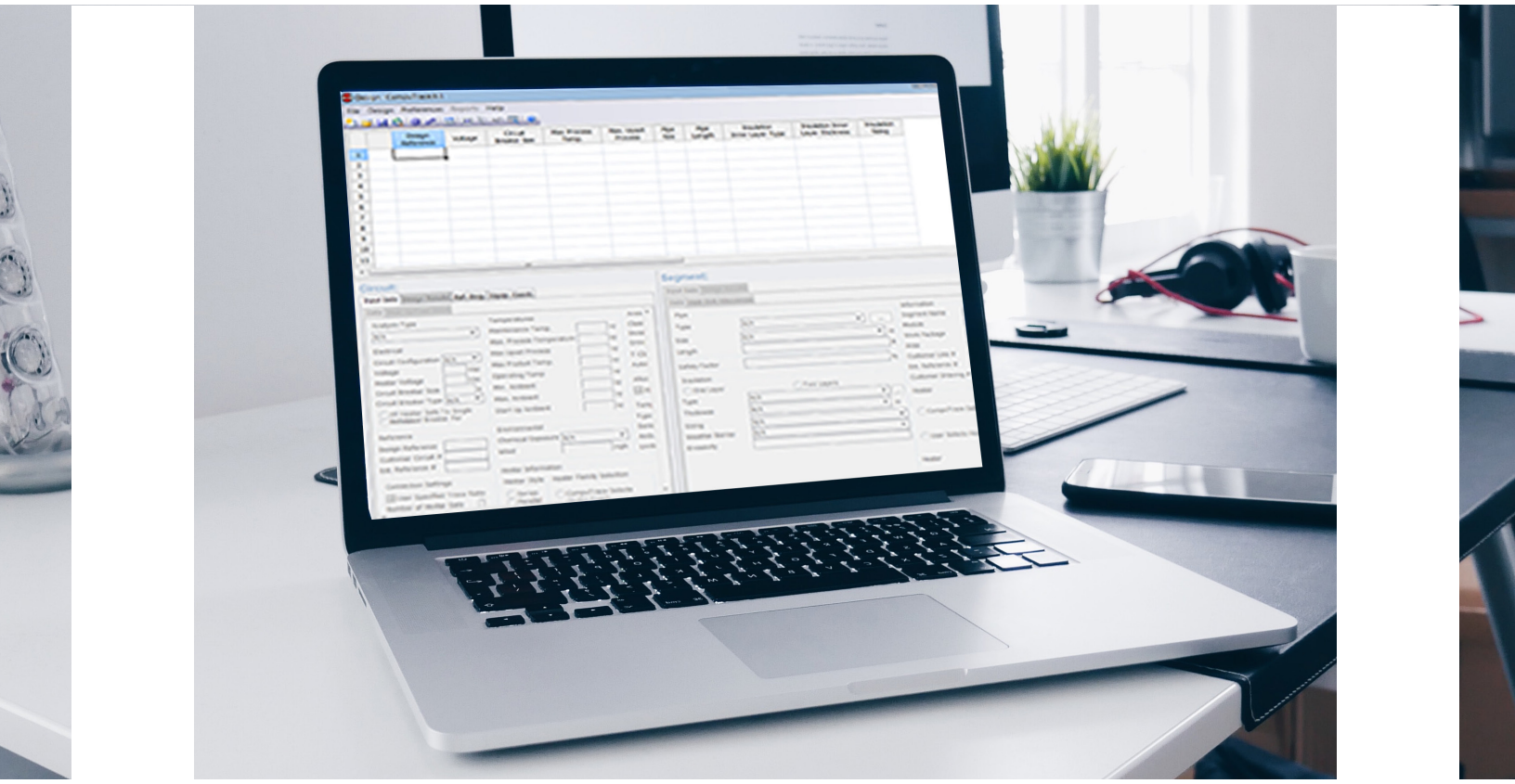


COMPUTRACE 6.1 USER'S GUIDE

Thermon, Inc.

Version 6.1 | Revision 5 | October 2022

www.thermon.com

Contents

Section I. CompuTrace Design Suite Overview

CompuTrace Design Suite heat tracing design and estimating software is provided for the users of Thermon, Inc. heating products. It provides the tools necessary to design and specify a complete electrical heat tracing system.

What's New?

- This latest rendition of the CompuTrace heat tracing design program has been updated with the main focal point being on making it more intuitive while adding new views and abilities. Some of the new features include:
- Improved User Interface Layout, including realignment of circuit-level and segment-level input fields
- New Circuit View Grid, offering an optional method for data entry
- Enhanced Bill of Materials customization abilities
- Improved Reporting Systems
- Improved Performance Speed for large projects

Minimum System Requirements

CompuTrace Design Suite is intended for use on any computer system running Microsoft Windows® and meets the following minimum requirements:

Thick Client Requirements

- Minimum requirement, Windows 7®; Windows 10 recommended
- Microsoft Excel 2007® or above
- 2 GB RAM (minimum) 4 GB RAM (recommended)
- 100 GB Hard Disk Drive (recommended)
- 3 GB Free Space (recommended)
- 1.5 GHz Processor
- Mouse or compatible pointing device

Design Limitations

CompuTrace Design Suite does not address the following design situations:

- Tank or vessel heating
- Underground piping
- Condensate prevention applications
- Flowing product applications

For information on or design assistance with any of these applications, please contact your local Thermon engineering office. Contact information can be found at www.thermon.com.

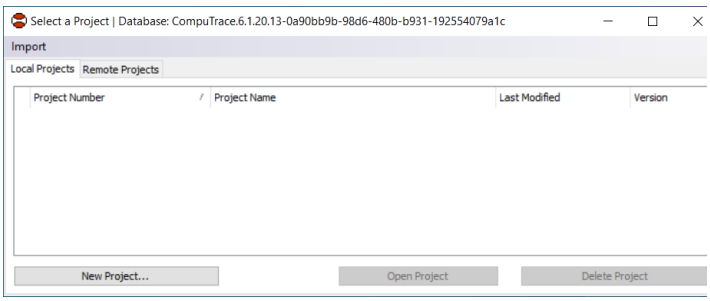
Section II. Getting Started with CompuTrace Design Suite

To Start CompuTrace Design Suite:

Click on the application icon on your desktop.

Select a Project Screen

The screen below will appear:



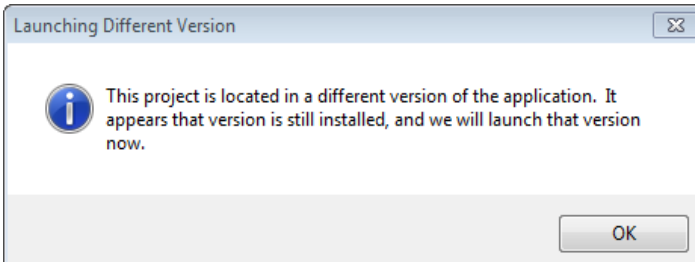
This screen includes an **Import** menu above two project tabs, **Local Projects** and **Remote Projects.**

Import - This menu item allows the user to import an existing CompuTrace 6.1 file on the local computer via .xml format.

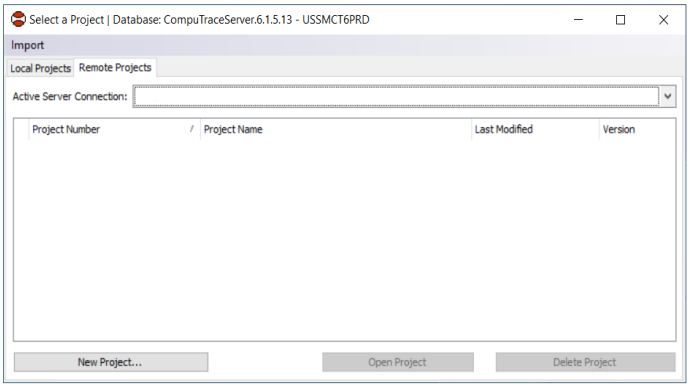
N.B. A CompuTrace 6.0.x file cannot be imported and used in CompuTrace 6.1.x

Local Projects - This type of project mode is for single users working on a project. The user is able to either create, open, or delete a project. If CompuTrace Design Suite has been installed for the first time on the computer, no previous projects have been created. Thus, both the **Open Project** and **Delete Project** options are disabled.

NOTE: If any CompuTrace version 6.0 or higher has been installed on the computer and projects designed with that version exist on the computer, these will be listed in the **Local Projects** pane. If a project from a previous version is selected, the program will open the project selection screen for the version of CompuTrace used to perform the designs.



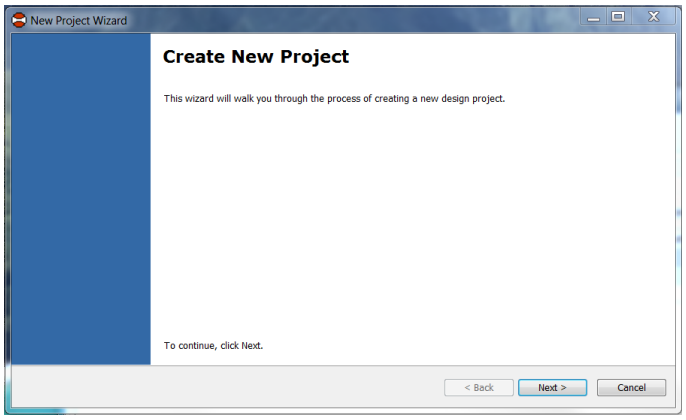
Remote Projects - This type of project mode is for users working in collaboration with other users on a project. The user is able to either create a new Remote Project or connect to a currently existing Remote Project that another user has already created.



Refer to the **Using Remote Projects** section of this manual for more information on how to set up new Remote Projects or connect to existing Remote Projects.

Create New Project Wizard

When "New Project" is selected on the Local Projects screen or on the **Remote Projects** screen, the Create New Project Wizard is opened.

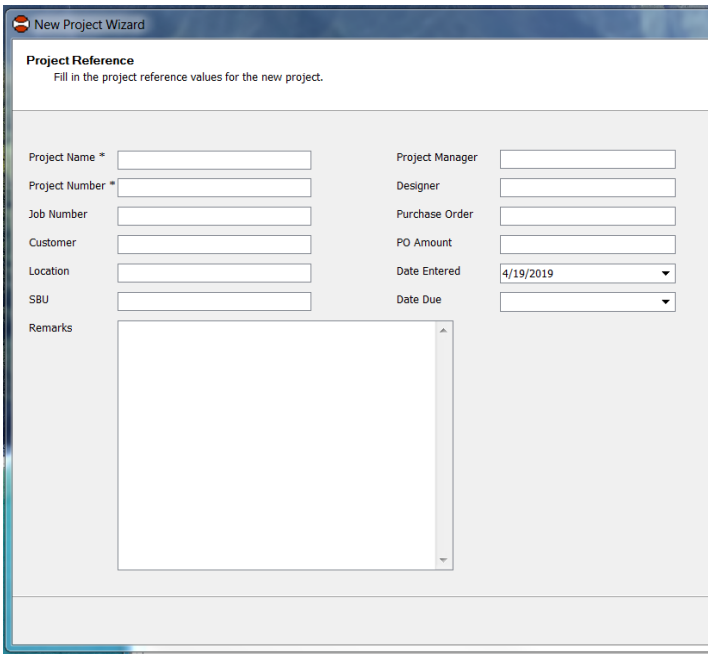


The New Project Wizard will now begin the creation of a new design project, click **Next**.

NOTE: In the next few steps, if the **Cancel** button is pressed at any time, the new project currently being set up will be voided and return the user to the **Select a Project** screen.

Project Reference

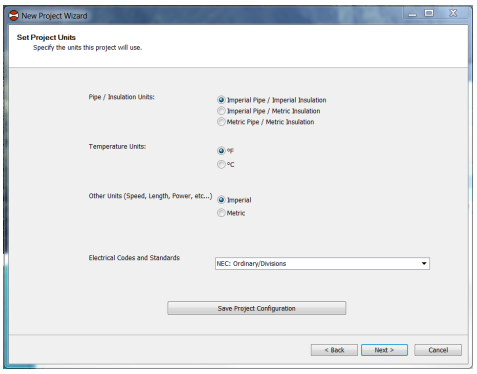
The information on this screen is for reference only, which will be displayed on reports and can be used to identify the origin of the report or design.



NOTE: All fields are optional with the exception of **Project Name** and **Project Number**. Project Name must be unique for each project in the database.

Enter the applicable reference information and click **Next**.

Set Project Units



The information on this screen sets the project units for measurements and temperatures, as well as the selection of the applicable electrical codes and standards for the application.

NOTE: The selections made on this screen can only be changed at this level and cannot be altered once the project has been created.

Electrical Codes & Standards: Selections under this menu define the electrical standards used to define how design aspects such as T-ratings are determined. All designs in a project must be performed to meet the same standard. This selection cannot be changed after the project is created.

ATEX (Ordinary/Zones): European Union using IEC Zones method approach to European Norms (EN) based on IEC 60079-10-1/-2, EN 60079-30-1/-2 and IEC/EN 62395-1/2.

IECEx (Ordinary/Zones): International (other than European Union and North America) using IEC Zones method approach based on IEC 60079-10-1/-2, IEC/IEEE 60079-30-1/-2 and IEC/EN 62395-1/2.

NEC (Ordinary/Divisions): United States using Class and Divisions method approach based on NFPA 70 Article 500, IEC/IEEE 60079-30-1/-2 and IEEE 515.

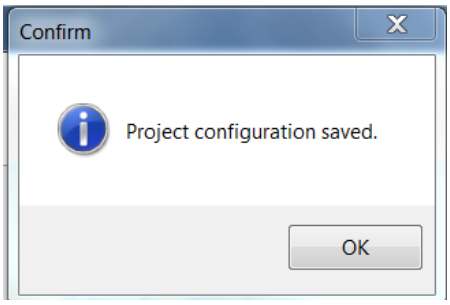
NEC (Ordinary/Zones): United States using Zones method approach based on NFPA 70 Article 505, IEC/IEEE 60079-30-1/-2 and IEEE 515.

CEC (Ordinary/Divisions): Canada using Class and Divisions method approach based on CSA 22.1 , CSA 22.2 No. 130 and CAN/CSA-C22.2 NO. 60079-30-1/-2.

CEC (Ordinary/Zones): Canada using Zones method approach based on CAN/CSA 22.1 Section 18, CSA 22.2 No. 130-16 and CAN/CSA-C22.2 No. 60079-30-1/-2.

When clicked, 

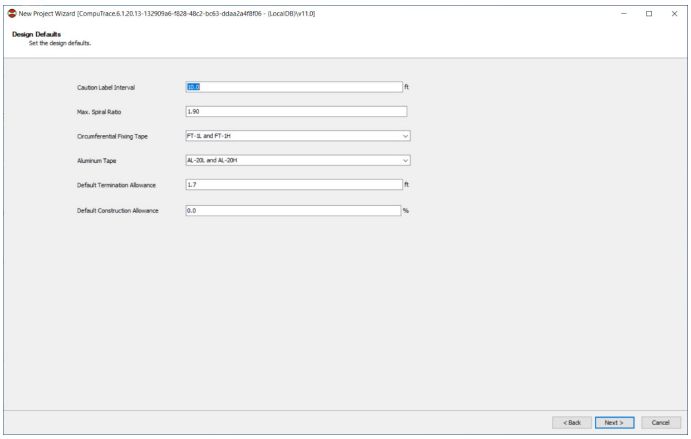
the button will save the current configuration settings for the units selected. Note: If the current configuration settings are saved using this button, any new projects created will utilize the unit selections just saved as the default.



Enter the appropriate information and click **Next.**

Design Defaults

The information on this screen sets defaults for various design criteria.



Caution Label Interval: Used to determine the number of caution labels to be included in the Bill of Materials. The default value is 10 ft (3.0 m).

Max. Spiral Ratio: For spiral designs, it signifies the length of heater per length of pipe. If the required spiral ratio exceeds the value in this field, the program will indicate that the design is inappropriate for spiraling and will recommend parallel passes. The maximum allowable input for this field is 1.90.

NOTE: The selection of a spiral ratio made on this screen can only be changed at this level and cannot be altered once the project has been created.

Circumferential Fixing Tape: A drop down list containing 2 entries:

FT-1L and FT-1H: When selected, allows the program to determine whether to include the lower temperature FT-1L or the higher temperature FT-1H in the Bill of Materials for each circuit.

FT-1H Only: When selected, dictates that the program should only include FT-1H in the Bill of Materials. If a design file contains designs appropriate for both types of fixing tape, there could be confusion in the field which could result in the incorrect tape being used. In these situations, it might be better to simply use the higher temperature tape for all designs.

Aluminum Tape: A drop down list containing four entries used to specify which aluminum tape to include in the Bill of Materials for circuits that utilize it.

AL-20L and AL-20H: Program selects the appropriate 2" wide aluminum tape based on temperature criteria for each design.

AL-20H Only: Program will only include the higher temperature 2" wide aluminum tape in the Bill of Materials for designs utilizing aluminum tape.

AL-30L and AL-30H: Program selects the appropriate 3" wide aluminum tape based on temperature criteria for each designs.

AL-30H Only: Program will only include the higher temperature 3" wide aluminum tape in the Bill of Materials for designs utilizing aluminum tape.

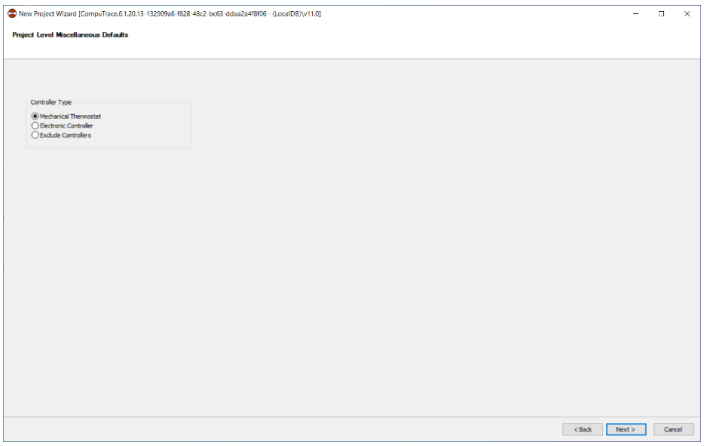
Default Termination Allowance: Applicable to parallel resistance heaters only, it is the length of heater that the program should allocate for each power connection, splice or termination. This field allows the program to include the amount of heater traveling from the pipe to the junction box in the amperage calculations and Bill of Materials. Default value is 1.7 ft (0.5 m).

Default Construction Allowance: Enter a percentage of the design length that should be added to account for field variances and routing. The allowance will be added to the overall heater length when determining heater output, electrical loading and BoM quantities.

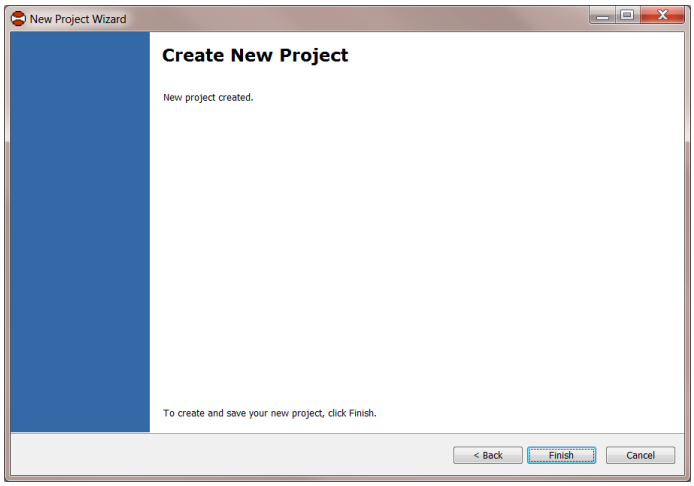
Enter the appropriate information and click **Next**.

Project Level Miscellaneous Defaults

This screen allows the user to define the default controller type that will be available for later selection in the circuit bills of materials. Note: this can be overridden within the project if required.



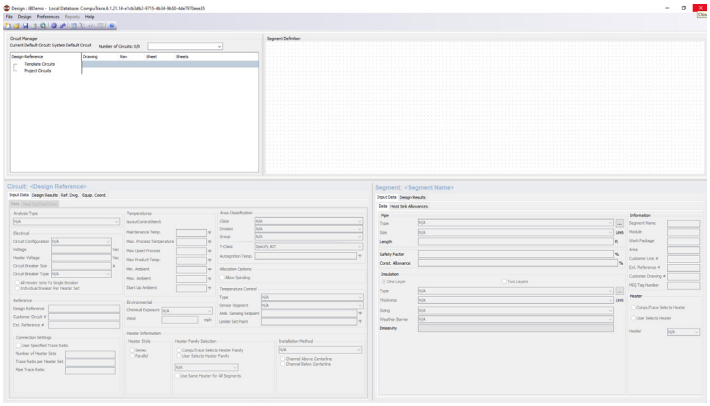
New Project Created Confirmation Screen



This screen confirms the design parameters entered for the new project. Should any modifications need to be made to any of the previous data, click **Back**, Cancel to void, or **Finish** to confirm the creation of the new project.

When **Finish** is chosen the confirmation screen will close.

NOTE: It may take a few seconds for CompuTrace Design Suite to load the main project screen shown below:



This is the main design window of the new CompuTrace Design Suite program which has incorporated the settings defined in the New Project Wizard. The user can now start creating designs.

