection requires the user to select the collection in the heating collection field of the air-sides component's properties. While sensors from a controller can be attached to the collection, a controller cannot be used to control any of the properties of a collection.

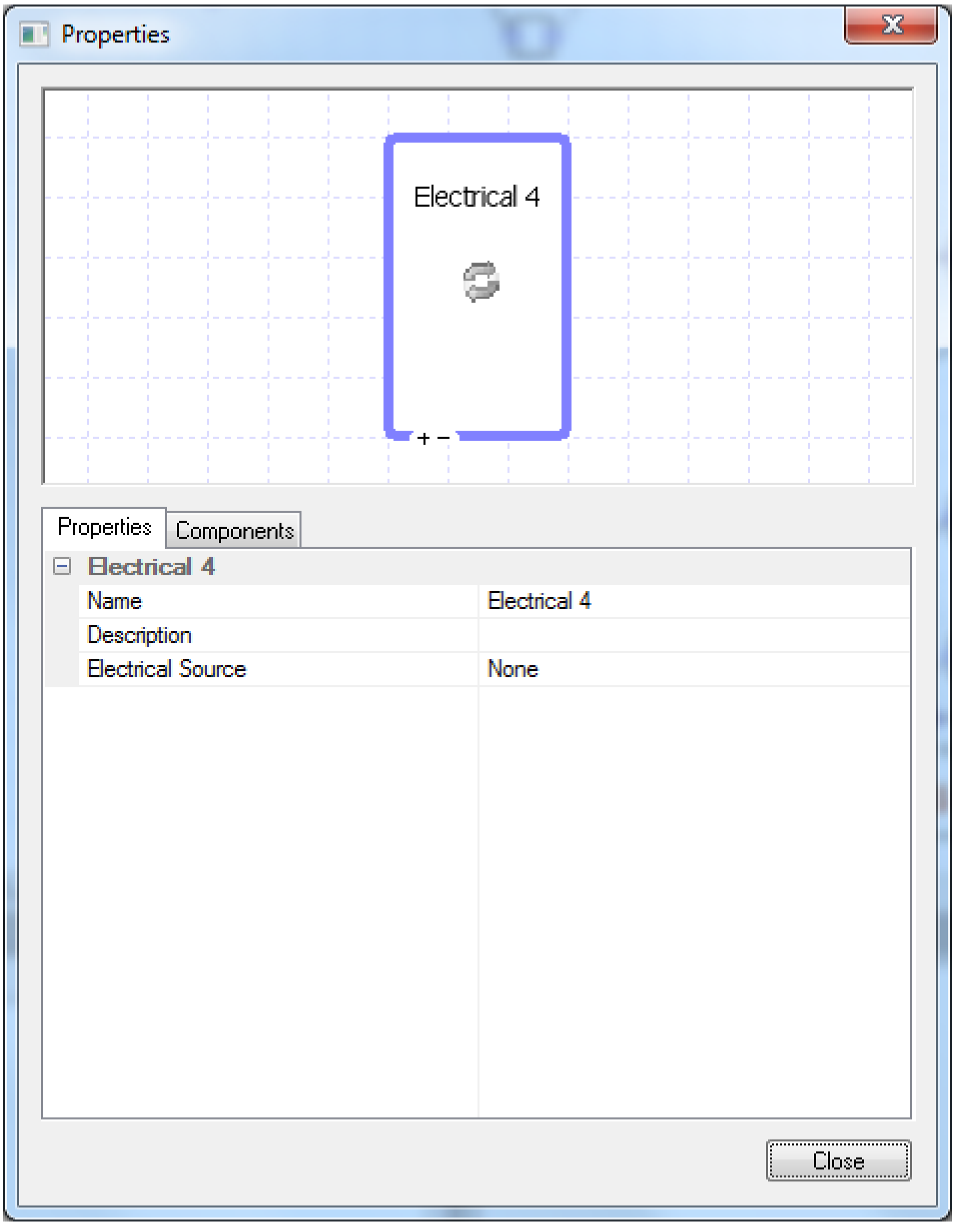
Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Fuel Source -** With this field the user can choose the fuel source of the fuel collection. For the fuel collection, the fuel selected here will be burned to produce the heat at each coil /

component which is a member of this collection. The options provided in the drop down menu for the fuel source come from the fuel sources placed in the fuel source folder. If no fuel sources have been placed in this folder, the only option available will be the “none” option. If the “none” option is used, you will obtain a warning and the loads of the component will be discarded.

- **Components Tab –** This tab allows the user to view all of the components that are part of this collection. The user can reorder this list if they wish, please note that this will have no impact on the results.

5.6 Electrical Collection



The main use of an Electrical collection is to group together electrical demands from lighting and equipment for reporting purposes, although it can be used instead of a heating collection to group together a selection of heating coils / heating components that generate heat using electricity as a fuel source. As the electrical collection is used to model electricity being used by the components directly in the air-side systems, it does not connect up to any plant room components. To add an air-side component to an electrical collection requires the user to select the collection in the heating collection or electrical collection field of the air-sides component's properties. While sensors from a controller can be attached to the collection, a controller cannot be used to control any of the properties of a collection.

Properties:

- **Name –** This is the name of the component, it will be used in reports or error messages. You can rename components as you wish.
- **Description –** The Description field allows the user to enter a description of the component. By default it is left blank.
- **Electrical Source –** With this field the user can choose the Electrical source of the electrical collection. The electrical source chosen here will be the electricity provided to the members

of this group. The options provided in the drop down menu come from the fuel sources placed in the fuel source folder, so the user will be able to choose a non-electrical fuel source but this is not advised. If no fuel sources have been placed in this folder, the only option available will be the "none" option. If the "none" option is used, you will obtain a warning and the loads of the component will be discarded.

- **Components Tab –** This tab allows the user to view all of the components that are part of this collection. The user can reorder this list if they wish, please note that this will have no impact on the results.