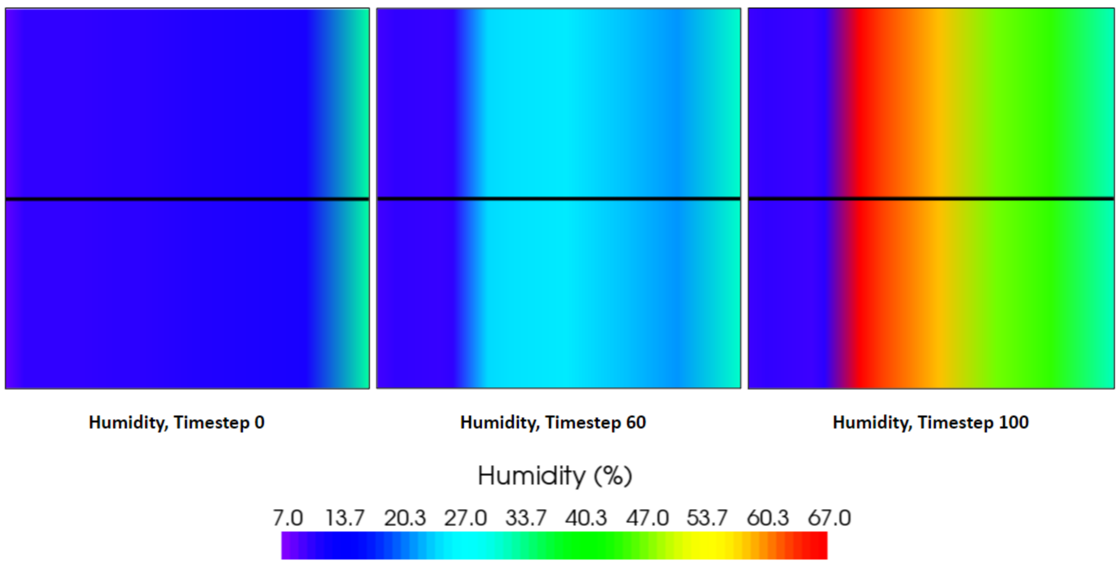
LBL 37371 Rev. 8
November 2020

Program Description

A PC Program

THERM 8



for Analyzing Two-Dimensional Heat and Moisture Transfer Through Building Products

Windows and Daylighting Group
Building Technologies Department
Environmental Energy Technologies Division
Ernest Orlando Lawrence Berkeley National Laboratory
Berkeley CA 94720 USA

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Building Technologies Program, of the U.S. Department of Energy under Contract no. DE-AC02-05CH11231.

LBL-37371 Rev. 8

THERM 8: Program Description

A PC Program for Analyzing the Two-Dimensional Heat Transfer Through Building Products

Christian Kohler, Charlie Curcija, Robin Mitchell, Dariush Arasteh
Stephen Czarnecki, Simon Vidanovic, Ling Zhu
Windows and Daylighting Group
Building Technologies Department
Environmental Energy Technologies Division
Lawrence Berkeley National Laboratory
Berkeley, California 94720
http://windows.lbl.gov/software/software.html

Charlie Huizenga
Center for Environmental Design Research
University of California
Berkeley, California

November 7, 2020

This work was supported by the Assistant Secretary for Energy Efficiency and Renewable Energy, Building Technologies Program, of the U.S. Department of Energy under Contract no. DE-AC02-05CH11231.

1. INTRODUCTION

1.1. Overview

THERM is a Microsoft Windows™-based computer program developed at Lawrence Berkeley National Laboratory (LBNL) for use by building component manufacturers, engineers, educators, students, architects, and others interested in two-dimensional heat transfer. Using THERM, you can model two-dimensional heat-transfer effects in building components such as windows, walls, foundations, roofs, and doors, appliances, and other products where thermal bridges are of concern. THERM's heat-transfer analysis allows you to evaluate a product's energy efficiency and local temperature patterns, which may relate directly to problems with condensation, moisture damage, and structural integrity.