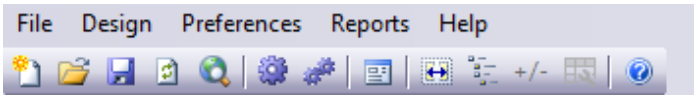
Navigating the Main Design Window

The main design window contains three basic sections.

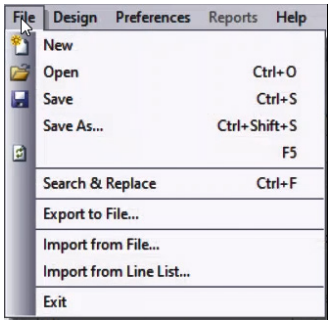
- Menu Bar / Tool Bar
- Circuit Manager / Segment Definition Layout
- Circuit / Segment Input Data Tabs

Menu Bar

This section serves as the main settings control for the program. Here, functions such as opening/ saving a project file, import/export, reporting, as well as help on specific topics can be found. Listed by its corresponding menu is a brief description of the functionality of each command.



File Menu



New

Use this menu item to open the **New Project Wizard.**

Open

Selecting this item will introduce the **Select a Project** dialog box, allowing the user to select an existing project file to open. This command can also be executed with the keyboard shortcut Ctrl+O (hold down the Ctrl key and press the letter 'o').

Save

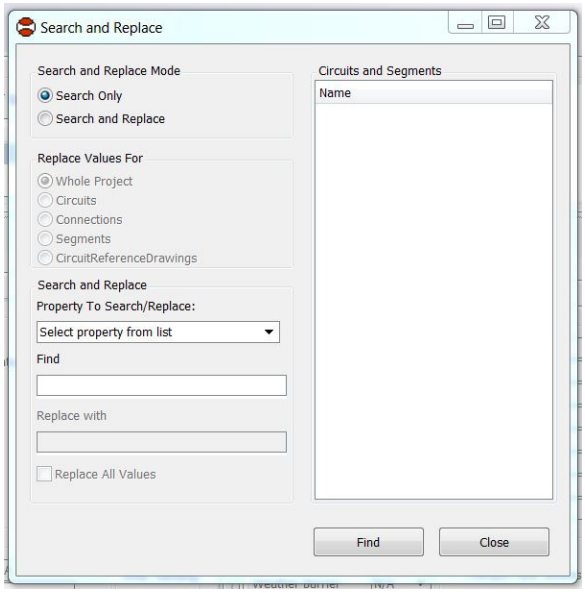
Select this menu item to save the changes to the open project. This command can also be executed using the keyboard shortcut Ctrl+S (hold down the Ctrl key and press the letter 's').

Save As

Selecting this menu item allows the user to save a copy of the current project under a different name. This command can also be executed using the keyboard shortcut Ctrl+Shift+S (hold down the Ctrl and Shift keys and press the letter 's').

Search & Replace

Selecting this command will open the Search and Replace window below. This feature serves as an aid to the user in searching for specific data by a specific property input value, as well as replacing that data by a specific property input value. This feature functions in a similar manner as other "Search & Replace" in other programs. For a complete explanation of this feature, see **Using the Search and Replace Feature** section in this manual.



Export to File

Selecting this command will export the current project to an .xml file. This can be used to transfer the file to another designer or to archive the project.

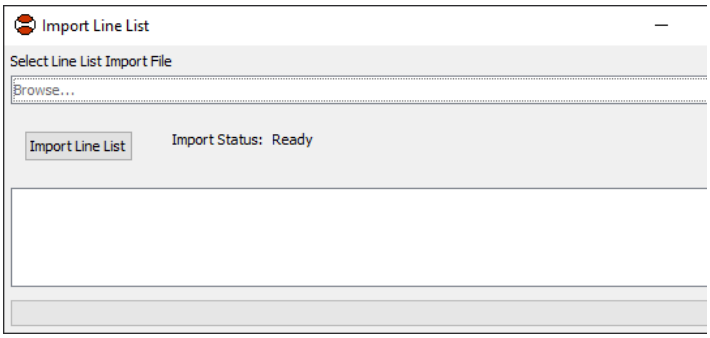
Import from File

Selecting this command will allow the user to import a project saved in an .xml format. The current project will be closed when a new project is imported.

Import from Line List

Selecting this command will open the Import Line List dialog box shown below.

Select File



Clicking this button will open a standard Open dialog box to navigate to the desired Line List.

Get Template

Clicking this button opens a properly formatted Excel® file to use as a template for the line list import file.

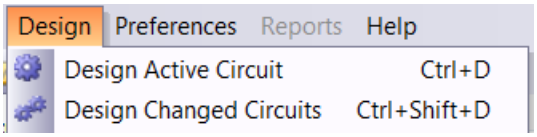
Import Line List

Clicking this button begins the import process for the selected line list. The window below the **Import Line List** button will confirm the import is complete and whether all segments were successfully imported. Any cell left blank in the Line List Template will be set to default CompuTrace values when imported. Once the import is complete, close the import window by clicking on "X".

Exit

This command will close the project file and exit CompuTrace Design Suite. If there have been any changes to the file since the last save command was issued, the program will prompt the user whether the changes should be saved before closing.

Design Menu



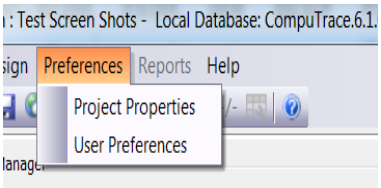
Design Active Circuit

Selecting this option will cause CompuTrace to (re) generate a design using the design conditions defined by the currently selected circuit.

Design Changed Circuits

Selecting this option will initiate CompuTrace to (re)generate designs using the design conditions defined for all modified/undesigned circuits.

Preferences Menu

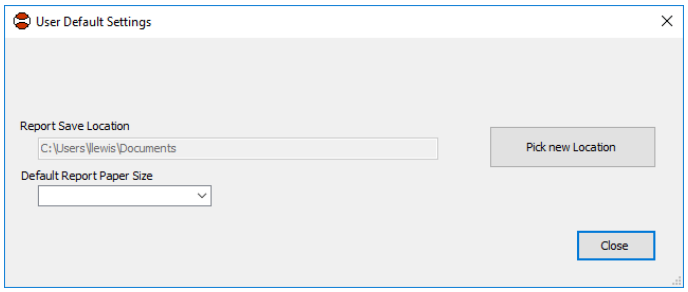


Project Properties

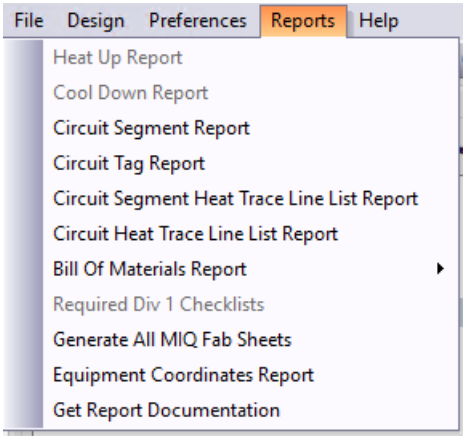
Selecting Project Properties opens a dialog box showing some of the values for the project level variables that were set when the project was initiated. CompuTrace will allow these fields to be adjusted from this point with the exception of the Max Spiral Ratio, project units and electrical code selections.

User Preferences

Selecting User Preferences opens a dialog box that allows the user to select a file location for CompuTrace to download/save all reports created by the program.



Reports Menu



The Reports Menu will only be available **after** a successful design has been created. Otherwise, it is not a selectable option. These reports are a summary of the design on a project level. Each report is created in Microsoft Excel®.

The following list of reports each provide a summary of design results in various formats as their titles depict:

Heat Up Report

This report will show the input and output values, including a graph of heat up time and temperature for each segment and circuit that has an **Analysis Type** set to Heat Up.

Cool Down Report

This report will show the input and output values, including a graph of cool down time and temperature for each segment and circuit that has an Analysis Type set to Cool Down.

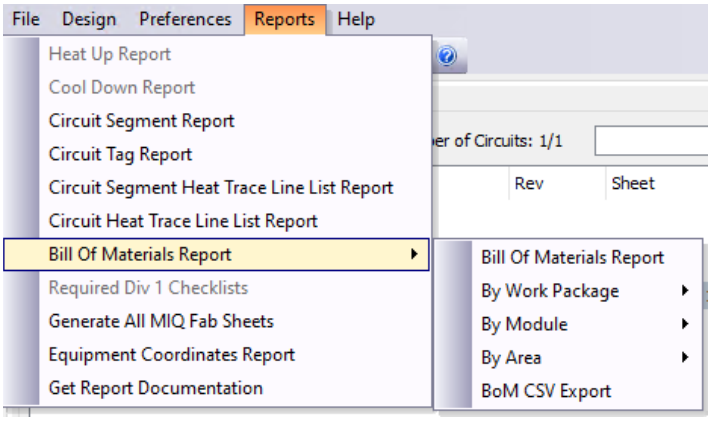
Circuit Segment Report
This report gives the result summary for each segment in the circuit.

Circuit Tag Report
This report gives a list of the user defined tag information provided for the Connection Kits and other components in the project.

Circuit Segment Heat Trace Line List Report
This report gives a summary of both the Circuit Segment and Line List data.

Circuit Heat Trace Line List Report
This report gives a summary of the Line List data.

Bill of Materials Report
Similar to the Bill of Materials in past CompuTrace versions, this report provides a summary list of all the physical components required based on the design. In addition to an overall summary, this report can be filtered by Work Package, Module or Area.



Required Div 1 Checklists
A document that is mandatory to be filled out and retained for designs in which a Division 1 Area Classification is selected in projects using the NEC: Ordinary/Divisions Electrical Codes. This checklist is enabled under this area classification selection only.

NOTE: The Division 1 Checklist is provided in a .pdf format and will download only once for each project; the Checklist will automatically be saved to the location defined by the user (refer to User Preferences for more information).

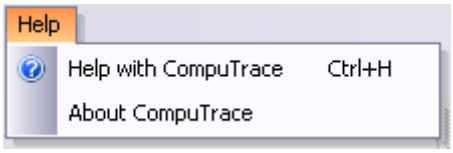
NOTE: Generate All MIQ Fab Sheets
This enables the user to generate all MIQ fabrication sheets for designed circuits utilizing MIQ heaters.

Equipment Coordinates Report
This report provides a summary list of all the heating system components required based on the design, which can include design reference, physical coordinates and user tag if input in the design process.

Get Report Documentation
Clicking this button opens a Microsoft Word® compatible file showing the definitions of the terms used in the various Reports.

Additional summary reports on a circuit level only can also be found using a menu in the Circuit Manager. See **Circuit Manager Reports** in this manual for more information.

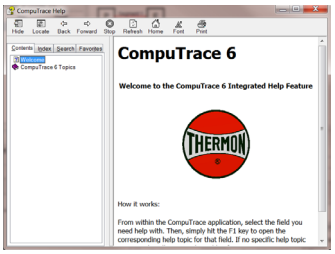
Help Menu



Help with CompuTrace
Selecting this menu item will open the context-sensitive help system. From this window, the user will be able to select from the Table of Contents, Index Tabs, or Search for the term for which they need further explanation.

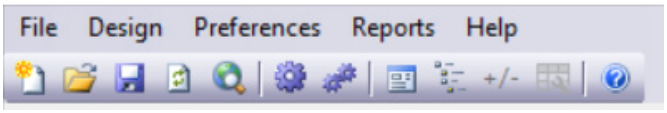
The Help system will also be available from the main screen by selecting the field in question and pressing the **F1** key.

About CompuTrace



Selecting this menu item will open the **About CompuTrace** dialog box, which will display the program version as well as contact information for additional assistance if needed.

Tool Bar



The Tool Bar provides one-click access to some of the more common commands found in the Menu Bar, as well as new features associated with the NEW Grid View pane.

New Circuit - creates a new default circuit in the system

Open - prompts the user to save the current project and open another project

Save – saves the current project

Refresh – **NEW FEATURE** - pulls latest project information from the database. Useful for updating changes from other users on a collaborative, server based project.

Search and Replace – refer to **Using Search and Replace** in this manual

Design Active Circuit - designs the active circuit

Design Changed Circuits – designs all circuits that have not been designed

Manage External Load Circuits - adds circuit load data and BOM components to the project from circuits not designed in CompuTrace. Once added, these materials will be included in the project reports and loads can be included in the assignment of circuits to circuit breakers. Refer to **Manage External Load Circuits** in this manual for more information.

Grid View – **NEW FEATURE** – this radio button toggles between the standard Tree View and the **NEW** Grid View. Refer to Grid View Layout in this manual for more information.

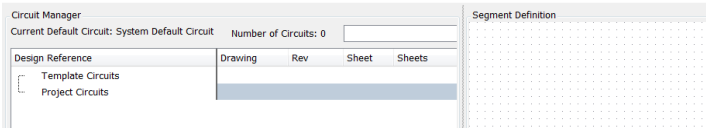
Expand/Collapse Grid View - expands/collapses the Grid View to full screen and back

Grid View Configuration – selects the "Displayed Columns" to be shown in the Grid View

Help – activates the Help Menu

Circuit Manager/Segment Definition Layout

The second basic section of the main design window is one of the key features in the CompuTrace Design Suite, presenting the user with the ability to design multi-segment circuits. The layout is comprised of two panes: **Circuit Manager** and **Segment** Definition. Each section is associated to one another so that if changes are made to one pane, such as addition or deletion of segments, then the other pane will also be affected. When a new project is opened, the Circuit Manager and Segment Definition panes will be blank.



Definitions of General Design Terms:

Segment: A pipe and associated components with a defined beginning and end.

Heater Set: A length of heater with a power connection and end termination. A heater set is composed of one or more heater segments.

Heater Circuit: A heater or group of heaters connected to a single circuit breaker. A heater circuit is composed of one or more heater sets.

Circuit Manager Pane

The **Circuit Manager** pane is a tree diagram-style control containing two levels: **Circuit** and **Segment** levels. Each Circuit consists of one or more Segments as defined by the user.

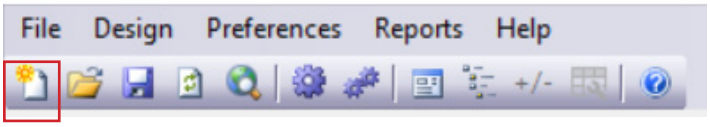
NOTE: The default names for each Circuit and Segment, as well as their respective Revision & Sheet numbers, can be renamed with a single mouse left-click on the name/number itself.

Segment Definition Pane

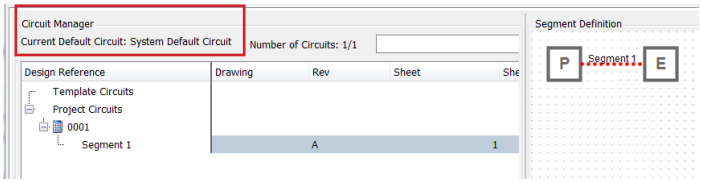
The **Segment Definition** pane is the visual representation of the Circuit Manager. Here, the user is able to create circuits by connecting segments with nodes. The relationship between the segments is defined by the type of node that connects them. All changes made to the circuit in the Segment Definition Pane are reflected in the Circuit Manager.

Working with the Circuit Manager Pane

Clicking the "New Circuit" icon on the Tool Bar will insert a system default Project Circuit in both the Circuit Manager pane and the Segment pane.

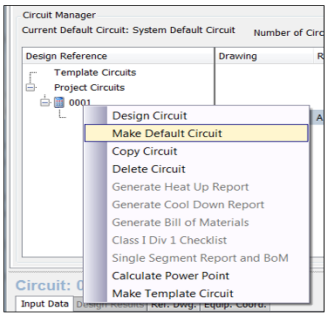


Since the Circuit Manager and Segment Definition panes are linked, any changes made in one pane will be reflected in the other. However, there are some functions that are only possible in the Circuit Manager pane. Two of those are setting the default Project Circuit and setting Template Circuits.



Default Project Circuit

This can be customized by modifying any circuit as needed, then right-clicking on the selected circuit and choose "Make Default Circuit". Once a new default circuit is created, this will be the circuit added each time the "New Circuit" icon is clicked.

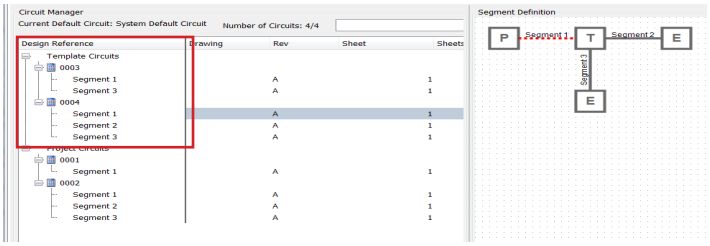


This process can be repeated and the newest saved default Project Circuit will overwrite the previously saved version.

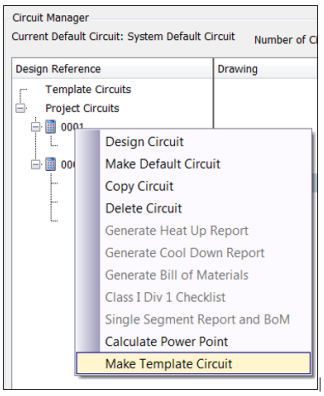
It is also important to note that changing the default circuit configuration only affects the current project, and will not be used in other projects. Thus, the process must be repeated for each new project. The system default circuit configuration will always be used when a new project is opened.

Template Circuits

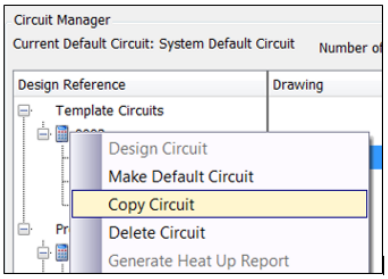
New to CompuTrace 6.1 is the ability to create Template Circuits that can be used instead of the default Project Circuit. This will save time if the project has more than one basic circuit type, since multiple Template Circuits can be created and saved.



Template Circuits can be created by modifying the initial default circuit as needed, then by right-clicking on the desired circuit and selecting "Make Template Circuit".



All Template Circuits will be available and accessible by right-clicking on the desired Template Circuit and selecting "Copy Circuit", which will add the copied circuit to the bottom of the Project Circuits list.



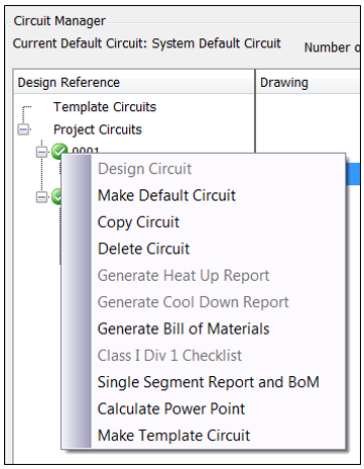
NOTE: Template Circuits are not part of the system design and will not be reflected in any reports.

Circuit Manager Reports

In addition to the actions outlined above regarding default Project Circuits and Template Circuits, summary reports similar to those available in the Reports Menu but on a **circuit level only** can be found using the menu in the Circuit Manager pane.

For example, if there were more than one circuit being designed, by right-clicking on the desired circuit, the menu options available would be for that circuit exclusively.

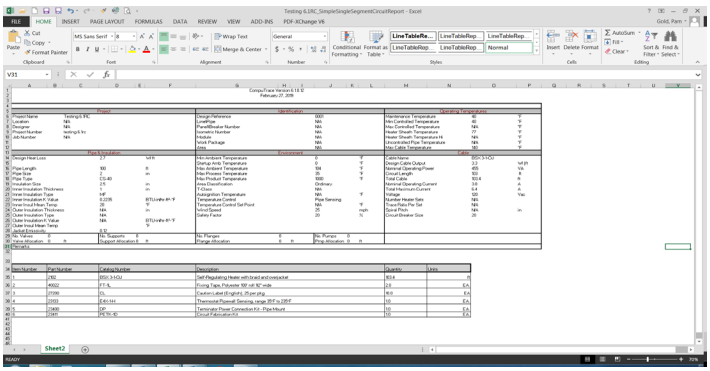
On the Circuit Manager, right-click on the circuit name to access the following pop up menu:



NOTE: Only actions available for the selected design will be enabled. If MIQ tracers are selected, two additional reports are also available through this menu along with the ability to define MIQ fabrication sheets on a circuit level as described below.

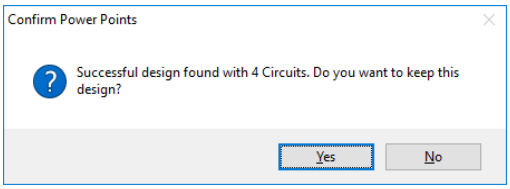
Single Segment Report and BoM

This report provides a summary of a **single segment** design only. The report shows input and output values along with the Bill of Materials for the circuit.

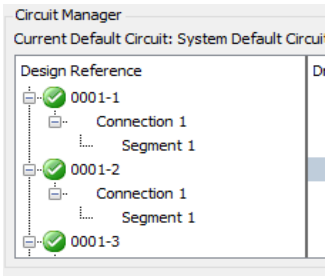


Calculate Power Point

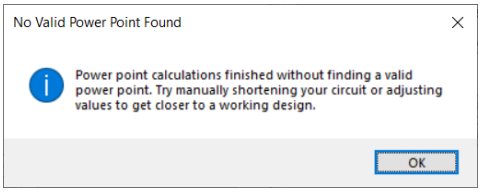
Usually selected for longer piping segments, the program will automatically calculate the number of circuits required based on the user inputs. After the program has finished the calculations, the user will be presented with a dialog box that indicates the number of circuits required and asks if one would like to keep the circuits.



If yes is clicked, then the resulting circuits will be added to the Circuit Manager beneath the original circuit using the original circuit name referenced with **-1**, **-2**, etc. appended to the end. The accepted multiple power point designs will be reflected in reports while the original circuit will remain in the Circuit Manager pane without a design.

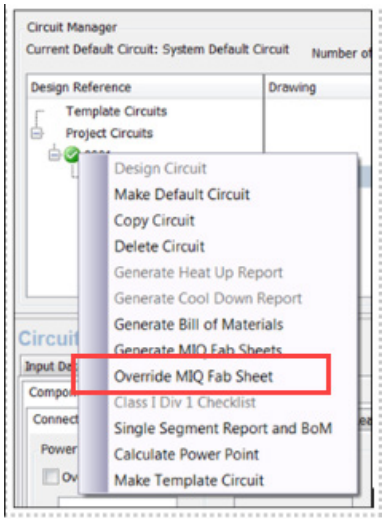


If more than 9 additional designs are required, a dialog box will be presented stating the calculations finished without finding a valid result. Suggesting input values be changed to get a workable design.



MIQ Fab Sheets

After running a design using MIQ heaters, the user has the option to use the default Fabrication Sheets or override the defaults with user-specified heater

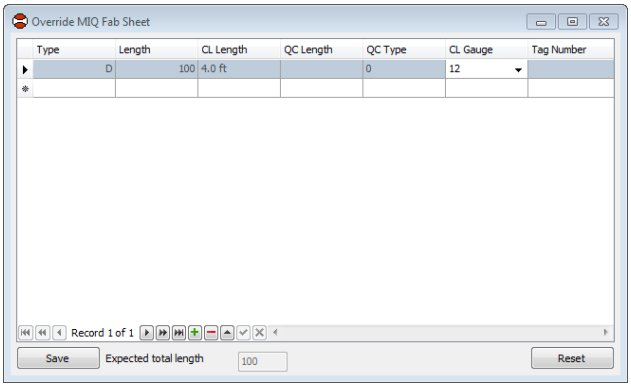


Generate MIQ Fab Sheets

Creates a Fabrication Sheet for use exclusively with MIQ heaters. The report is created in Microsoft Excel®.

Override MIQ Fab Sheet

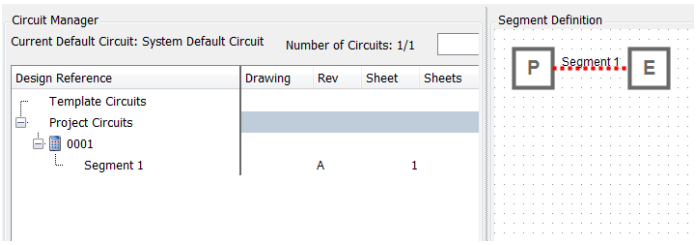
The program will present the form shown which will allow the user to define the heater segments.



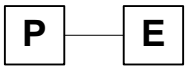
NOTE: MIQ Fabrication Sheets are required for every design utilizing MIQ Heaters

Working with the Segment Definition Pane

Click on the **New Circuit** button and the following layout will appear.



The Segment Definition pane is comprised of two main shapes: a line and box called segment and node, respectively. This default initial circuit configuration consists of a power connection, single segment, and an end termination:



The following is a list of all nodes available for creating heat tracing designs:

Power Nodes

P	Power Source
PS	Power Splice
PT	Power T-Splice

End Termination Node

E	End Termination

Splices, Loops, and Continuations

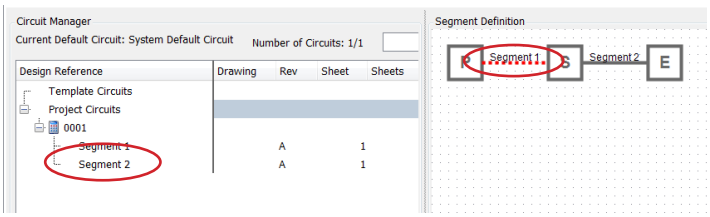
S	Splice
T	T-Splice (Available only for Parallel Cables)
L	Loop
C	Continuation

General Rules About Working with the Segment Definition Pane

Being able to select a specific node or segment is a key aspect when creating a circuit. As previously noted, both the Circuit Manager & the Segment Definition panes work in conjunction with one another, thus changes made in one area will affect the other.

A segment that is currently selected will be depicted by a dashed line as opposed to a solid line.

For Example, Circuit 1> Segment 1 is selected on the Circuit Manager:



Adding/Converting Single Nodes

Nodes and segments can be added to a circuit by converting an existing node to another type, or by adding a node to an existing end termination. A conversion operation replaces the selected node with a new node of the desired type. Depending on the conversion, one or more segments and end terminations are added to the new node. When a segment is added as a result of a node addition/conversion operation, the segment properties are copied from the segment connected to the original node.

Node addition/conversion can be performed with a **right-click**, on the node and selecting an action. See below for diagrams of each available type of node addition/conversion.

Elements in **Black** dashed lines represent the original node selected to be modified.

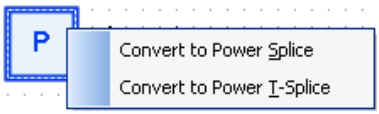
Blue elements represent converted elements as a result of the conversion.

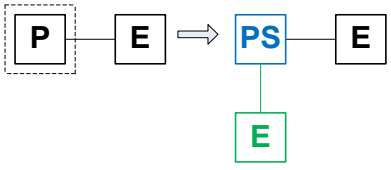
Green elements are newly added elements as a result of the conversion.

Red elements represent removed elements as a result of the conversion.

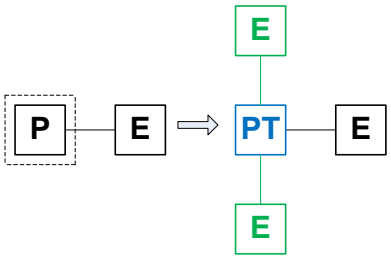
Power Nodes P PS PT

Convert Power Source to Power Splice node:

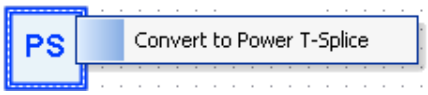


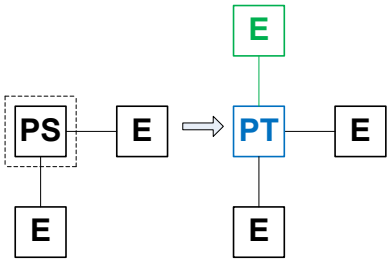


Convert Power Source to Power T-Splice node:

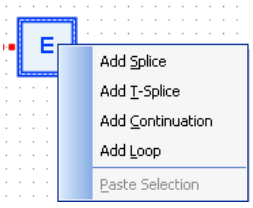


Convert Power Splice to Power T-Splice node:



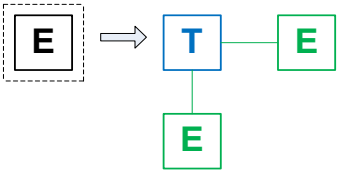


End Termination Node E

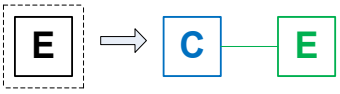


Add a Splice to an End node:

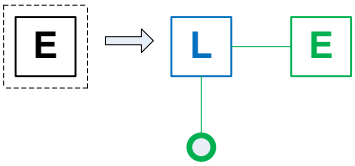
Add a T-Splice to an End node:



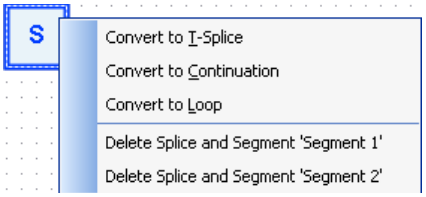
Add a Continuation to an End node:



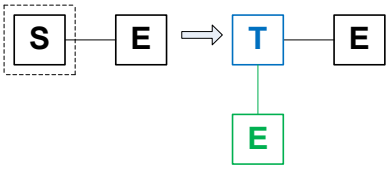
Add a Loop to an End node:



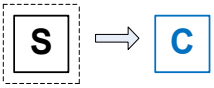
Splices, Loops, and Continuations 



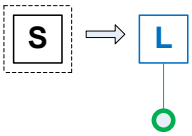
Convert Splice to T-Splice node:



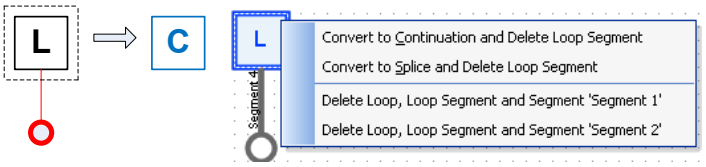
Convert Splice to Continuation node:



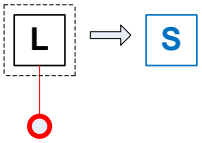
Convert Splice to Loop node:



Convert Loop to Continuation node:

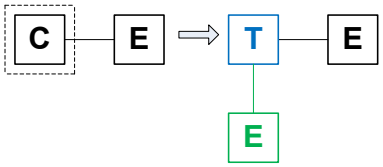


Convert Loop to Splice node:

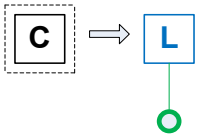


Convert Continuation to

Convert Continuation to T-Splice node:



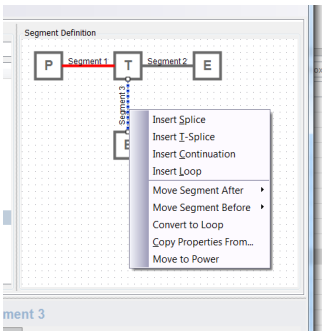
Convert Continuation to Loop node:



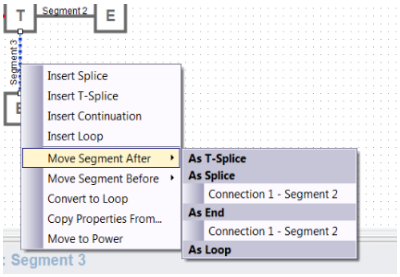
Inserting Single Nodes Into an Existing Segment

The segment definition pane allows the insertion of Splices, T-Splices, Loops, and Continuations into an existing segment. A node insertion operation also adds one or more new segments to the circuit. The new segment properties (including length) are copied from the original segment. The original segment remains unchanged.

Insertion is performed by **right-clicking** on the segment itself and selecting the appropriate option from the context-sensitive menu. Right-clicking on the segment line will result in the following menu:

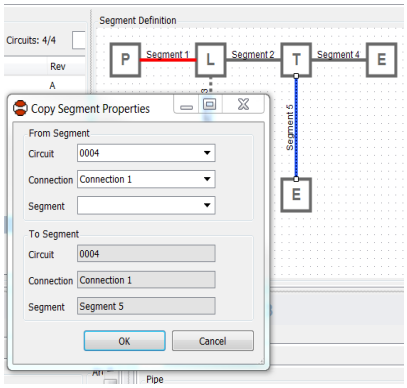


"Move Segment After" and "Move Segment Before" will provide the user with choices depending on the segments and configuration of the circuit.



"Convert to Loop" will change the selected segment to a loop.

"Copy Properties From…" will allow the user to choose another segment from any circuit and replace the design properties of the "To" segment with those of the "From" segment.

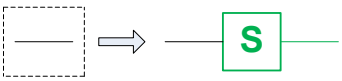


"Move to Power" will allow the user to copy any Segment and connect it to the power connection node.

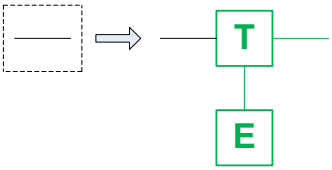
NOTE: Either the original Segment must be deleted by the user or one of the Segments must be renumbered after "Move to Power" is chosen.

The following diagrams illustrate the node operations supported by the application:

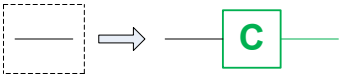
Insert Splice:



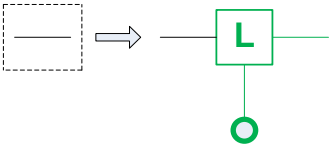
Insert T-Splice:



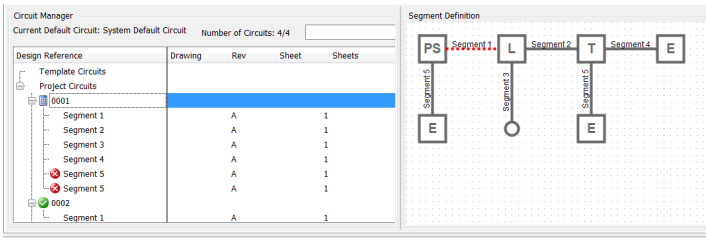
Insert Continuation:



Insert Loop:



In the below example, Segment 5 was moved to power. Either the original Segment 5 (connected to a T splice) must be deleted by the user or one of the Segments must be renumbered in order to be able to design the circuit.



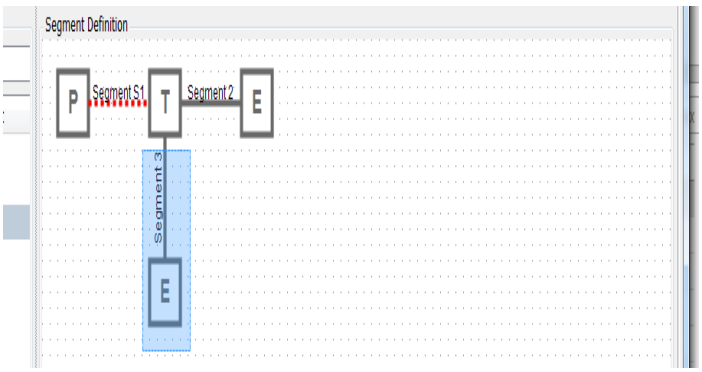
Multiple Node and Segment Operations

Selecting Multiple Nodes and Segments

The segment definition pane allows moving, copying, deleting, and pasting sets of segments from one circuit to another. In order to perform any of these operations, the selected segments and nodes must meet the following conditions:

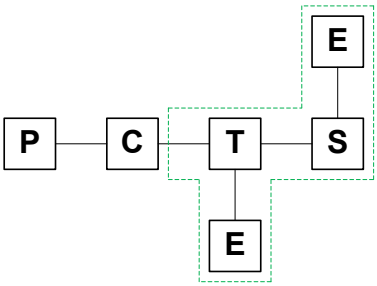
1. All segments and nodes must be connected to each other.
2. The selection must include one and only one additional open segment.
3. All selected circuit branches must be terminated by end nodes.
4. Power nodes cannot be included in the selection.

To select the Segments and Nodes for any operation, the user simply highlights the Segment(s) and Node(s) holding the left mouse button. In the below example, Segment 3 and the End Node have been selected for modification.

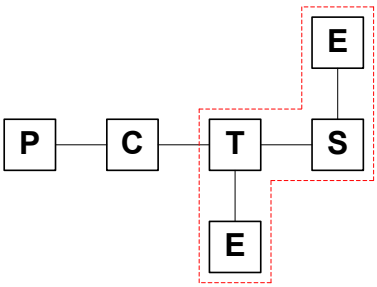


The following illustrations show valid and invalid selections:

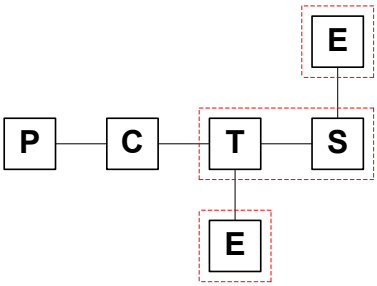
Valid Selection



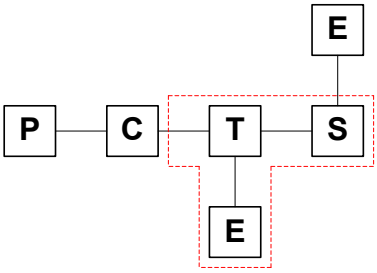
Invalid Selection No open segment included in the selection



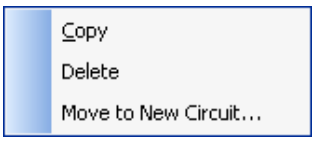
Invalid Selection Some nodes are not connected



Invalid Selection Not all branches are terminated



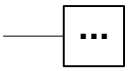
Once a valid selection is selected, **right-click** on the selected node or segment and the following menu will appear:



The next three sections will describe each feature in detail.

Deleting and Moving Multiple Nodes and Segments
The user can select a set of nodes and segments and delete them from a circuit or move them to a new circuit. The selected set of nodes and segments must meet the criteria described in the **Selecting Multiple Nodes** and **Segments** section. Deleting or moving a set of segments will also cause certain nodes to be converted to a different type.

For **illustration** purposes, a valid set of nodes and segments will be represented by the following symbol:



The following diagrams illustrate the deletion and move operations supported by the application.

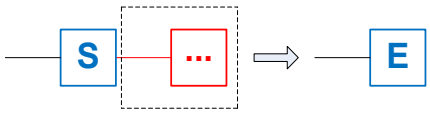
NOTE: Deleting or moving segments connected to a regular power node (not a Power Splice or Power T-Splice) is **NOT** supported.

Elements enclosed in **black** dash lines represent node selected to be modified.

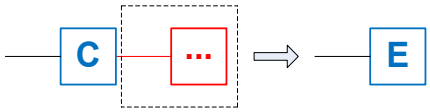
Elements in **red** represent segment and end terminations deleted from the circuit.

Elements in **blue** represent converted nodes.

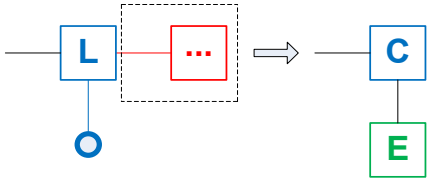
Delete or move a set of segments connected to a Splice:



Delete or move a set of segments connected to a Continuation:

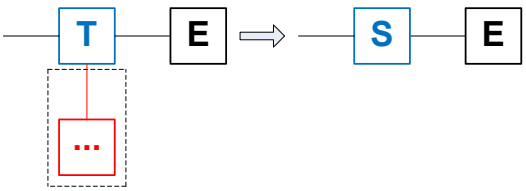


Delete or move a set of segments connected to a Loop:

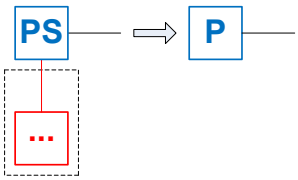


(**Note** that the loop segment is also deleted)

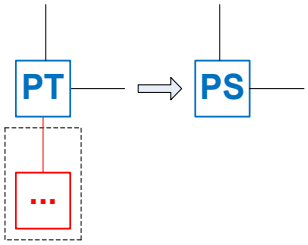
Delete or move a set of segments connected to a T-Splice:



Delete or move a set of segments connected to a Power Splice:

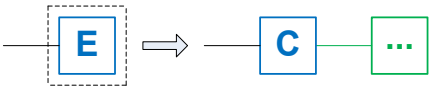


Delete or move a set of segments connected to a Power T-Splice:

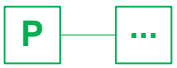


Copying/Pasting Multiple Nodes and Segments

Any set of segments and nodes that meet the criteria described in the **Selecting Multiple Nodes and Segments** section can be copied to the application clipboard. The selection can then be pasted into a new circuit or an existing circuit. Pasting into existing circuits is only allowed at end terminations. The user must select the end node where the segments will be pasted. The end termination is converted to a continuation and the pasted segments are connected to the rest of the circuit as shown below:



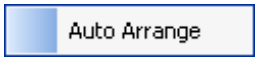
When pasting into a new circuit, a power node is added automatically and the pasted segments are connected to it as shown below:



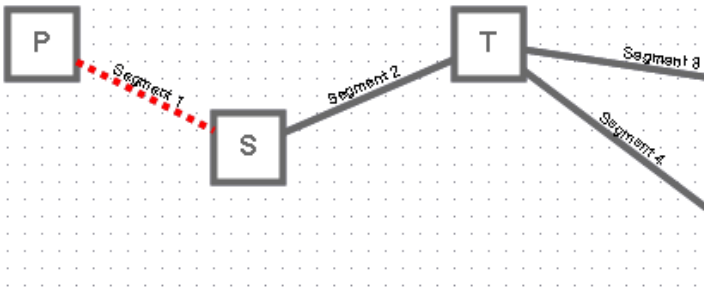
Using the Auto Arrange Command

This command is used to automatically position current nodes and segments within the Segment Definition Pane in an organized and efficient manner.

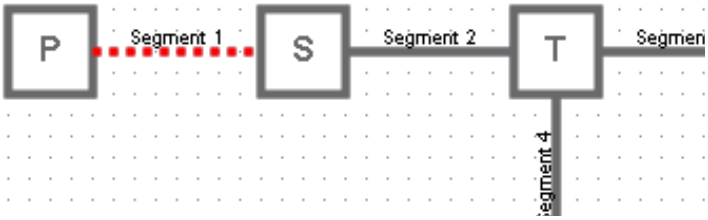
To access the command, right click anywhere on the segment definition pane where the mouse pointer is not in contact with either a node or segment (any empty area) and the **Auto Arrange** command will appear. Click on it, and the segment will be automatically arranged.