THERM's two-dimensional conduction heat-transfer analysis is based on the finite-element method, which can model the complicated geometries of building products. The program's graphic interface allows you to draw cross sections of products or components to be analyzed. To create the cross sections, you can trace imported files in DXF or bitmap format, or input the geometry from known dimensions. Each cross section is represented by a combination of polygons. The material properties are defined for each polygon and the environmental conditions to which the component is exposed are defined by the boundary conditions surrounding the cross section. Once the model is created, the remaining analysis (mesher and heat transfer) is automatic. You can view results from THERM in several forms, including U-factors, isotherms, heat-flux vectors, and local temperatures.

THERM's results can be used to define the frame elements in the Berkeley Lab WINDOW program's center-of-glass optical and thermal models to determine total window product U-factors and Solar Heat Gain Coefficients.

Program downloads, update information, and Knowledge Base articles about THERM and the other software tools from the Windows and Daylighting Group at LBNL can be found at: windows.lbl.gov. If you have questions or problems about using the program, email **ThermHelp@lbl.gov** or go to the THERM Forum (Google Group) at https://groups.google.com/forum/#!forum/lbnl-therm where you can post and reply to questions about the program.

1.2. THERM 8 Transient Moisture Model

The THERM simulation engine been extended to model time dependent (transient) simulations (a "time domain" has been added to THERM's original 2-D Finite Element Numerical Model (FEM)), and a moisture transfer model has been added.

The transient thermal and moisture simulation engine in THERM (called HygroThermFEM) allows modeling of thermal bridges and non-homogeneities in building construction without approximation. Such elements are primary pathways and causes for condensation and moisture in building and it is important to model them correctly in a thermal/moisture analysis. By failing to account for the moisture characteristics in the thermal envelope, designers and building can introduce problems that endanger the health and safety of building occupants as well as the durability of the building itself.

The HygroTHERMFEM model in THERM8 will allow building simulation practitioners to accurately model wall, roof, foundation constructions taking into account both the thermal and moisture characteristics of those constructions.

1.3. Changes from THERM 7 to THERM 8

The following are the changes made to THERM 8 to accommodate the new moisture model:

- Simulation engine options (File / Properties)
  - Steady-state without moisture (THERM 7)
  - Transient/moisture
- Material and Boundary Condition Libraries
  - The Material and Boundary Condition Libraries for the transient / moisture model are stored in the XML file format and have a new grid view
- Visualization "window" for viewing the moisture results

This document is not a complete THERM user manual, but instead contains the changes made to THERM 8 for the Transient Moisture Model.

2. Transient Moisture and Thermal Simulations

2.1. Overview

THERM allows modeling both transient moisture and thermal simualtions, as well as stead-state thermal simulations.

The sections below describe the settings for transient moisture and thermal simulations.

2.2. File / Properties

The File / Properties dialog box controls what model is used for a given file. All the settings in the three tabs of the File / Properties dialog box are saved with individual files, not as program-level settings.

Calculation Options tab

Below is a description of the fields in the File / Properties / Calculation Options tab.