Before Auto Arrange:



After Auto Arrange:



Circuit and Segment Data Fields

The third basic section of the main design window consists of the data input section. Here, users can define their design on both the Circuit and Segment levels. The tabs on the left contain input fields for variables on the **Circuit** level while the tabs on the right contain input fields for variables that can be changed at the **Segment** level.

The data input section is comprised of different types of input controls. The following is a listing of all the types used in CompuTrace Design Suite:

Definition of Input Control Terms:

Text Box: A control used to accept typed entry from the user. The maintenance temperature field is an example of a Text Box.

Drop Down List: A control that presents the user with a list of options when clicked. The insulation type control is an example of a Drop Down List.

List Box: A control that presents a list of options for the user to select. It may include scroll bars if the options do not fit in the available space. The Product List is an example of a List Box.

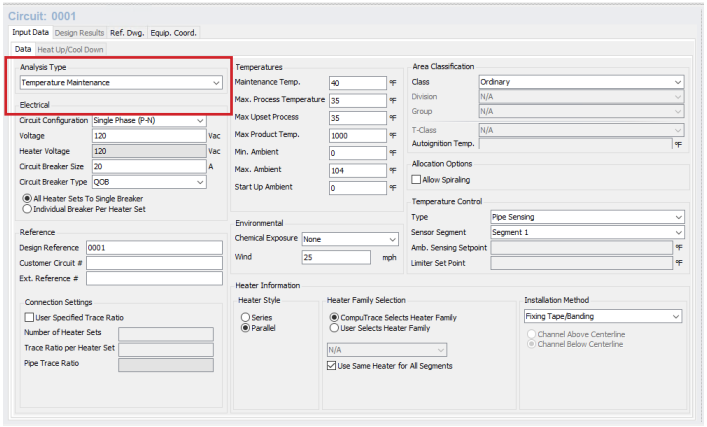
Check Box: A control that allows the user to either select or unselect a single option by clicking it. When the option is selected, a check appears in the box. The Multiple Segments input is an example of a Check Box.

Radio Button: A control that allows the user to select from several, mutually exclusive options. The options are grouped together where only one can be selected at a time. The options presented in the Analysis Type group are an example of Radio Buttons.

Circuit Data Fields

Input Data - Data Tab

Analysis Type Group



This dropdown list is used to define the type of analysis CompuTrace is to perform.

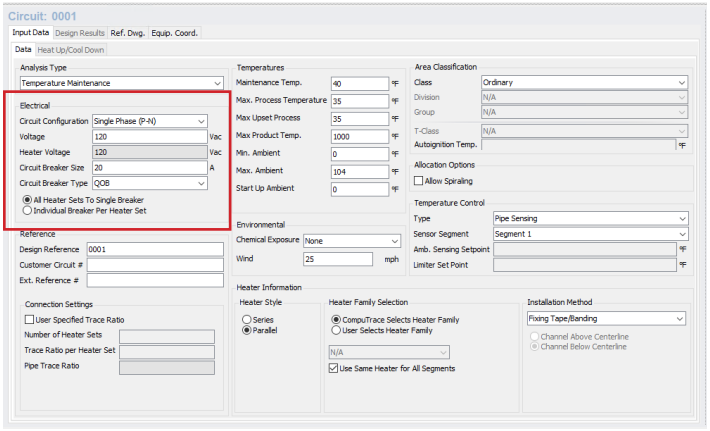
Temperature Maintenance: When selected, CompuTrace will use the input data provided by the designer to determine the heat loss from the pipe(s) at the specified maintenance temperature. A suitable heater will then be selected to offset this heat loss. If different pipe sizes and/or insulation values are specified for multiple segments, then the temperature for each segment will be calculated. If the selected or user specified heater does not meet the requirements for all pipe segments, then a warning will be generated.

Heat Up: This analysis begins with a temperature maintenance analysis, then determines the time required to heat the pipe(s) from the user specified initial temperature to the specified final temperature.

Cool Down: This analysis determines the time required for the pipe(s) to cool from the user-specified initial temperature to the specified final temperature. This calculation is performed without consideration of any trace heating.

NOTE: Heat up and Cool Down analysis are performed under non-flowing conditions.

Electrical Group



Circuit Configuration: Drop down list used to specify the electrical configuration of the heater. This information is used to determine the voltage across the heater circuit and how the amperage is calculated.

> Single Phase (P-N): Indicates that the value selected or entered in the Voltage field is a phase-to-neutral voltage.
>
> Phase to Phase: This selection indicates that the value selected or entered in the Voltage field is a phase-to-phase voltage.
>
> Three Phase Wye: This selection is applicable for single conductor series resistance heaters and three conductor series resistance heaters.

Voltage: Text box for entering the supply voltage for the circuit.

Heater Voltage: Displays the potential across the heater circuit based on the specified applied voltage and circuit configuration. For example, if the user enters 480 volts and 3-phase wye connection, then this field would display 277. If the entry in the voltage field is 240 volts and the circuit configuration is Single Phase or Phase to Phase, then this field would display 240.

Circuit Breaker Size: Text box for specifying the size of the circuit breaker that will be used to supply the heating circuit. This entry is used to determine whether a single breaker can supply the specified circuit. If the user has selected ATEX or IECEx Zones under the Electrical Codes and Standards menu, then the program will default to a 16 amp breaker. Otherwise, the default value is 20 amps.

Circuit Breaker Type: Drop down list used to specify the breaker type.

B type breakers have a trip curve based on 3 to 5 times the nominal amperage.

C type breakers have a trip curve based on 5 to 10 times the nominal amperage.

NV type breakers have a trip curve based on 7 to 10 times the nominal amperage.

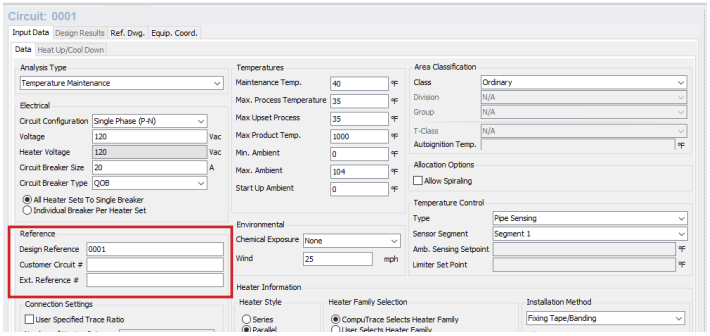
QOB type breakers have a trip curve based on 10 to 20 times the nominal amperage.

All heater sets to single breaker: When selected, the loading of the breaker is determined based on the amperage associated with all heater sets in the circuit. This is the default setting for all circuits and mandatory for Power Tees, and Power Splices.

Individual breaker per heater set: When selected, the loading of the breaker is determined based on the amperage associated with each individual heater set. This selection will yield longer maximum lengths when multiple passes or sets are involved. This option will ONLY be available when the user is specifying the number of passes. By default, all heaters in a heat tracing circuit will go to the same circuit breaker. If the circuit requires the heaters to be fed from individual breakers then an asterisk will be appended to the end of the circuit name in the Circuit Manager tree.

NOTE: Refer to **Trace Ratio per Heater Set** in this manual for more information on multiple passes.

Reference Group

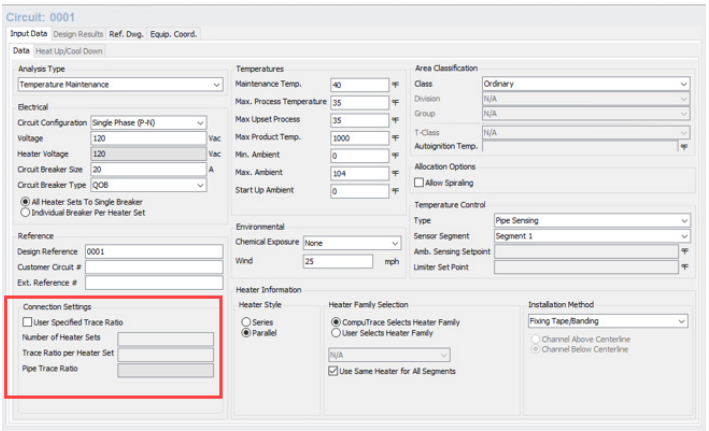


Design Reference: Enter the design name or number. This is the reference that will appear in the Circuit Manager and on reports. All project circuits must have a unique Design Reference.

Customer Circuit #: This is an optional informational field tied to a circuit. The intended use is so that the user can map a circuit in CompuTrace with a circuit on their line list.

Ext. Reference #: This is an optional informational field for the user to designate as a reference value.

Connection Settings Group



It is important to note that Series heaters default to Series passes and Parallel heaters default to Parallel passes.

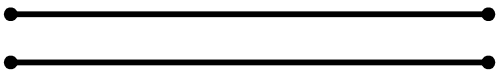
User Specified Trace Ratio: If checked, CompuTrace will attempt to design the segment using the specified number of passes rather than calculating the number of passes required.

Number of Heater Sets: Text box that will be enabled if the user selects User Specified Trace Ratio allowing the user to define the exact number of heater sets to be used.

Trace Ratio per Heater Set: Text box that will be enabled if the user selects User Specified Trace Ratio. It is used to specify the number of passes per heater set. If multiple passes per set is specified, then the passes are calculated to be in series with each other regardless of the selected Circuit Configuration in the Electrical Group on the Input Data tab. The Trace Ratio per Heater Set can be greater than 1 for parallel resistance heaters if the connection is made up of one segment. It cannot be greater than 1 if the connection has more than one segment.

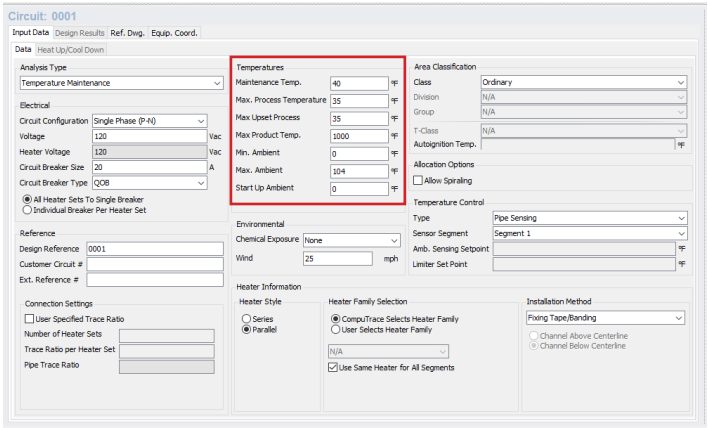
For example, the following diagram shows one heater set with a trace ratio per heater set of two (two passes in series):

The following would be two heater sets with a trace ratio per heater set of one (two passes in parallel):



Pipe Trace Ratio: This field will display the trace ratio per unit length of process pipe based on the entries in the Number of Heater Sets and Trace Ratio per Heater Set fields. In both of the previous diagrams, the pipe trace ratio is two.

Temperatures Group



Maintenance Temp.: Text box for specifying the temperature that the heating system is to maintain the process pipe. This value is used to determine the heat loss from the pipe.

Max. Process Temperature: This is the maximum temperature the pipe will be exposed to during normal operation. No entry is required if the maximum Process Temperature is less than the maintenance temperature or the maximum ambient temperature.

With ambient sensing, ambient proportional or no control, the cable is assumed to be energized while exposed to the Maximum Process Temperature.

With pipe sensing control or control limited designs, the cable is assumed to be de-energized when the pipe is at the Maximum Process Temperature.

Exposures which occur for less than a cumulative 1000 hours over the life of the installation are defined as intermittent and should be entered in the Max Upset Process field.

Max Upset Process: This is the maximum temperature the pipe will experience due to process upset conditions or steam out. No entry is required if the maximum upset temperature is less than the maintenance temperature or the maximum ambient temperature. It is assumed the exposure to this temperature is intermittent if the total cumulative exposure is less than 1000 hours. For example, HTSX may be exposed to temperatures greater than 204 °C but less than 250 °C when energized as long as the exposure is less than 1000 hours over the life of the installation. This temperature is used to select a suitable heater for the application. It also impacts maximum heater temperature calculations.

With ambient sensing, ambient proportional or no control, the cable is assumed to be energized while exposed to the temperature entered in this field.

With pipe sensing control or control limited designs, the cable is assumed to be de-energized when the pipe is at this temperature.

Max Product Temp.: This is the maximum allowable temperature the fluid in the pipe may be exposed to without degradation. This temperature is used to evaluate heating system designs to ensure that the applied system will not overheat the process fluid. The program will reject designs with no control or ambient sensing control selected if the Minimum

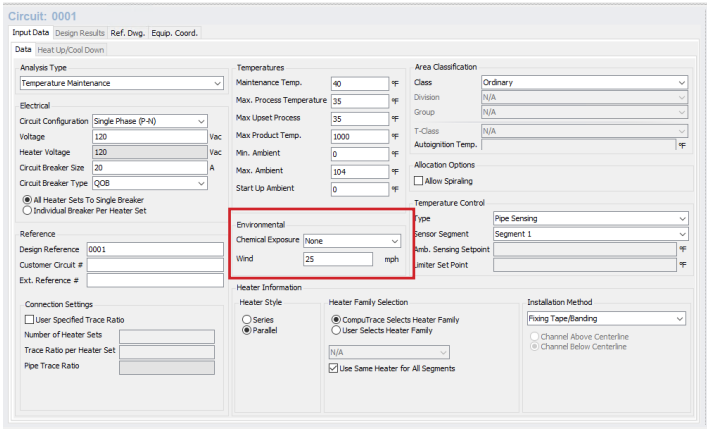
Controlled Temperature exceeds the maximum product temperature. NOTE: Finite Element Analysis (FEA) is required to predict temperature distribution around and through the pipe and product.

Min. Ambient: Text box for specifying the minimum ambient temperature. This temperature is used to determine the heat loss from the process pipe and should represent the expected minimum temperature, not the historical low.

Max. Ambient: Text box for specifying the maximum ambient temperature. This value is used to determine the maximum heater temperature for designs in nonhazardous (ordinary) locations. This entry is also used to determine the stabilized operating pipe temperature for systems designed without pipe sensing control.

Start Up Ambient: Text box for specifying the temperature to be used when determining the maximum amperage associated with the startup of the heating system.

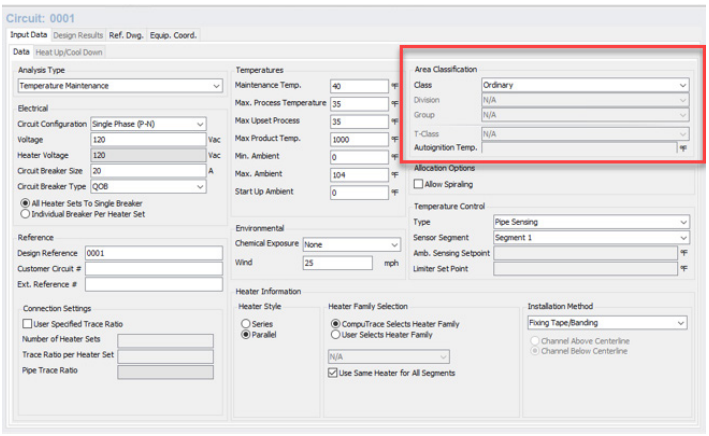
Environmental Group



Chemical Exposure: A drop down list containing chemical exposure options (None, InOrganics, Organics). The selection will affect the selection of jacket material for some heaters.

Wind: Text box for accepting the user specified wind speed. This value will be used in determining the design heat loss for the pipe segments.

Area Classification Group



This list consists of the classification of the plant area where the heat tracing will be installed.

Class: The entries in this list will vary depending on the user selection under the Electrical Codes & Standards in the Set Project Units menu. The selected entry will be applied in determining the maximum pipe and heater sheath temperatures.

Division: The entries in this list will vary depending on the user selection under the Electrical Codes & Standards in the Set Project Units menu.

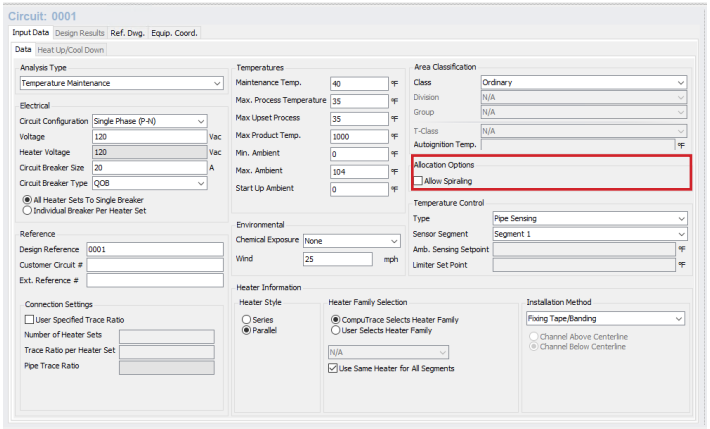
Group: The entries in this list will vary depending on the user selection under the Electrical Codes & Standards in the Set Project Units menu.

T-Class: Drop down list containing the available T-Class options. The content of the list is dependent on the user selection under the Electrical Codes & Standards in the Set Project Units menu.

Autoignition Temp.: Text box for user specified autoignition temperature (AIT). This option is not available when ATEX and IECEx Electrical Codes are specified.

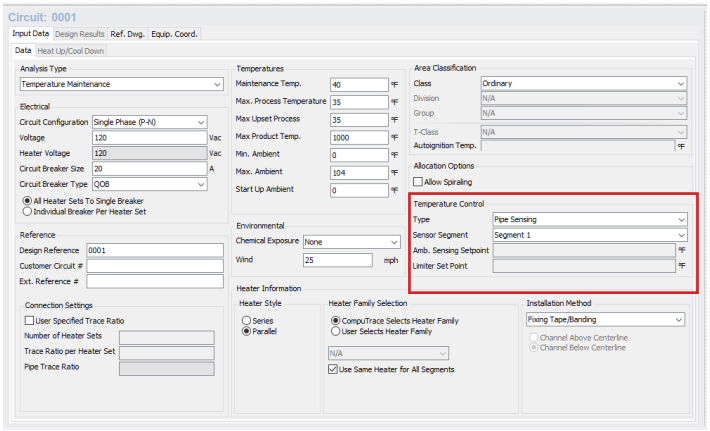
Selections in this group will impact both design calculations and component selection.

Allocation Options Group



Allow Spiraling Checkbox: If checked, CompuTrace will determine an optimum spiral pitch to most closely match the heater and the heat load. The maximum spiral ratio is determined during the initial project setup and cannot be altered; the maximum spiral ratio set for the project can be viewed set by selecting Preferences > Project Properties. Spiraling is only allowed for parallel heater style. This option will return an error message if series resistance heater style is selected.

Temperature Control Group

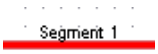


Type: A drop down list containing the control options:

Ambient Sensing: When selected, the Sensor Segment drop down list and Limiter Set Point field are disabled. The Ambient Sensing Setpoint field is enabled. The program will determine the stabilized operating pipe and heater temperatures for each segment of pipe in the circuit under Minimum ambient conditions and for the condition when the ambient temperature is equal to the ambient sensing setpoint. These results will provide an operating temperature range for the pipes in the circuit. The low ambient results, Min Controlled Temperature and Heater Sheath Temperature, will be based on user specified wind and nominal heater output. The calculations performed at the ambient setpoint, Max Controlled Temperature and Heater Sheath Temperature Hi, will also be based on nominal output but will assume no wind.

Pipe Sensing: When selected, the Sensor Segment drop down list will be enabled and the Ambient Sensing Setpoint and Limiter Set Point fields will be disabled. Based on the heat loss and applied power on the pipe indicated by the Sensor located on this segment, CompuTrace will determine the operating temperature for the other pipe segments in the circuit.

NOTE: Hazardous area requirements for designs using this type of control will be validated using stabilized design calculations.

On the Segment Definition pane, the segment chosen to be pipe-sensed, will be depicted with a red line instead of grey: 

Controlled Design (w/o Limiter): Similar to pipe sensing control in that an individual temperature controller, with a single sensor, is used for each heat tracing circuit. This same sensor is used to limit the maximum pipe temperature. This type of control is suitable for use in Zone 2 and Division 2 classified areas to meet T-rating requirements.

Controlled Design (w/Limiter): A type of pipe sensing control in which a separate controller and limiter device are utilized for each heat tracing circuit. The limiter limits the maximum pipe temperature by de-energizing the tracing circuit when the Limiter Set Point is exceeded. The tracing circuit cannot be re-energized until the user acknowledges the condition. This type of control is suitable for use in Zone 1 & 2 as well as Division 1 & 2 classified areas to meet T-rating requirements.

NOTE: Mechanical thermostats do not meet the requirements for Controlled Design with or without a limiter.

Ambient Proportional Control: A modified version of Ambient Sensing Control. When selected, the Sensor Segment drop down list and Limiter Set Point field are disabled and the Ambient Sensing Setpoint field is enabled. CompuTrace will determine the stabilized operating pipe and heater temperatures under low ambient conditions and when the ambient is equal to the ambient sensing setpoint. The stabilized temperatures under Minimum ambient conditions will be determined using nominal heater output, user specified wind, and 100% heater output. The stabilized temperatures for the ambient conditions equal to the ambient setpoint temperature will be determined assuming no wind, nominal heater output reduced by a proportioning value that can be as low as 20% depending on the values specified. Circuits designed with this type of control are controller-dependent limited.

No Control: When selected, all other input fields in this group (Sensor Segment, Ambient Sensing Setpoint and Limiter Set Point) are disabled. CompuTrace will determine the minimum and maximum controlled (stabilized) operating temperatures for the pipes in the circuit. The lower controlled (stabilized) operating pipe and heater temperatures will be determined based on minimum ambient temperature, wind, and heater output. The upper controlled (stabilized) operating pipe and heater temperature will be determined using the user specified maximum ambient temperature, no wind, and nominal heater output.

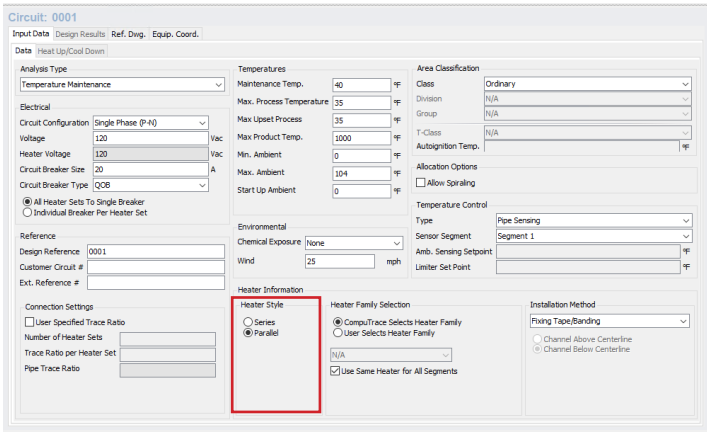
Sensor Segment: Drop down list box used to specify which pipe segment in the heat tracing circuit will have the temperature sensor on it. CompuTrace will calculate the temperature of all other pipe segments in the circuit based on the segment on which the sensor is placed.

Ambient Sensing Setpoint: Text box for accepting the ambient temperature below which the heat tracing circuit will be energized. This field is enabled only when Ambient Sensing or Ambient Proportional Control is selected.

Limiter Set Point: Text box for accepting the user specified temperature above which the heat tracing will be de-energized. This field is only enabled if the Control Design (w/Limiter) option is selected. CompuTrace will use this value as the max pipe temperature when determining the suitability of a design for classified areas.

Heater Information

Heater Style Group

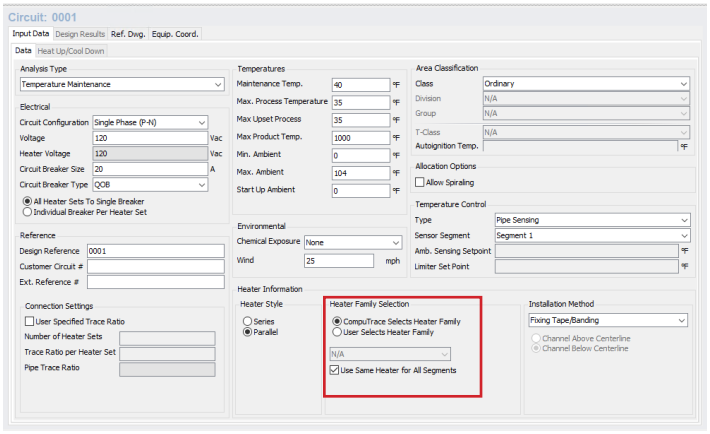


This group defines whether series or parallel heaters will be used in the design of the heat tracing circuit and which heater families will be presented in the Heater Family Selection field.

Series: Typically used where circuit lengths exceed the limits of parallel resistance heating cables or the design conditions dictate the use of mineral insulated cables. Applications that require high maintenance temperatures or exposure temperatures are best suited for Thermon's MIQ series resistance heating cables.

Parallel: Typically used for freeze protection and process maintenance applications for complex piping.

Heater Family Selection Group



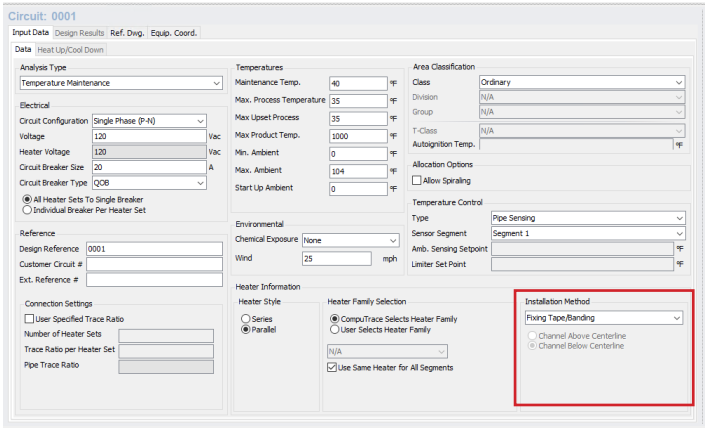
CompuTrace Selects Heater Family: Default for parallel heaters only. User MUST define family for series resistance heaters. CompuTrace will determine the proper heater for the design based on the user-defined requirements. The **Heater Family** drop down list is disabled.

User Selects Heater Family: When selected, CompuTrace will attempt to generate a suitable design for the specified conditions using the user-selected heater family.

Heater Family: This drop down list is enabled if **User Selects Heater Family** is selected, allowing the user to select a specific heater to be considered for the current circuit. If the user selects an inappropriate heater, a warning will be presented.

Use Same Heater for All Segments Check Box: When checked, the same heater will be used for all segments in the circuit. If unchecked, then the program, or designer, can change the heater from one segment to the next using the Segment Data Fields – Input Data – Data Tab in the Heater group. There are certain segment types, such as continuation, that require the use of the same heater as the preceding segment.

Installation Method Group



Select from this list the method of attachment for the heater. Not all selections are valid for all heaters. If an invalid selection is made for the type of heater selected, then the default attachment method will be used for design and Bill of Materials considerations.

Fixing Tape/Banding: CompuTrace will select the proper fixing tape or banding based on the selected heater.

Aluminum Tape: CompuTrace will calculate the necessary amount of aluminum tape for parallel coverage of each heater pass. This selection will also adjust the heat transfer coefficient between the heater and process pipe, which will modify the heater output, temperatures, and associated amperage. This option is not available for all design applications, temperature limits for the aluminum tape are considered in the design.

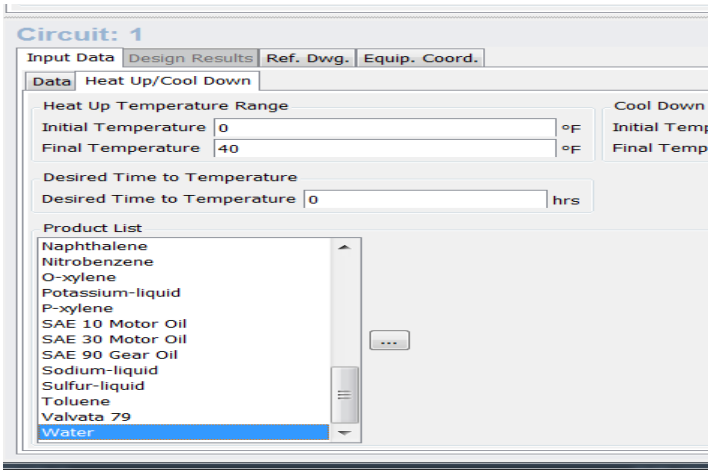
Heat Transfer Compound (MIQ Only): CompuTrace will select the proper heat transfer compound based on the temperatures associated with the MIQ design. This selection will also adjust the heat transfer coefficient between the heater and process pipe, which will modify the heater output, temperatures, and associated amperage.

Channels: CompuTrace will calculate the heater output and temperatures assuming the heaters have been pulled through channels and do not have any physical attachment to the process pipe. This will affect the calculation of the heater output, amperage, and temperatures.

When Channels are selected as the installation method, the user can then select whether the channel is installed above or below the centerline of the pipe. This installation method is not allowed for Series cables.

Input Data - Heat Up/Cool Down Tab

This tab will only be enabled if the Analysis Type is set to either Heat Up or Cool Down.



Heat Up Temperature Range

This group will be enabled only if the analysis type is set to Heat Up.

Initial Temperature: Text box for specifying the temperature of the pipe and product at the beginning of the heat up.

Final Temperature: Text box for specifying the temperature at the pipe and product at the end of the heat up.

Cool Down Temperature Range

This group will be enabled only if the analysis type is set to Cool Down.

Initial Temperature: Text box for specifying the temperature of the pipe and product at the beginning of the cool down.

Final Temperature: Text box for specifying the temperature at the pipe and product at the end of the cool down.

Desired Time To Temperature

Desired Time to Temperature: Text box for specifying the desired time to achieve the specified heat up or cool down. If the resulting time is greater than the specified desired time a warning message will be provided in the design result window when Heat Up analysis is selected.

Product List

Product List: List box containing process fluids. When the user selects a process fluid from the Product List, CompuTrace will indicate whether or not the change of phase occurs as well as the temperature at which the phase change happens and the associated Latent Heat.

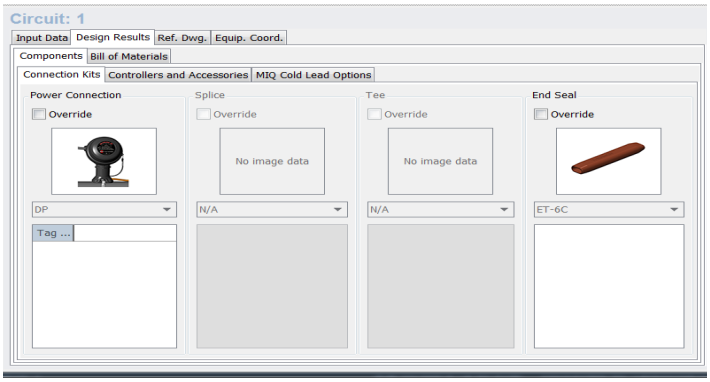
The list can be extended or modified by clicking the [...] button; which opens a form for viewing properties for each product listed and provides access for editing/adding products. Refer to **Creating/Editing Lists in CompuTrace** in this manual for details.

The results of the Heat Up or Cool Down calculations for each segment are displayed in a graph of Time vs. Temperature on the Segment->Design Results->Heat Up/Cool Down Graph tab.

Design Results – Components Tab

This tab is where the user will define the connection kits, temperature controllers, and other accessories to be included in the Bill of Materials once a circuit has been designed. CompuTrace will show all components for the currently selected circuit. Each circuit will have, at a minimum, a power connection kit and end termination kit. If a segment defined as a Tee splice or splice is included in the circuit, then the program will enable the appropriate group and indicate the default option based on design conditions and approvals requirements. This section of data can only be edited for a validated circuit, and will be cleared if the circuit settings are invalidated.

Connection Kits Tab



Power Connection Group

Override Power Kit: Defaults to unchecked. When checked, the user may select from the Power Kit drop down list beneath the check box to select a different Power Connection Kit.

Power Kit: Indicates the Power Connection Kit being used. In most cases, the default will be a member of the Terminator family of kits. If the user checks the Override Power Kit check box, then this list will be populated with all kits that are appropriate for the selected design conditions and approvals requirements.

The image box will display a general representation of the selected component.

Tag Number: CompuTrace will indicate the number of Power Connection Kits associated with the selected circuit and allow the user to enter a tag number for each box. Limit will be 10 per circuit.

Splice Group

If the circuit includes Splices, then this group will be enabled.

Override Splice Kit: Defaults to unchecked. When checked, the user may select from the Power Kit drop down list beneath the check box to select a different Splice Kit.

Splice Kit: Indicates the Splice Kit being used. In most cases, the default will be a member of the Terminator family of kits. If the user checks the Override Splice Kit check box then this list will be populated with all kits that are appropriate for the selected design conditions and approvals requirements.

The image box will display a general representation of the selected component.

Tag Number: CompuTrace will indicate the number of Splice Kits associated with the selected circuit and allow the user to enter a tag number for each box. Limit will be 10 per circuit.

Tee Group

If the circuit includes Tee Splices, then this group will be enabled.

Override Tee Kit: Defaults to unchecked. When checked, the user may select from the Tee Splice Kit drop down list beneath the check box to select a different Tee Splice Kit.

Tee Kit: Indicates the Tee Splice Kit being used. In most cases, the default will be a member of the Terminator family of kits. If the user checks the Override Tee Kit check box then this list will be populated with all kits that are appropriate for the selected design conditions and approvals requirements.

The image box will display a general representation of the selected component.

Tag Number: CompuTrace will indicate the number of Tee Splice Kits associated with the selected circuit and allow the user to enter a tag number for each box. Limit will be 10 per circuit.

End Seal Group

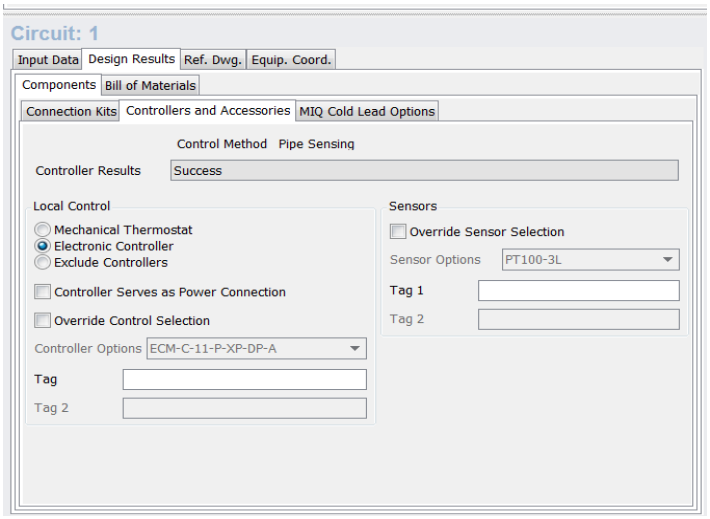
Override End Seal Kit: Defaults to unchecked. When checked, the user may select from the End Termination Kit drop down list beneath the check box to select a different End Termination Kit.

End Termination Kit: Indicates the End Termination Kit being used. In most cases, the default will be a member of the Terminator family of kits. If the user checks the Override End Termination Kit check box then this list will be populated with all kits that are appropriate for the selected design conditions and approvals requirements.

The image box will display a general representation of the selected component.

Tag Number: CompuTrace will indicate the number of End Termination Kits associated with the selected circuit and allow the user to enter a tag number for each box. Limit will be 10 per circuit.

Controllers and Accessories Tab



Control Method: A text box displaying the control method used in the design of the selected circuit. Reflects the value from the Temperature Control Group on the Setting tab.

Controller Results: A text box displaying the results status based on the chosen control method used.

Local Control Group

Mechanical Thermostat: Design will include mechanical thermostats that are available for the design based on the specified electrical codes and standards requirements.

Electronic Controller: Design will include electronic controllers. Only single point controllers are available at this time.

Exclude Controllers: Controllers of any type will not be included in the Bill of Materials.

Controller Serves as Power Connection: This option allows user to omit the power connection box from the BOM, and should only be checked if the controller is approved to be used as the power connection for the product and area classification indicated for the circuit.

Override Control Selection: Checking this box will enable the Controller Options drop down list

which will be populated with the mechanical thermostats or single point electronic controllers appropriate for the design conditions and approval requirements.

Tag 1 & Tag 2: Reference fields to identify individual controllers by number.

Sensors Group

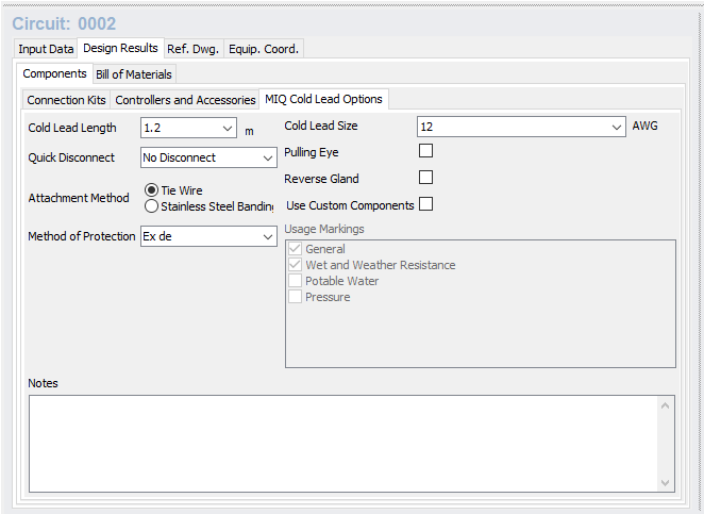
If an electronic controller is selected then the Sensors group will be enabled. CompuTrace will select an RTD based on the specified Electrical Codes and Standards and controller type.

Override Sensor Selection: Selecting this check box will enable the Sensors Options drop down list. When the user selects this box, CompuTrace will populate the list with all valid options based on the specified controller and Electrical Codes and Standards.

Sensor Options: Drop down listing of all available sensors based on current user design.

Tag 1 & Tag 2: Reference fields to identify individual sensors by number.

MIQ Cold Lead Options Tab



Cold Lead Length: Select from the list, the length of cold lead for the heater segment. The list will display the standard lengths by default but the user may enter a length between 4 ft. (1.2 m) and 25 ft. (7.6 m)

Quick Disconnect: Select from the list whether a quick disconnect is to be included and if so, what type.

Attachment Method: Method in which the MIQ heater will be attached to the object being heat traced.

Method of Protection: (Only available for projects using Zones classification method) The user may select the type of protection which can impact the MIQ part number and components.

Notes: Additional comments for the user to enter in regards to the MIQ heater design.

Cold Lead Size: Select from the list the desire gauge for the cold lead. CompuTrace will validate the selection against the circuit amperage to confirm that the cold lead is large enough.

Pulling Eye: Check the box if a pulling eye is to be included.

Reverse Gland: Check the box if a reverse gland is to be included.

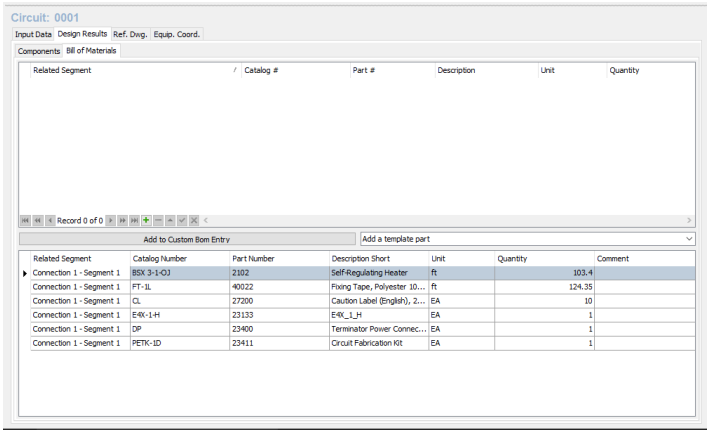
Use Custom Components: Check the box if custom components are to be included.

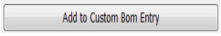
Usage Markings: For information only.

Design Results – Bill of Materials Tab

The Bill of Materials tab allows the user to customize the items and quantities designated for each specific heater segment, but with the added ability to choose items from the design-generated BoM or from a list of template parts. Note template parts can only be created during the project configuration setup when the project is created.

This section also allows the user to view the items and quantities required for the entire circuit in a bill of materials window at the bottom of the screen.



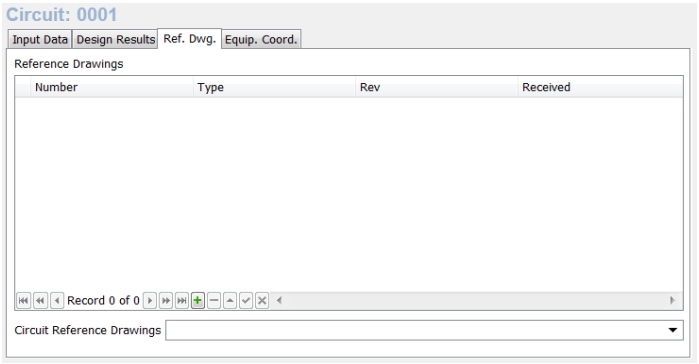
To add to the quantity of an item in the design BoM, highlight it and click 

. The additional quantity entered in the table will be added to the design BoM quantity for the BoM reports. Note the quantities of heater cable cannot be altered for the custom BoM since critical design results such as heater output, loading and cable temperatures are based on the actual design length of the heater cable.

To add a new custom part, first click the ➕ button at the bottom of the table. This will enter a new row in the table and the new part can be defined. Once defined, to enter the selection click the ☑ button at the bottom of the table.

Reference Drawings

This table is available for entering the reference drawings used for the selected circuit. This information is presented on several of the report formats.

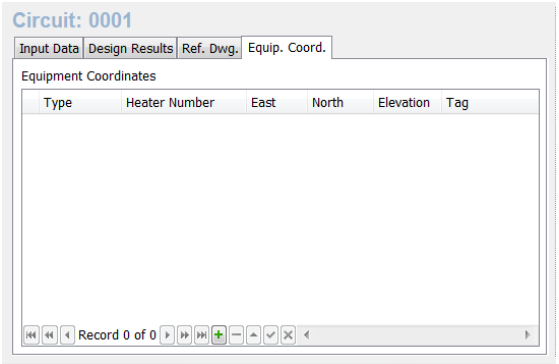


To add a reference drawing for a specific circuit, the user must first click the ➕ button at the bottom of the table. This will enter a new row in the table. The user can enter the drawing number, revision and date received, and use a drop down list to choose the drawing type.

To enter the selection(s) when finished, click the ☑ button at the bottom of the table.