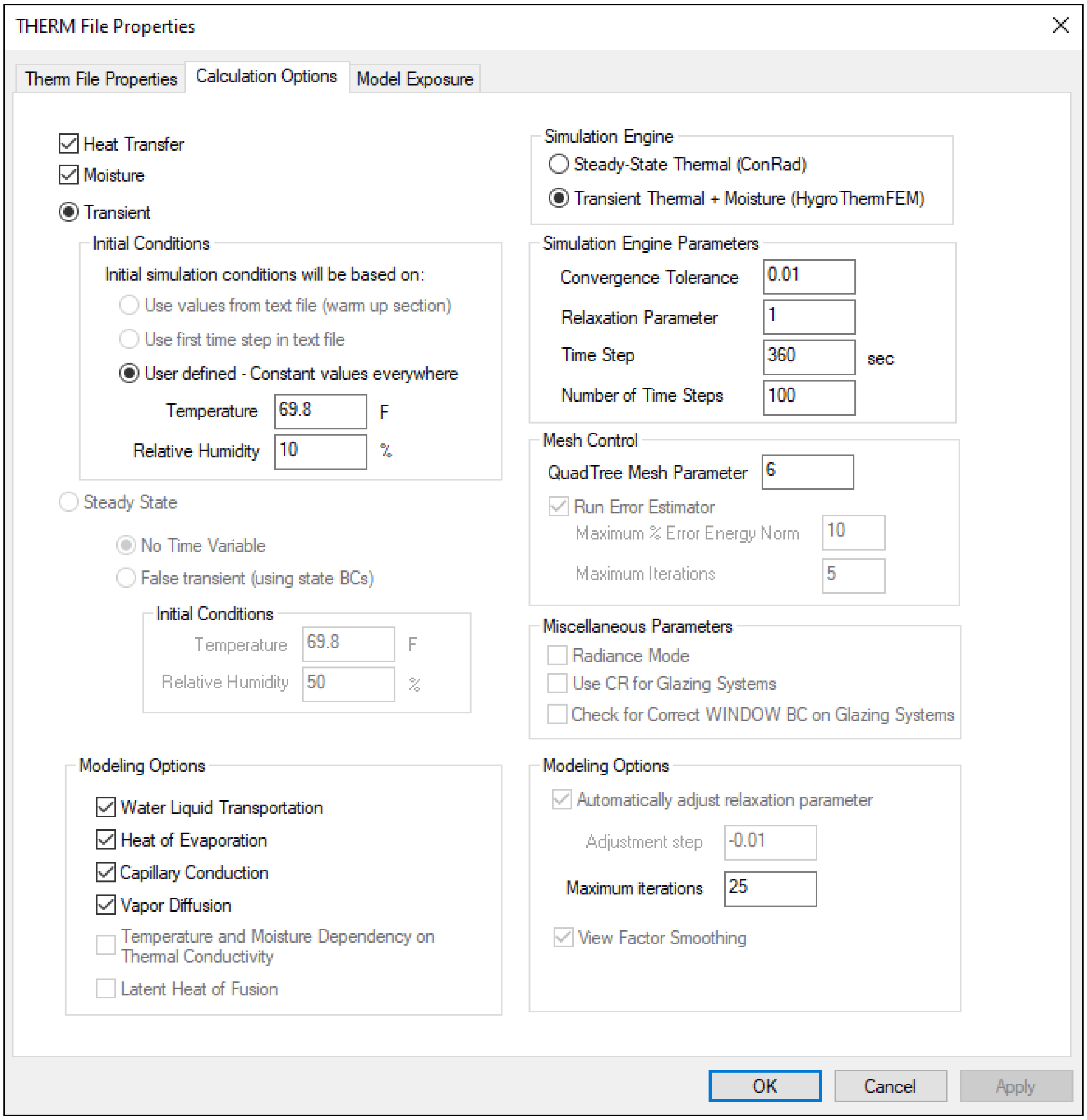


Figure 2-1. File/Properties contains settings for both the steady-state and transient models.

Heat Transfer	Check this box to model heat transfer. Checking this box is appropriate for ▪ Simulation engine = Steady-State Thermal (ConRad) ▪ Simulation engine = Transient Therm + Moisture (HygroThermFEM)
Moisture	Check this box to model moisture.

	Checking this box is appropriate for ▪ Simulation engine = Transient Therm + Moisture (HygroThermFEM)
Transient	

Initial Conditions	Initial simulation conditions will be based on either:
	▪ User values from text file (warm up section): *currently not implemented*
	▪ Use first time step in text file: *currently not implemented*
	▪ User defined – Constant values everywhere
	    ▪ Temperature: Default: 21 C.
	    ▪ Relative Humidity: Default: 50. Units: percentage (%)