Equipment Coordinates

This table is available for entering the heat tracing system equipment coordinates used for the selected circuit. This information is available in the Equipment Coordinates.



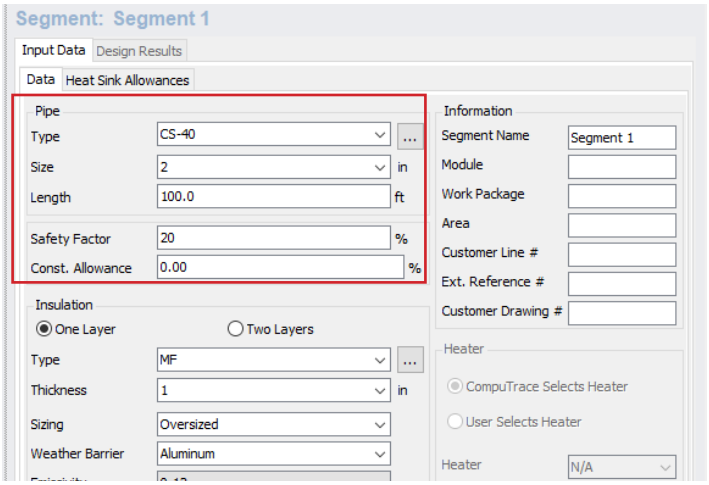
To add coordinates for equipment for a specific circuit, the user must first click the ➕ button at the bottom of the table. This will enter a new row in the table. The user can use the drop down list to choose the equipment type, and enter the heater number, coordinates and tag number.

Once defined, to enter the selection click the ☑ button at the bottom of the table.

Segment Data Fields

Input Data – Data Tab

Pipe Group



Type: This drop down list contains the pipe specifications that are available for this project. CompuTrace will provide some standard pipe specifications but the user may add specifications that can be saved and presented in this list.

The list can be extended or modified by clicking the … button; which opens a form for viewing properties for each product listed and provides access for editing/adding products. Refer to Creating/Editing Lists in CompuTrace for details.

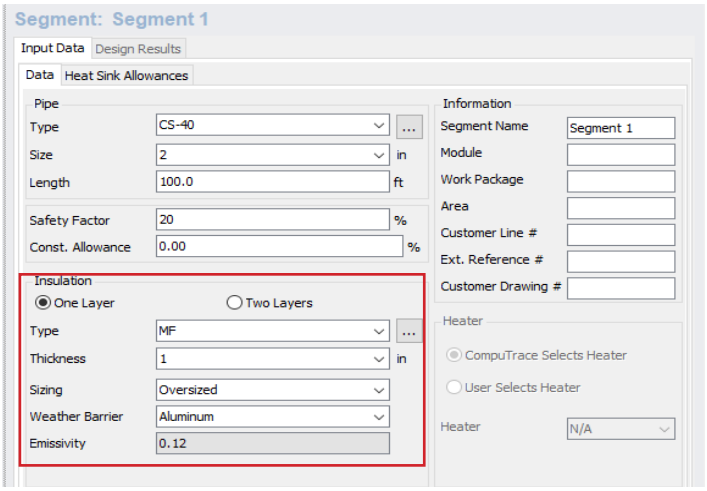
Size: This drop down list is populated from the options provided in the Pipe Type.

Length: The length of the segment in the user specified unit of feet or meters.

Safety Factor: Text box for user specified heat loss safety factor. This entry is used to increase the theoretical heat loss when sizing the heat tracing system. For freeze protection and low temperature maintenance applications, a minimum value of 20% is recommended. Higher values are recommended for higher maintenance temperatures.

Default Construction Allowance: Enter a percentage of the design length that should be added to account for field variances and routing. The allowance will be added to the overall heater length when determining heater output, electrical loading and BoM quantities. For MIQ consider at least a 3% allowance to permit 'snaking' of MIQ on straight pipes.

Insulation Group



One Layer/Two Layers Selection: This control enables the user to select either One Layer or Two Layers of insulation for a Segment design. The default option is One Layer insulation.

Type: This drop down list contains the insulation type specifications that are available for this project. CompuTrace will provide many insulation types based on the ASTM standards as well as some based on European norms (See Editing Lists in CompuTrace for a representative sampling of the insulation types).

The list can be extended or modified by clicking the … button; which opens a form for viewing properties for each product listed and provides access for editing/adding products. Refer to Creating/Editing Lists in CompuTrace in this manual for details.

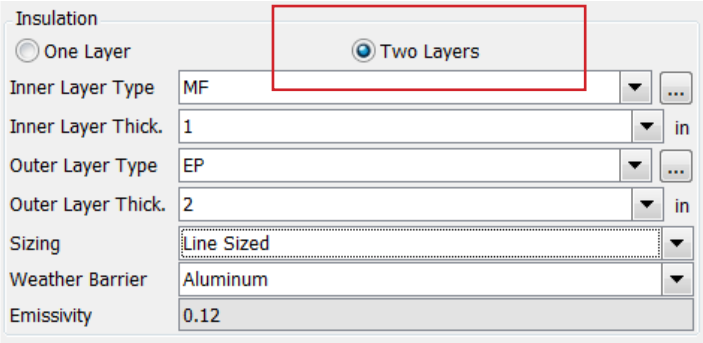
Thickness: Drop Down list contains the insulation thickness values defined for the selected insulation type.

Sizing: The list will allow the user to select line sized, oversized, or pre-insulated pipe insulation. If line sized is selected, then the program will calculate the heat loss based on the insulation size that matches or most closely matches the selected pipe size. If oversize is selected, then the program will calculate the necessary insulation size based on the size and number of passes of heaters that are being used. The last option, pre-insulated pipe refers to pipe that has been prefabricated with insulation sections already attached to it. If this last option is chosen, then the **Channels** option must be selected as the Installation Method (Circuit – Input Data - Heater Information - Installation Method).

Weather Barrier: Drop down list that allows the user to select the cladding material. This selection will specify the emissivity of the surface, which is used in the heat loss calculation.

Emissivity: Text box used to display the effective emissivity for the selection in the Weather Barrier drop down list. This field is read-only and does not accept manual input.

If Two Layers of insulation is selected...



The user must now specify the type and thickness of the inner and outer insulation layers for designs using composite insulation.

Inner Layer Type: Defines the insulation material of the inside layer of the design. This value is equivalent to its 'Type' counterpart when One Layer insulation is selected.

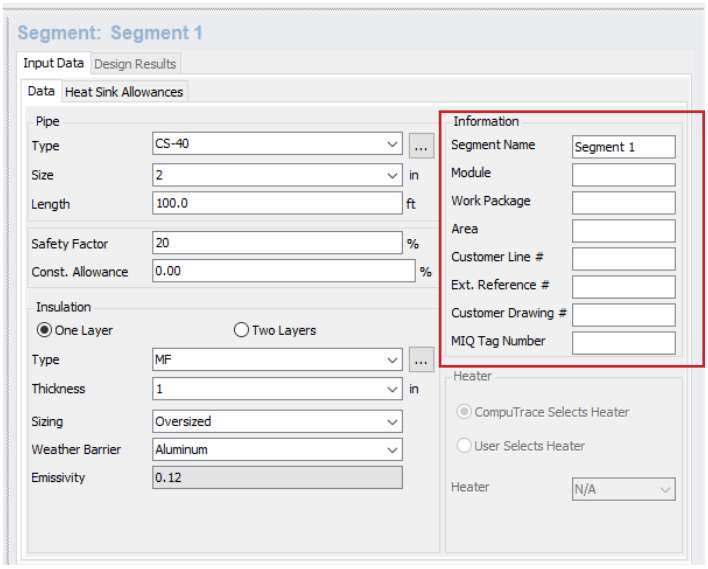
Inner Layer Thick.: Defines the thickness of the inner layer of the insulation material. This value is equivalent to its 'Thickness' counterpart when One Layer insulation is selected.

Outer Layer Type: Defines the insulation material of the outer layer of the design.

Outer Layer Thick.: Defines the outer layer thickness of the insulation material.

Both Inner and Outer Type lists can be extended or modified by clicking the [...] button; which opens a form for viewing properties for each product listed and provides access for editing/adding products. Refer to **Creating/Editing Lists in CompuTrace** in this manual for details.

Information Group



Segment Name: This is the name assigned to a given segment at creation time. It must be unique within a given circuit, but the user can modify it here.

Module: This is an optional informational field tied to a segment. This field can be used as a key to filter the summary bill of materials by module.

Work Package: This is an optional informational field tied to a segment. This field can be used as a key to filter the summary bill of materials by work package.

Area: This is an optional informational field tied to a segment. This field can be used as a key to filter the summary bill of materials by area.

Customer Line #: This is an optional informational field tied to a segment. The intended use is so that the user can map a pipe segment in CompuTrace to a pipe segment on their line list.

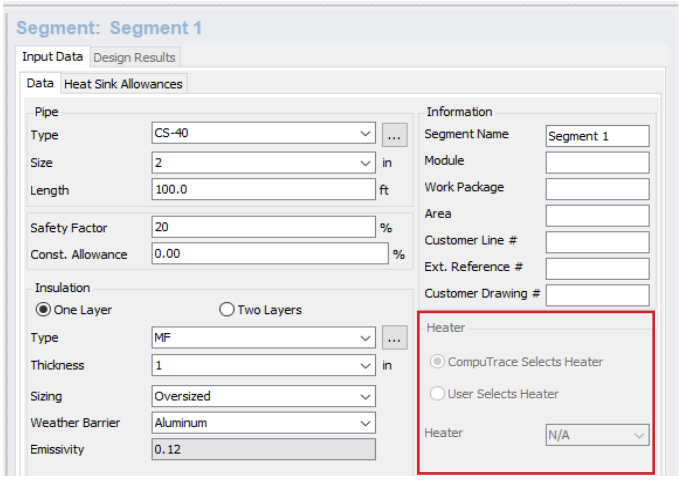
Ext. Reference #: This is an optional informational field to be used as an alternate reference by the user.

Customer Drawing #: This is an optional informational field tied to a segment. The intended use is so that the user can map a pipe segment in CompuTrace to a pipe segment on their drawing list.

MIQ Tag Number: This field is only presented when the design utilized MIQ heaters. This is an optional informational field.

Heater Group

This group is only enabled if the **Use Same Heater for All Segments** checkbox is unchecked on the Circuit - Input Data – Data - Heater Information - Heater Family Selection group. These fields allow the user to select specific heaters for the selected segment.

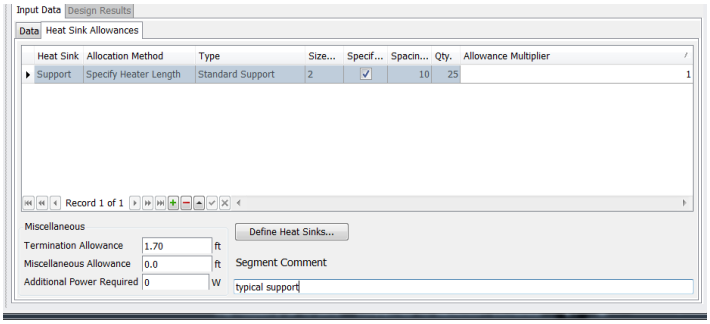


CompuTrace Selects Heater: When selected, CompuTrace will determine the proper heater for the design based on the user-defined requirements. The Heater drop down list is disabled.

User Selects Heater: When selected, CompuTrace will attempt to generate a suitable design for the specified conditions using the user-selected heater.

Heater: This drop down list is enabled if User Selects Heater is selected, allowing the user to select a specific heater to be considered for the current segment. If the user selects an inappropriate heater, a warning will be presented and corrective actions will be recommended.

Input Data - Heat Sink Allowances



This tab contains the fields where the designer can allocate additional heater length for valves, pumps, flanges and supports. Supports have the option of specifying a spacing or quantity. If the spacing is entered the program will calculate the number of supports on the segment.

To select a component, the user must first click the button at the bottom of the table. This will enter a new row in the table. Once the heat sinks are defined, to enter the selections click the button at the bottom of the table.

NOTE: CompuTrace Design Suite contains pre-defined heat sinks for the Specify Heater Length and Calculate Heat Loss allocation methods; the Calculate Heat Loss method is only valid for valves and supports. The Equivalent Pipe Length and Specified Power options are included for user defined heat sinks.

Heat Sink: Drop down list containing the basic types of heat sinks that have been defined for this project. CompuTrace will provide many default options but the list can be extended and modified by selecting the **Define Heat Sinks** button on this pane. Part of the definition for the components will be whether the heater allowance comes from a table (user editable) specifying either a heater length, power, equivalent pipe length, or a calculated heat loss.

Allocation Method: Drop down listing of the procedure to be used to calculate the extra heater allowance required based on the heat sink. These methods include "Specify Heater Length", "Specify Power", "Specify Equivalent Pipe Length" and "Calculate Heat Loss". The program comes preloaded with values which are user editable using the **Define Heat Sinks** button.

Type: Drop down list containing the specific types of heat sinks that have been defined for this project. CompuTrace will provide many default options but the list can be extended and modified by selecting the Define Heat Sinks button. Part of the definition for the components will be whether the heater allowance comes from a table (user editable) specifying either a heater length, power, equivalent pipe length, or a calculated heat loss. This field is available dependent on which selection is made from the Allocation Method and Type.

Size: Drop down listing of various pipe sizes appropriate for the heat sink selected. This field is available dependent on which selection is made from the Allocation Method and Type.

Specify Spacing: Check box that enables the user to specify the average distance between supports.

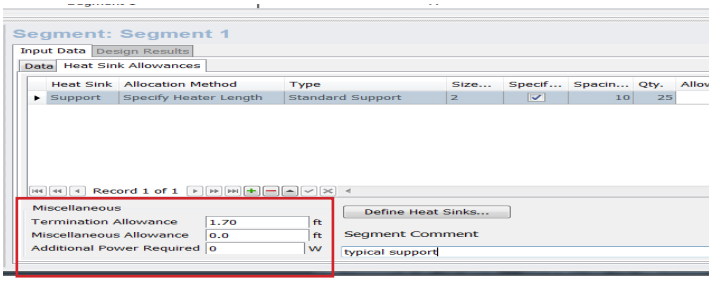
Spacing: User defined average distance between supports. This field is available if "Specify Spacing" is checked for supports.

Qty.: Text box for the user-specified quantity for the associated heat sink.

Allowance Multiplier: This value will be applied to the default allowance specified.

The lists can be extended or modified by clicking the Define Heat Sinks... button which opens a form for viewing properties for each product listed and provides access for adding new items or editing added items. Refer to Creating/Editing Lists in CompuTrace in this manual for details.

Miscellaneous Group



Termination Allowance: Specify the length of heater to be allocated for each power/end connection. This field is disable when Series Heater Style on the Circuit - Input Data – Data - Heater Information - Heater Style group

Miscellaneous Allowance: Specify a length of heater for each set to cover a miscellaneous need such as a flow meter or other equipment type not covered in the table above.

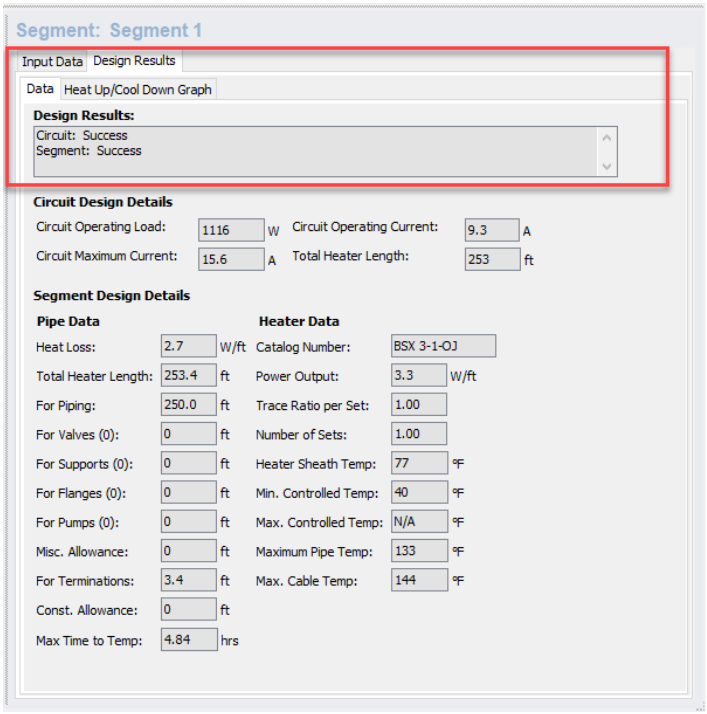
Additional Power Required: Specify any additional power requirements over and above the calculated heat loss that additional heater must be allocated for. This is another means of covering a piece of equipment not listed in the table above.

Segment Comment: Information that can be entered by the user for reference only

Design Results – Data Tab

At the end of the design calculations, CompuTrace will enable the Design Results tab showing the Circuit level and Segment level results for the selected circuit. If multiple circuits have been defined, then clicking on the circuit in the Circuit Manager pane will display the results for the selected circuit.

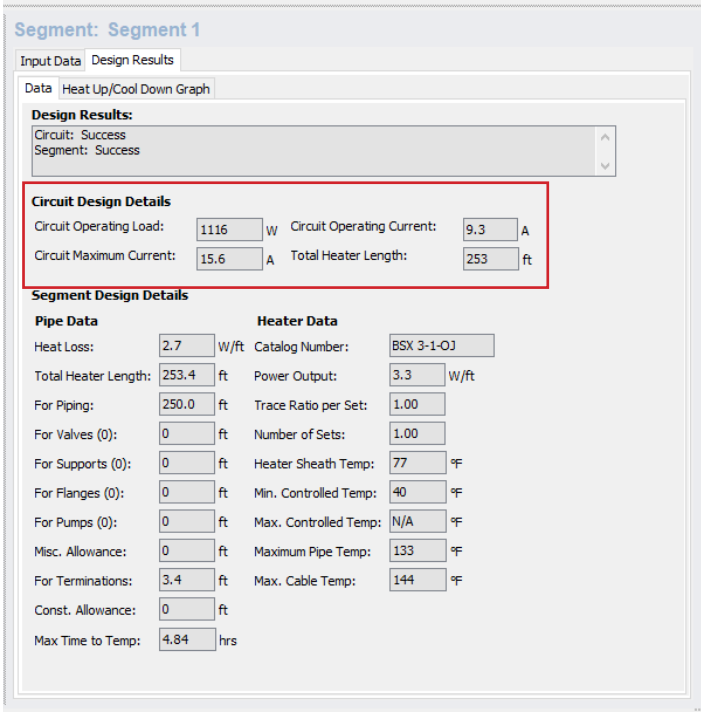
Design Results



Circuit: A successful design or an error message will be displayed based on the results of the calculations.

Segment: A successful design or an error message will be displayed based on the results of the calculations.

Circuit Design Details



Circuit Operating Load: The total applied power for all segments in the circuit.

Circuit Operating Current: The sum of the operating amperage for all segments in the circuit under operating conditions.

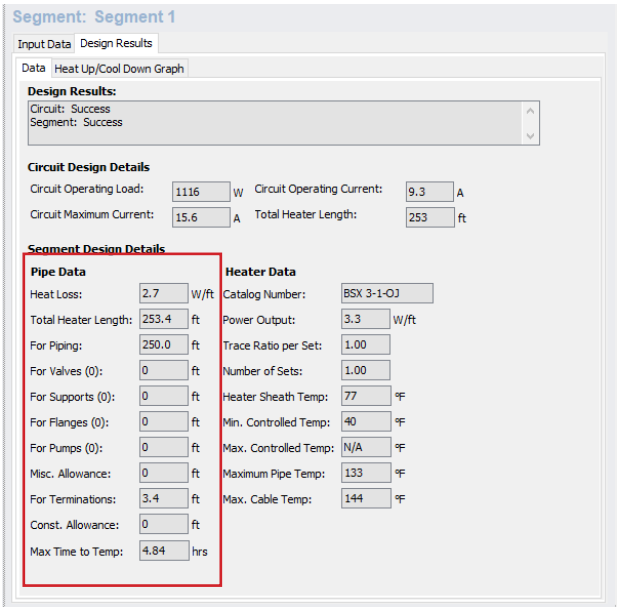
Circuit Maximum Current: The sum of the steady state current for all heaters in the circuit under startup conditions.

Total Heater Length: The sum of the entire length of heater for the designed circuit. This may include a mix of different heaters if the heater changes across segments.

Segment Design Details

CompuTrace will display the results for the current segment for both Pipe Data and Heater Data. If multiple segments have been defined, then clicking on the segment in the Segment Definition or Circuit Manager pane will display the pertinent results for the selected segment.

Pipe Data



Heat Loss: Displays the calculated heat loss, with safety factor, for the selected segment.

Total Heater Length: Lists complete length of heater for the selected segment.

For Piping: Length of heater for the selected segment that is allocated for pipe.

For Valves(x): Length of heater that is allocated to valves for the selected segment. (X) indicates the number of valves in the segment. Default number is 0.

For Supports(x): Length of heater that is allocated to supports for the selected segment. (X) indicates the number of supports in the segment. Default number is 0.

For Flanges(x): Length of heater that is allocated to flanges for the selected segment. (X) indicates the number of flanges in the segment. Default number is 0.

For Pumps(x): Length of heater that is allocated to pumps for the selected segment. (X) indicates the number of pumps in the segment. Default number is 0.

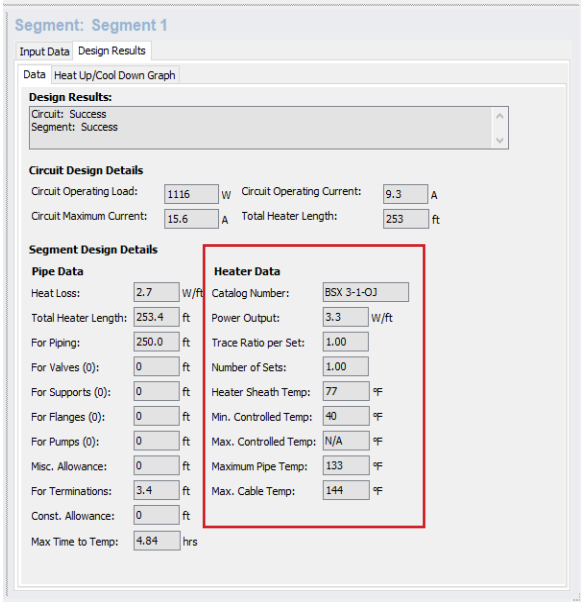
Misc. Allowance: Length of heater allocated to miscellaneous allowance for the selected segment.

For Terminations: Length of heater allocated to terminations for the selected segment.

Const. Allowance: Length of heater allocated to account for field variances and routing.

Max Time to Temp: The time in hours to achieve the specified heat up or cool down if the analysis type of Heat Up or Cool Down has been selected.

Heater Data



Catalog Number: Catalog number for the heater selected for the current segment.

Power Output: Output of heater selected for the current segment under normal operating conditions.

Trace Ratio per Set: Ratio of heater length per heater set to unit length of pipe for the selected segment.

Number of Sets: The total number of heater sets required to trace the selected segment.

Heater Sheath Temp: The temperature of the heater under operating conditions.

Min. Controlled Temp.: The minimum temperature the selected segment will be maintained with the currently selected control type, sensor location, and other input data.

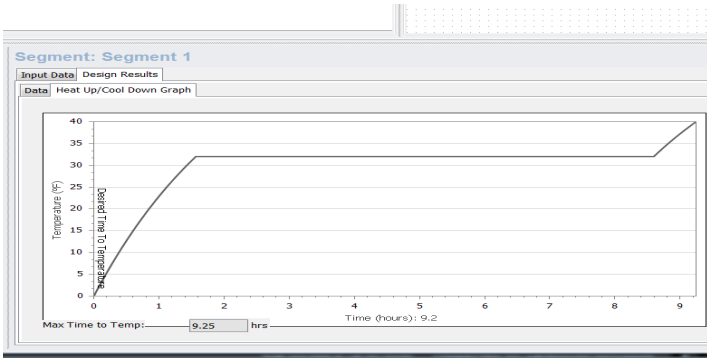
Max. Controlled Temp.: The maximum temperature the selected segment will be maintained with the currently selected control type, sensor location, and other input data.

Maximum Pipe Temp.: This is the stabilized pipe temperature assuming the heater is operating uncontrolled and at maximum ambient conditions as defined by the standards body selected under the Electrical Standards and specific area classification indicated in the Environmental group.

Max. Cable Temp.: The heater sheath temperature associated with the Maximum Pipe Temperature.

Design Results - Heat Up/Cool Down Graph

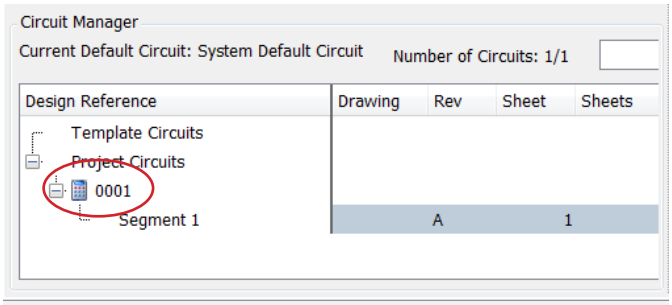
This tab will be available only if the analysis type on the Input Data tab is set to Heat Up or Cool Down.



The graph represents the heat up or cool down process in a Time vs. Temperature plot for the selected segment.

Generating Circuit Designs

In the circuit design creation process, there are several indicators to alert the user of the status of their current design. The symbol can indicate that either a design has not yet been generated or a design has already been generated, but has been altered in some manner and so needs to be redesigned.



Once a circuit has been defined and the user has attempted to design it, one of four symbols will appear:

Successful design

Design with caution

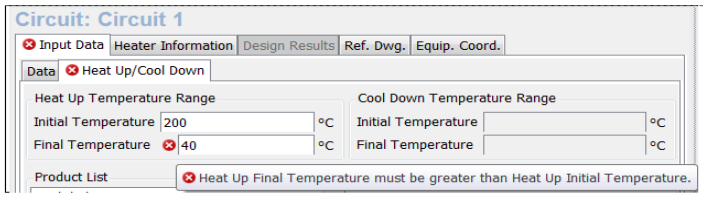
In addition to the symbol on the Circuit Manager pane, a warning message will be included in the Design Results tab, such as low temperatures on specific segments. It is the responsibility of the user to determine if the provided warning(s) affect the integrity of the overall design.

⚠ Warning

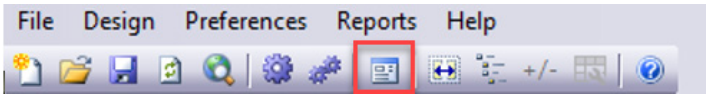
In addition to the symbol on the Circuit Manager pane, a warning message will be included in the **Design Results** tab. These warnings normally signify that although results are provided, it is up to the user to ultimately decide if the specified design is realistically feasible.

Error

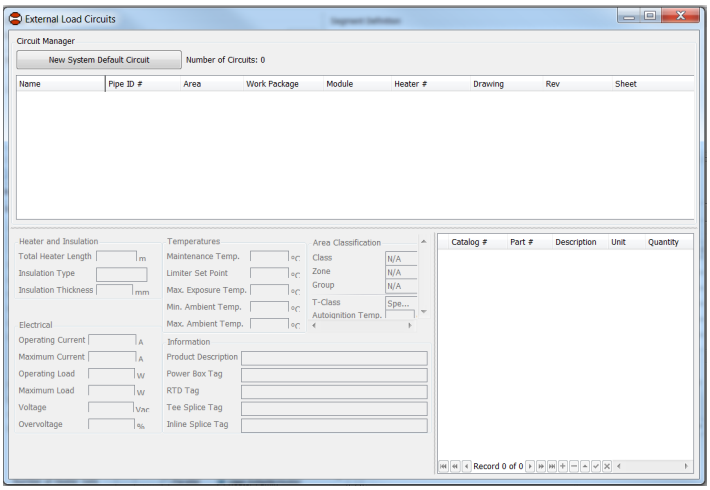
The ⊗ symbol in the Circuit Manager pane indicates an unacceptable value has been entered. This symbol will also appear on the associated Input Data tab, which helps guide the user to the field containing the error. Hover the cursor over the error symbol to see a message stating the nature of the error.



Manage External Load Circuits



This feature opens a window that enables the user to add circuit load data and BoM components to the project from circuits not designed in CompuTrace. Once added, these materials will be included in the project reports.



Clicking New System Default Circuit will add a new circuit to the system, adding the next available number sequence under "Name". The name assigned to this external load circuit can be modified by the user once added, but if not renamed, the number will be skipped when new circuits to be designed are added to the Circuit Manager.

NOTE: This is not the same as either a default Project Circuit or default Template Circuit as there is no design criteria added to the system for this external load circuit.

Once a new circuit name is added, all additional fields are enabled for data entry. This window also has a pane for adding material items associated with the external load circuit to be shown in reports.

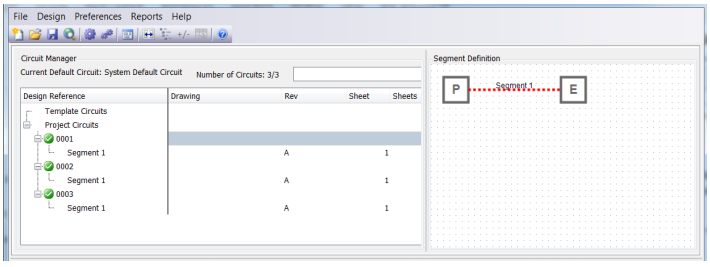
When the user is finished adding information for external load circuits, simply closing the window will save the data entered.

NOTE: There are no checks or verifications performed on the data entered, as the information is stored only for inclusion in reports. Circuits names/numbers added through this window will not be represented in any screen views other than reports.

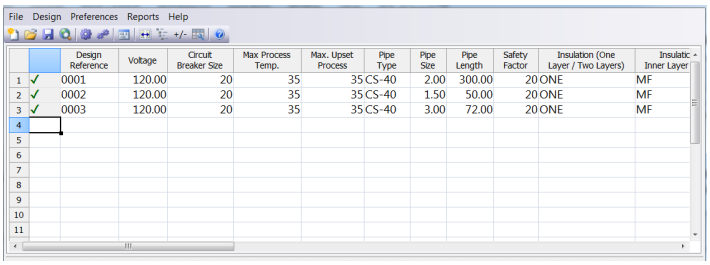
Grid View Layout

New to CompuTrace 6.1 is the addition of a Grid View that allows the user to enter, view and edit circuits in a spreadsheet environment. All data entered in the Circuit and Segment panes will be reflected in the Grid View; likewise, all data entered in the Grid View will be reflected in the Circuit and Segment panes.

 The Grid View button found on the tool bar in the Main Design Window will switch the view on the top half of the Design Window between the standard Tree View and the Grid View.



Standard Tree View

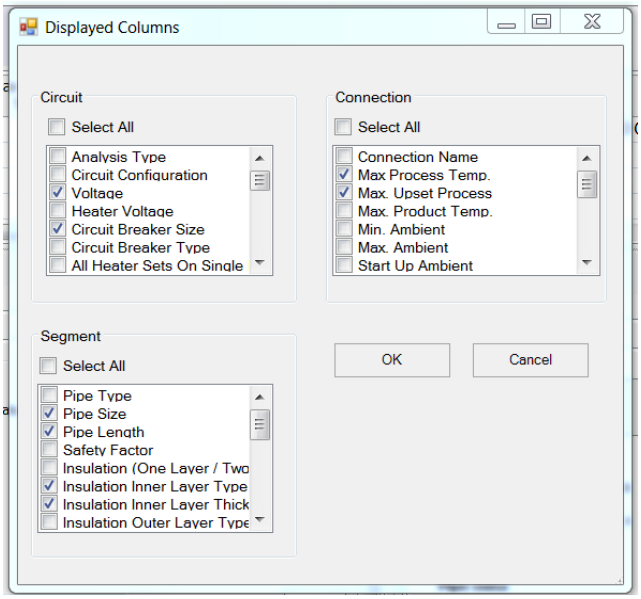


NEW Grid View

Since the functions of the Grid View are similar to Excel, the user is able to easily copy and paste multiple circuits and/or multiple design criteria entries quickly.


To add a new default Project Circuit to the design from the Grid View, the user simply clicks on the New Circuit button. To add a new circuit without starting from the default circuit, the user simply enters a new Design Reference name/number in the next blank row.


The Grid View Configuration button found on the tool bar in the Main Design Window opens a window where the user can choose which fields from the Circuit and Segment panes are displayed in the Grid View pane ("Displayed Columns"). The data for all fields in the Circuit and Segment panes is available in the Grid View pane for any specific heater.
To create a new circuit with multiple segments, the Segment Name column must be displayed. The user enters a segment name/number in this column leaving the Design Reference column blank.



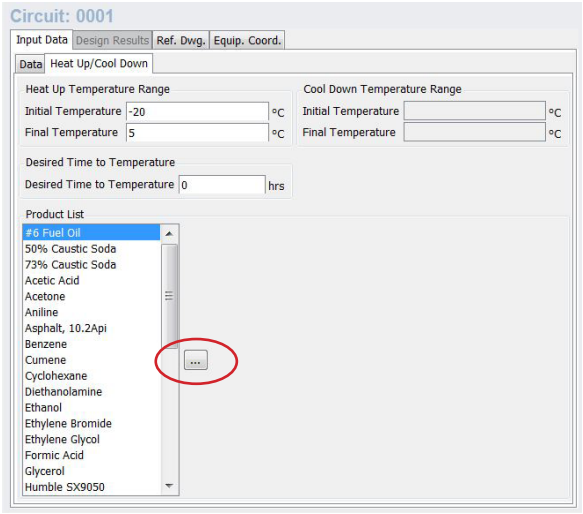
Creating/Editing Lists in CompuTrace

CompuTrace will use data stored in database tables to determine many values used in the design calculations. Although there will be many values already defined for pipe types, insulation types, various heat sinks and process fluids, the user can add custom values to the lists. Once these custom types have been defined, they become part of the database and will be available for future projects created in the database. The process for defining these custom properties is similar for each component although each one is defined using different types of information.

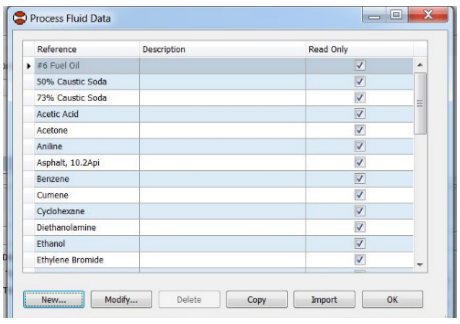
The functionality of the database record input for the following lists works in the same manner as described previously in the **Heat Sink Allowances** section. Click the  button to add a new row and the  button to save the entry in the database.

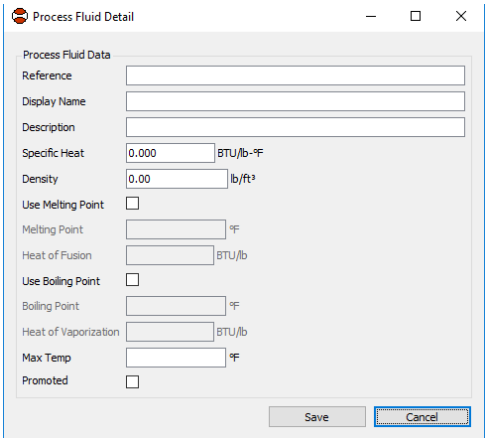
It is important to note that properties for default products are read-only and cannot be modified.

Process Fluid Data



Clicking the  button on the Circuit Input Data – Heat Up/Cool Down tab will display the screen of default products below:





Reference: Enter the text that will appear in the Process Fluid list and reports.

Display Name: This field is for reference only.

Description: Enter any additional information to describe the process fluid.

CallUse Melting Point: If the product will go through a phase change, check this box and enter the applicable melting point and heat of fusion.

Use Boiling Point: If the product will go through a phase change, check this box and enter the applicable boiling point and heat of vaporization.

Max Temp.: Enter the maximum allowable product temperature.

Promoted: Check this box if the defined process fluids and properties are to be available for use in other server projects. This will add the process fluid(s) to the server list that will be displayed when "Import" is selected (see below).

***All physical property fields must be filled in to successfully define a new process fluid.

Click on Modify... to **Edit** a user-created process fluid.

Click on Delete to **Delete** a default copy or user-created process fluid.

Click on Copy to make a **Copy** of a default or user-created process fluid.

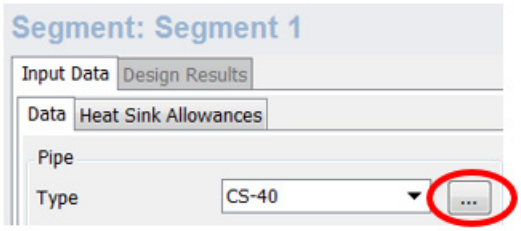
USER TIP: As a time-saving suggestion, instead of having to create a brand new Process Fluid product each time, select a product from the current list that is the most similar in properties as the new Process Fluid and use the 'Copy' button to duplicate it. Then, make the property changes as needed and change the Process Fluid name.

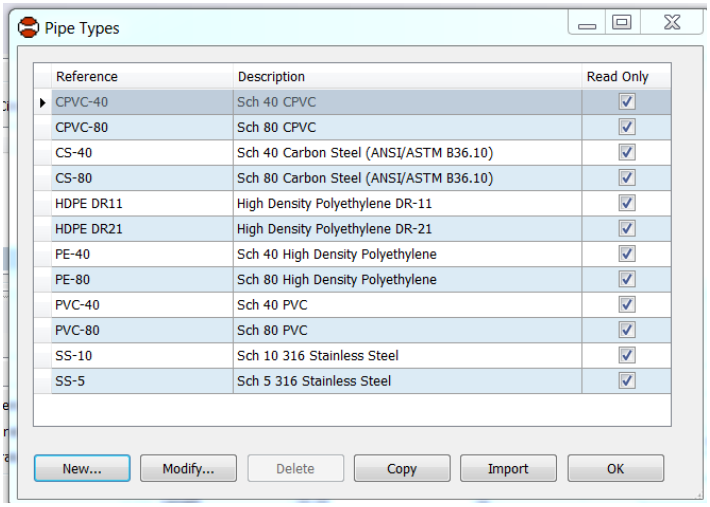
Click on Import to access a list of products and properties that are available on the server for use in other projects.

Click on OK to close this window and return to the Product List.

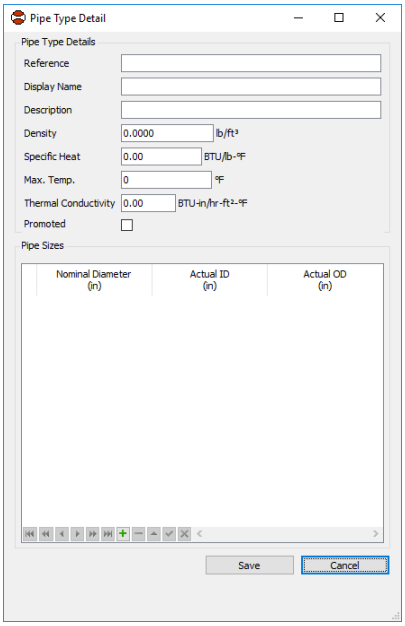
Pipe Data

Clicking the ... button on the Segment Input Data - Pipe Group will display the screen of default pipe types below:





Click on New... to **Create** a new pipe type and a window opens for the user to enter details.



Reference: Enter the text that will appear in the Pipe Type drop down list and on reports.

Display Name: This field is for reference only.

Description: Enter a longer description that will appear in the Pipe Type drop down list.

Density: Enter the mass density of the pipe material.

Specific Heat: Enter the specific heat of the pipe material.

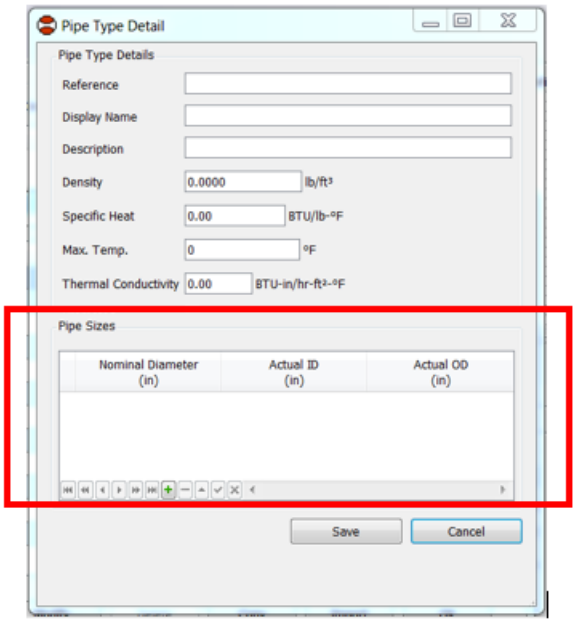
Max. Temp: Enter the maximum temperature of the pipe can withstand.

Thermal Conductivity: Enter the thermal conductivity of the pipe material.

Promoted: Check this box if the defined pipe types and properties are to be available for use in other server projects. This will add the pipe type(s) to the server list that will be displayed when "Import" is selected (see below).

Pipe Sizes Table

Enter the dimensional data for each pipe size represented by this pipe type.



Nominal Diameter: Enter the value that will be displayed in the Pipe Size drop down list and on reports.

Actual ID: Enter the actual inner diameter associated with the nominal pipe size

Actual OD: Enter the actual outer diameter associated with the nominal pipe size.

Click on Modify... to **Edit** a user-created pipe type.

Click on Delete to **Delete** a default copy or user-created pipe type.

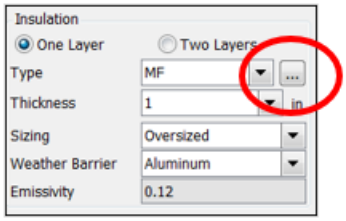
Click on Copy to make a **Copy** of a default or user-created pipe type:

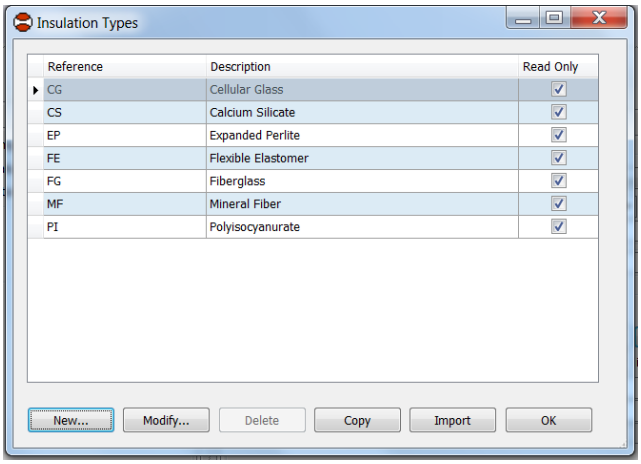
USER TIP: As a time-saving suggestion, instead of having to create a brand new Pipe Type item each time, select an item from the current list that is the most similar in properties as the new Pipe Type and use the 'Copy' button to duplicate it. Then, make the property changes as needed and change the Pipe Type name.