Steady State *(for HygroThermFEM engine)* *Currently not implemented in THERM 8*	
No Time Variable	This uses the same steady state equation to calculation heat transfer as ConRad, and there is no time variable. See technical documentation
False transient (using state BCs)	This runs a transient simulation until it reaches a steady state solution. Initial Conditions: this sets the temperature and relative humidity of the entire model to start the simulation. It is also necessary to have transient-style Boundary Conditions (ie, XML file) that have contant temperature and relative humidity values over the timesteps.
Modeling Options *For the HygorThermFEM simulation engine*	
	These are modeling options to set when modeling moisture using the HygroThermFEM engine: ▪ Water Liquid Transportation: Models transport of water in a liquid state. If this is not checked, then only water vapor is modeled. ▪ Heat of Evaporation: Models phase change of water. ▪ Capilary Conduction: Models capillary transport through ▪ Vapor Diffusion Conduction: Models diffusion of vapor thorugh material. ▪ Temperature and Moisture Dependency on Thermal Conductivity: *Currently not available* ▪ Latent Heat of Fusion: *Currently not available* Default: All options are selected by default.
Simulation Engine	
Steady-State Thermal (ConRad)	Steady-state thermal simulation

Transient Therm + Moisture (HygroTherm FEM)	Transient thermal and moisture simulation engine
Simulation Engine Parameters	
Convergence Tolerance	This controls the simulation error norm tolerance. If the error norm calculation is greater than this value, the program will display an "non-convergence" error message. If using the transient moisture model and it doesn't converge, this value could be reduced to 1e-02. Used in both the ConRad and HygroThermFEM simulation engines. **Default:** 1e-05.
Relaxation Parameter	This is the initial value for relaxation parameter used in iterations. Used in both the ConRad and HygroThermFEM simulation engines. **Default:** 1.
Time Step	This parameter is used by the HygroThermFEM transient model to set the length of the time step. The default value of 3600 seconds equals 1 hour. **Default:** 3600. **Units:** seconds.
Number of Time Steps	This parameter determines how many times steps are calculated when the HygroThermFEM engine is selected. It works in conjunction with the Time Step value. The default Time Step value of 3600 seconds (1 hour) used with the default value of the Number of Time Steps (8670) would result in a simulation at 1 hour intervals for a year. **Default:** 8670. **Units:** unitless
Mesh Control *For both the HygorThermFEM and ConRad simulation engines*	
Quad Tree Mesh Parameter	The relative size of the finite element mesh created for the model. The larger the number the finer the mesh. The upper limit of the mesh parameter is usually 9. Increasing this value may help mesh files that return "mesh errors". Default: 6.
Run Error Estimator	If this option is checked, the program will perform the calculation of the error energy norm. If the simulation does not meet the value in Maximum % Error Energy Norm, the program will increment the Quadtree Mesh Parameter (if Automatically increment mesh parameter is checked on the Simulation tab), until the Maximum Iteration value is reached. **Default:** checked.
Maximum % Error Energy Norm	THERM incorporates a local error estimator and subsequent local mesh refinement. The error estimator is based on a published and well regarded methodology based on the estimate of Error Energy Norm (EEN). In this technique,

	only regions with an EEN larger than the threshold value are refined, resulting in optimum mesh size. An EEN value of 10% is chosen for the default because it results in overall computational accuracy of well below 1%, which is the ISO 10211 requirement. **Default:** 10%
Maximum Iterations	The number of iterations for the program to run in an attempt to meet the Maximum % Error Energy Norm specified. **Default:** 5
Miscellaneous Parameters *For the ConRad simulation engine*	
Radiance Mode	If this option is checked, THERM will use Radiance to model the optical properties of a THERM model. This is used for modeling shading systems whose geometry is modeled in THERM. The THERM background will turn gray when this option is checked, to indicate that this option is selected, because this is not an option that would normally be used. Default: unchecked.
Use CR for Glazing Systems	If this option is checked, THERM will perform a Condensation Resistance simulation as well as the standard U-factor calculation. This option only applies to the ConRad Simulation Engine. Default: unchecked.
Check for Correct WINDOW BC on Glazing Systems	If this option is checked, THERM will check to see that the Boundary Conditions applied to an imported WINDOW glazing system match those that were imported with that glazing system. Default: unchecked.
Modeling Options	
Automatically adjust relaxation parameter	When equations are not converging, the relaxation parameter modifies previous iteration result in order to better converge. The values are from 0 to 1. The solver starts from a value of 1 and reduces it from there. *For ConRad simulation engine.* Default: checked
Adjustment step	Incremental reduction of the relaxation parameter from 1 to 0. *For ConRad simulation engine.* Default: -0.01

Maximum iterations	Number of iterations to apply the automatic relaxation parameter adjustment to. *For both the HygorThermFEM and ConRad simulation engines* Default: 25
View Factor Smoothing	This is used for gray body radiation calculation. If the view factors (surfaces being viewed) are very different, such as blocking surfaces, the view factor smoothing helps accuracy of results. See TARCOG technical documentation on the THERM documentation webpage. *For ConRad simulation engine.*

Typical settings for a transient thermal and moisture simulation (HygroThermFEM)

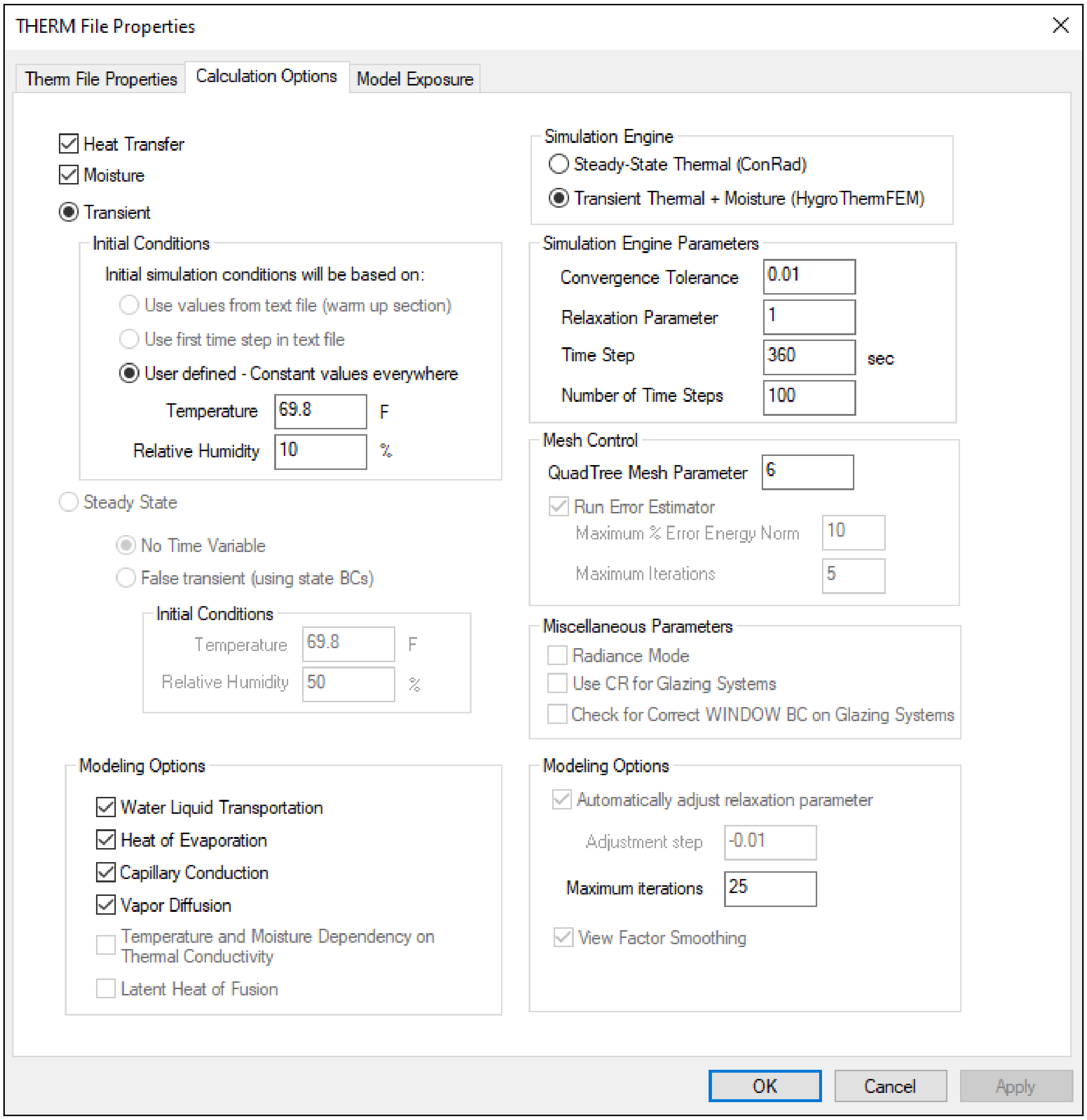


Figure 2-2. Typical File/Properties settings for a transient thermal and moisture model.

Typical settings for a steady-state thermal calculation (ConRad)