Click on Import to access a list of pipe types and properties that are available on the server for use in other projects.

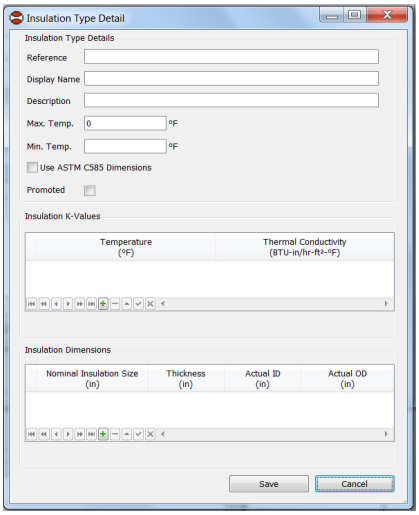
Insulation Data

Clicking the ... button under the Insulation Group will display the following screen listing the default insulation types :





Click on [New...] to **Create** a new insulation type:



Reference: Enter the text that will appear in the Insulation Type drop down list and on reports.

Display Name: This field is for reference only.

Description: Enter a description that will appear in the Insulation Type drop down list.

Max. Temp.: Enter the maximum temperature the insulation can be exposed to.

Min. Temp.: Enter the minimum temperature the insulation can be exposed to.

Use ASTM C585 Dimensions: Check this box if the insulation being added conforms to the dimensions defined in the ASTM C585. The program will automatically populate the Insulation Dimensions table with the appropriate values. Otherwise, this table must be populated by hand with all insulation dimensions for each thickness and pipe size.

Promoted: Check this box if the defined insulation type and properties are to be available for use in other server projects. This will add the insulation(s) to the server list that will be displayed when "Import" is selected (see below).

Insulation K-Values

Use this table to define the thermal conductivity of the insulation as a function of temperature.

Insulation Dimensions

Use this table to define the insulation dimensions for each pipe size and thickness. If the Use ASTM C585 Dimensions checkbox is checked then this table will be filled in automatically.

Nominal Insulation Size: The pipe size this entry would fit.

Thickness: The value displayed in the insulation thickness drop down list and on reports.

Actual ID: Enter the actual inner diameter for this entry.

Actual OD: Enter the actual outer diameter for this entry.

Click on [Modify...] to **Edit** a user-created insulation type.

Click on [Delete] to **Delete** a default copy or user-created insulation type:

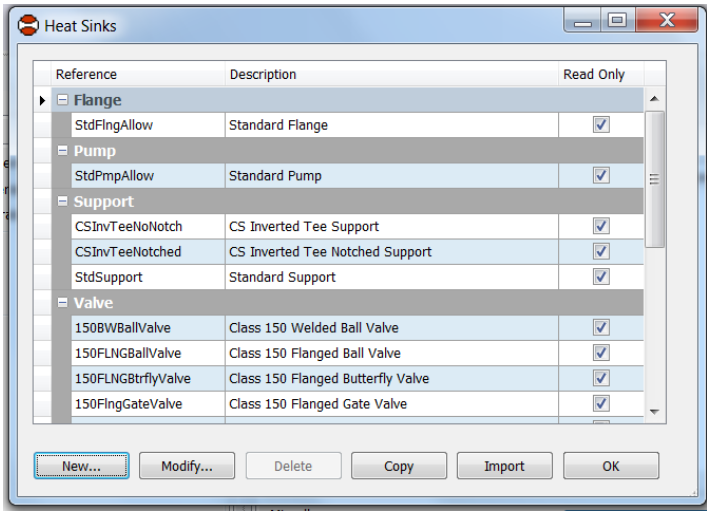
Click on [Copy] to make a **Copy** of a default or user-created insulation type.

USER TIP: As a time-saving suggestion, instead of having to create a brand new Insulation material each time, select a material from the current list that is the most similar in properties as the new Insulation and use the 'Copy' button to duplicate it. Then, make the property changes as needed and change the Insulation name.

Click on [Import] to access a list of insulations and properties that are available on the server for use in other projects.

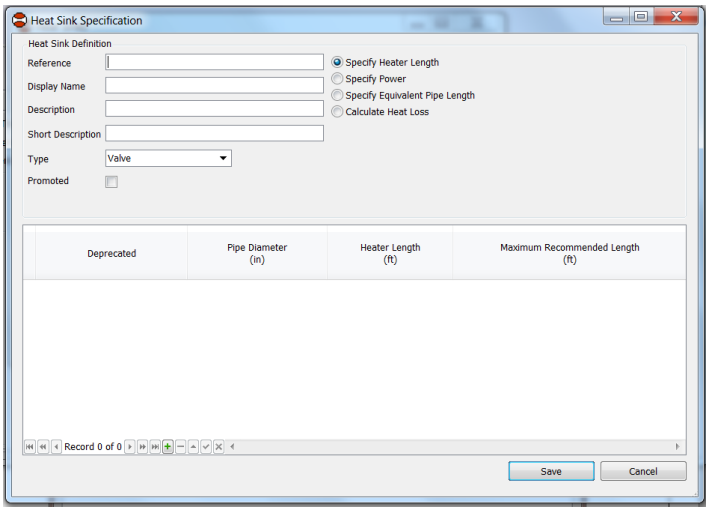
Heat Sink Data

When [Define Heat Sinks...] is clicked for any one of the heat sink allowance types, the following window will appear listing the default heat sinks:



This list contains all of the heat sink entries in the current project. The list is sorted by type (e.g. Flange, Pump, Support, etc.).

Click on New... to **Create** a new heat sink:



Reference: A Text Box field which contains a short alphanumeric string used to identify this component in the tables.

Display Name: This field is for reference only.

Description: A Text Box field which contains a description of the component to be displayed in drop down lists and reports.

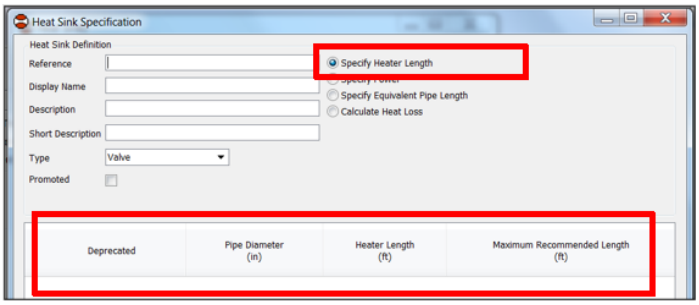
Short Description: A Text Box field which contains a short description of the component to be displayed in drop down lists and reports.

Type: Drop Down List which displays the options the user can select from to define the type of heat sink. Options will be Flange, Valve, Pump, or Support.

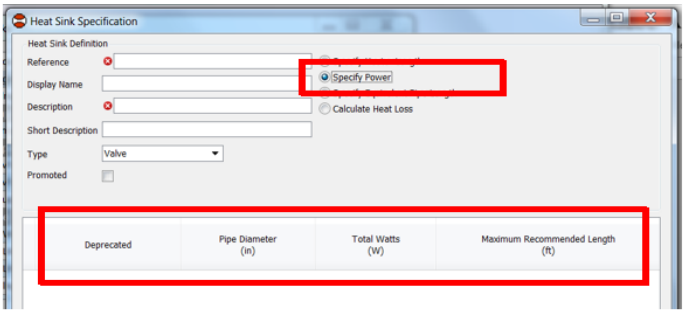
Promoted: Check this box if the defined heat sinks and properties are to be available for use in other server projects. This will add the heat sink(s) to the server list that will be displayed when "Import" is selected (see below).

Deprecated: In previous versions of CompuTrace this feature indicated a value included for historic purposes only, which could not be added to new projects. This was used to avoid compatibility issues with imports from older versions.

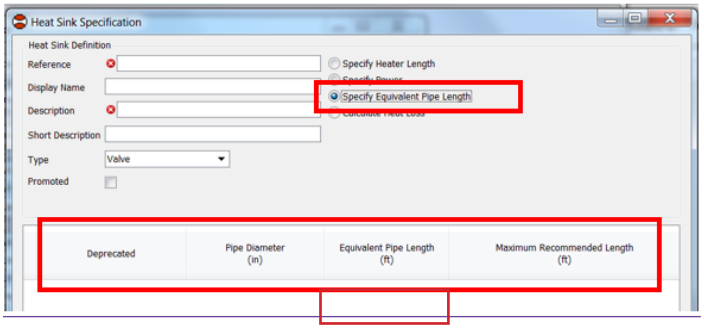
Specify Heater Length: When selected, CompuTrace will include the Heater Length column for the user to populate for each pipe size.



Specify Power: When selected, CompuTrace will include the Total Watts column for the user to indicate the required power to be applied for each pipe size.

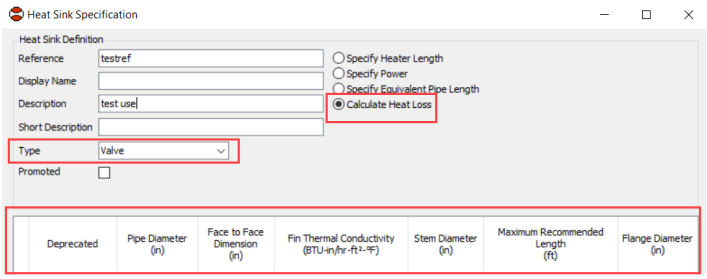


Specify Equivalent Pipe Length: When selected, CompuTrace will include the Equivalent Pipe Length column for the user to enter the equivalent pipe length for each pipe size.

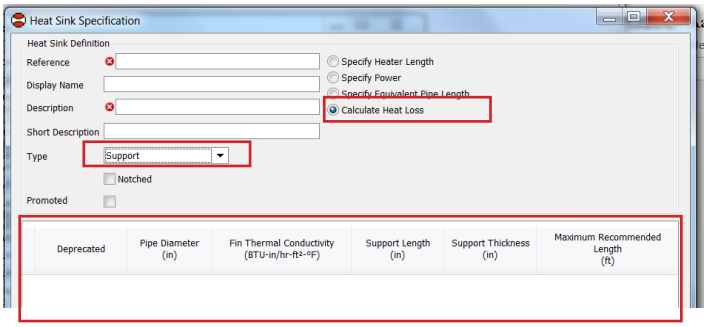


Calculate Heat Loss: This option will only be available when the user specifies either Valve or Support in the Type drop down list. When selected, CompuTrace will provide the following columns for the user to complete for each pipe size:

Valve: Face to Face Dimension, Fin Thermal Conductivity, Stem Diameter and Flange Diameter

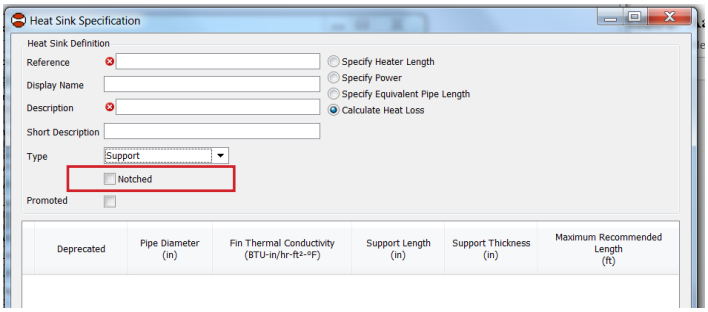


Support: Fin Thermal Conductivity, Support Length, and Support Thickness



These values will be used to determine the heat loss from the component.

NOTE: That all tables contain the **Maximum Recommended Length** column. This value is used to determine if the user has specified a condition that requires more heater than can actually be installed on the component. If this occurs, then the program will issue a warning on the design.



Notched Pipe Supports : The calculation of the heat loss from an inverted T pipe support is performed assuming an infinite fin model. The resistance to heat flow through the insulated portion of the support is considered as pure conduction. The temperature of the support as it emerges from the insulation is treated as the base temperature of an infinite fin.

Either of two cases can be considered, one without a notch and one with. The case without a notch is calculated as described above. The case with a notch is modified to consider the reduced contact area with the pipe and models the heat transfer across the notch as convective heat transfer. This modification impacts the base temperature for the infinite fin.

The functionality of the database record input works in the same manner as described in the **Heat Sink Allowances** section.

Click on Modify... to **Edit** a user-created heat sink.

Click on Delete to **Delete** a default copy or user-created insulation type.

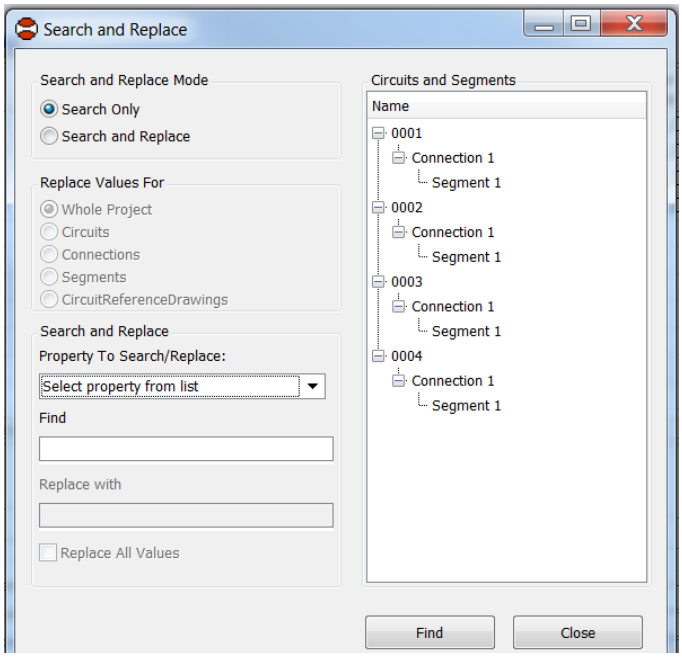
Click on Copy to make a **Copy** of a default or user-created heat sink.

USER TIP: As a time-saving suggestion, instead of having to create a brand new Heat Sink item each time, select an item from the current list that is the most similar in properties as the new Heat Sink and use the 'Copy' button to duplicate it. Then, make the property changes as needed and change the Heat Sink name.

Click on Import to access a list of heat sinks that are available on the server for use in other projects.

Using the Search and Replace Feature

CompuTrace Design Suite provides the ability to search and replace a variety of properties at the project, circuit and segment levels:



Search and Replace Mode: The user will be able to select from searching for a specific value only or searching as well as replacing the value with a specified input value.

Replace Values For: This collection will contain a radio button grouping in which the user must select one of five options: Whole Project, Circuits, Connections, Segments or Circuit Reference Drawings. Note that this grouping is only activated if the Search and Replace radio button is selected.

The tree diagram on the right will be empty or disabled if a project-level replacement is selected.

Property to Search/Replace: A drop down box which specifies the category to be searched and/or replaced by both its Entity Type and Property. The user can only replace one type of property at a time.

Find: The user specifies the value to be searched for in the category chosen in the Property to Search/ Replace drop down box.

Replace with: The user specifies the replacement value to be used based on input from the Find textbox for the category chosen in the Property to Search/Replace drop down box. Note that this grouping is only activated if the Search and Replace radio button is selected.

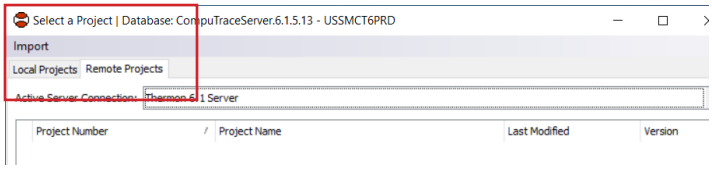
Replace All Values: By checking this checkbox, all values will be replaced with the specified value in the chosen circuits or segments, and the Find textbox will be disabled. Note that this grouping is only activated if the Search and Replace radio button is selected.

Circuits and Segments: This panel is a tree control interface with multi-select functionality. The user will select the circuits or segments they wish to replace values for. For example, selecting a circuit while doing a segment-level replace should apply to all segments in that circuit.

Section III. CompuTrace for Server Projects

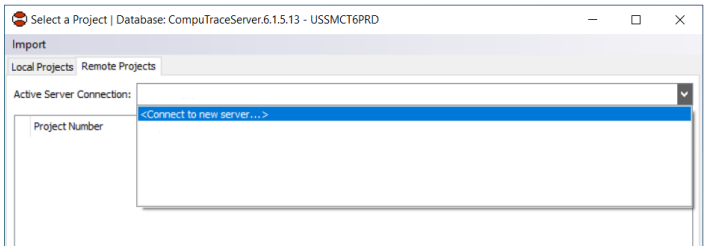
Using Server Projects

When working on a project to be saved on a server, the user will select **Remote Projects** at the **Select Project** screen after opening CompuTrace.

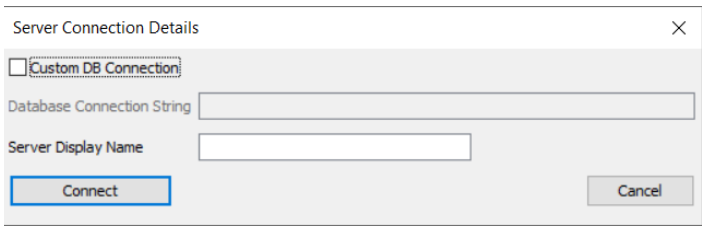


Connecting to a New Server

To connect to a new server select the **Remote Projects** tab on the Select Project screen, open the Active Server Connection drop down list and select **Connect to new server...**

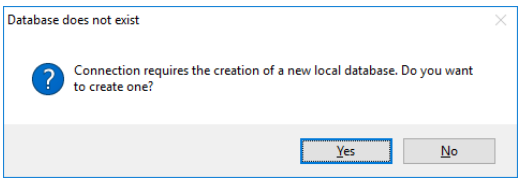


The user will need to enter the required information to connect to a new server.

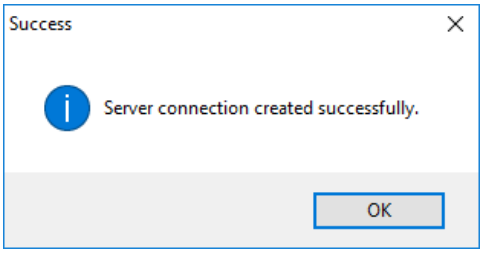


Check the Custom DB Connection box and enter the connection string in the Database Connection String field.

The Server Display Name is the name selected for this server connection to be displayed in the Active Server Connection drop down list on the Select Project Server Projects tab. When all information is entered, click Connect.



When prompted to create a new local **database**, click Yes.

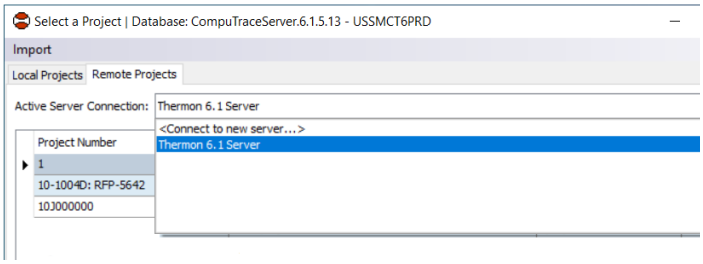


Once the server connection is complete, the Server Projects screen will return.

CompuTrace can store connections to multiple servers. To do this, repeat the above process for each server to be added. To switch between servers use the **Active Server Connection** drop down list to select the desired server.

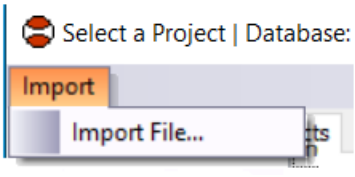
Connecting to an Existing Server

This type of project mode is for users working in collaboration with other users on a project via a shared server. On the Server Projects tab, open the Active Server Connection drop down list to select the server to be used. This screen shows the current projects available in this server. In the example shown below there are three projects that have already been created by other users.



Import Options for Remote Projects

As noted previously in this manual, the **Select Project** screen includes an **Import** feature. When the **Remote Projects** tab is selected, this feature offers the same option of importing from an xml file as when the **Local Projects** tab is selected.

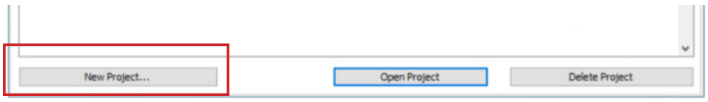


Server Projects Tab

The following is a list of actions possible from the Server Projects Tab.

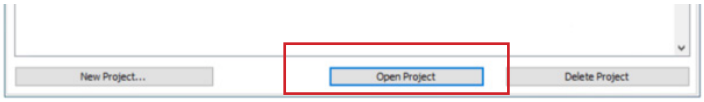
Create a New Remote Project

To create a new Remote project, click **New Project** to open the **Create New Project Wizard**.



Connect to an Existing Remote Project

To connect to an existing remote project that another user has created and saved, click on the Project Name to be opened and then click **Open Project** at the bottom of the screen.



THERMON

7171 Southwest Pkwy | building 300 suite 200 | Austin, TX | 78735 | United States

TMP0050-0223

www.Thermon.com