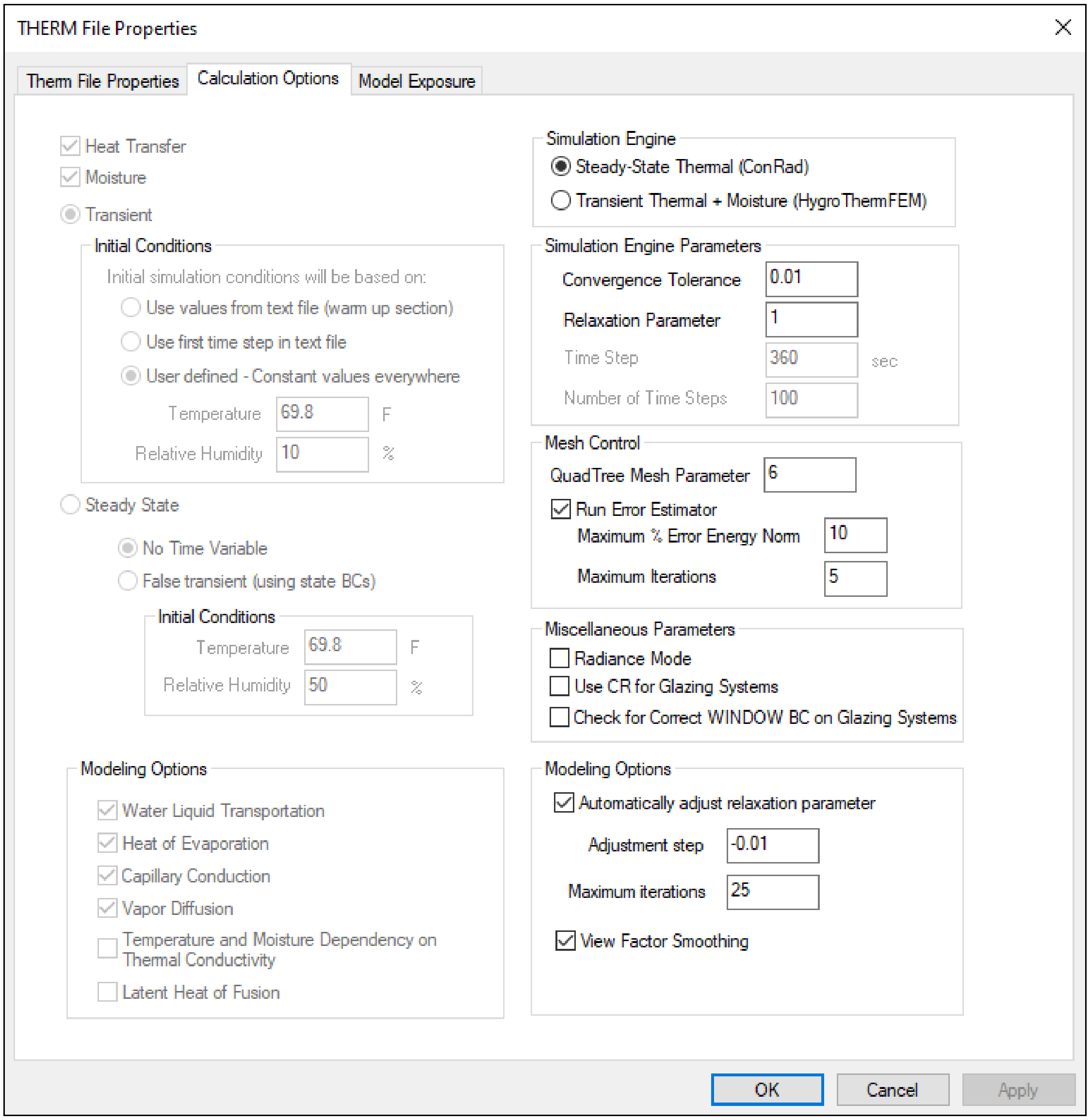


Figure 2-3. Typical File/Properties settings for a steady-state thermal model.

Model Exposure tab

Below is a description of the fields in the File / Properties / Model Exposure tab.

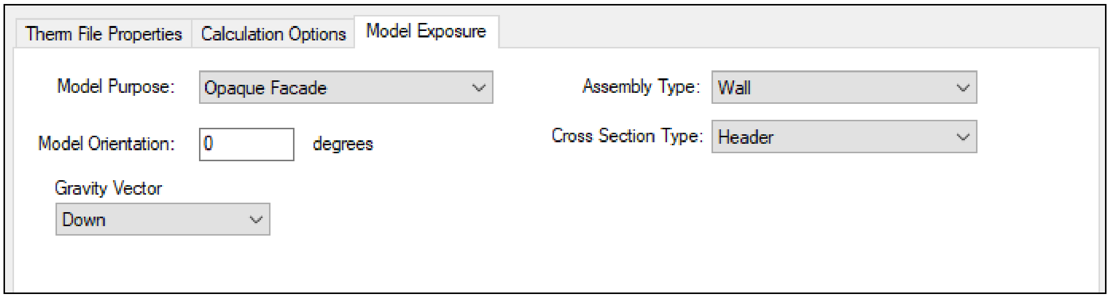
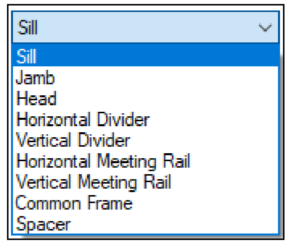
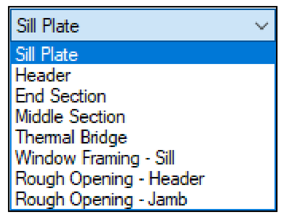
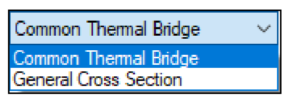
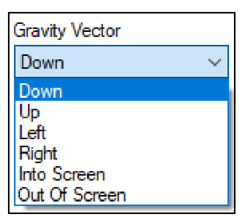


Figure 2-4. File Properties / Model Exposure tab.

Model Purpose	This pulldown list consists of the following options ▪ Window / Transparent Façade ▪ Opaque Façade ▪ Other
Assembly Type	If **Model Purpose** is set to **Opaque Façade**, this pulldown list becomes active with the following choices: ▪ Wall ▪ Roof If **Model Purpose** is set to either **Window/Transparent Façade** or **Other**, this value is set to "N/A".
Model Orientation	The direction the exterior surface of the model faces. This orientation of the exterior surface is used to determine the wind direction for the simulation. Below are the standard Cardinal orientations, but any value from 0 – 360 may be entered. ▪ North = 0 ▪ East = 90 ▪ South = 180 ▪ West = 270 **Legal values:** 0 – 360. **Units:** degrees
Cross Section Type	The values in this pulldown depend on the Model Purpose selection. ▪ If Model Purpose = Window / Transparent Façade, the choices are

	 *Figure 2-5. Cross-Section Type if Model Purpose = Window/Transparent Facade.* *Note: this setting was located under the Therm File Properties tab in previous versions of the program.* ▪ If Model Purpose = Opaque Façade, the choices are  *Figure 2-6. Cross-Section Type if Model Purpose = Opaque Facade.* ▪ If Model Purpose = Other, the choices are  *Figure 2-7. Cross-Section Type if Model Purpose = Other.*
Gravity Vector	The choices for the gravity vector are  *Figure 2-8. Gravity Vector options.* ***Note:*** *Setting the gravity vector was done from the View menu in previous versions of THERM* ***Note:*** *This setting will change the View/Gravity Arrow to match. Don't change gravity vector from the View menu in this version of THERM.*

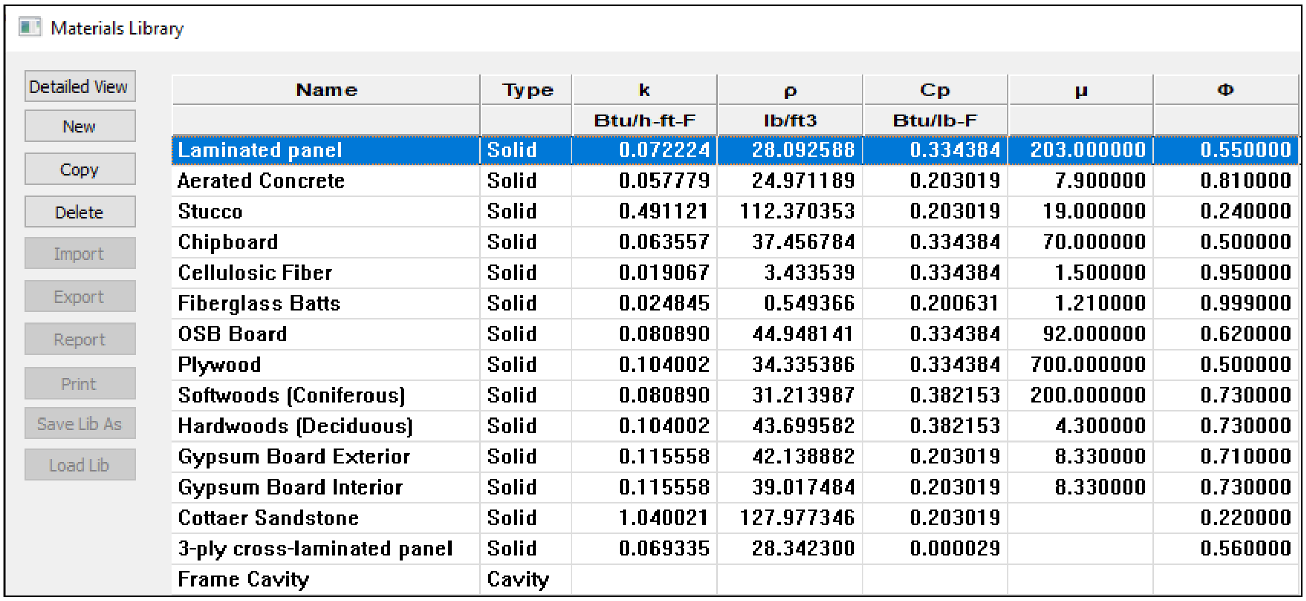
2.3. Material Library

The Material Library is different depending on the Simulation Engine selected.

- ConRad: If the ConRad simulation engine is selected, the Material library is the same as in previous versions of THERM.
- HygroThermFEM: If the HygroThermFEM simulation engine is selected, the Material library is much different.

2.3.1. Material Library List View

Selecting the Material choice from the Libraries menu when the HygroThermFEM simulation model is selected displays the Materials in “List View”.



Name	Type	k	ρ	Cp	μ	Φ
		Btu/h-ft-F	lb/ft3	Btu/lb-F		
Laminated panel	Solid	0.072224	28.092588	0.334384	203.000000	0.550000
Aerated Concrete	Solid	0.057779	24.971189	0.203019	7.900000	0.810000
Stucco	Solid	0.491121	112.370353	0.203019	19.000000	0.240000
Chipboard	Solid	0.063557	37.456784	0.334384	70.000000	0.500000
Cellulosic Fiber	Solid	0.019067	3.433539	0.334384	1.500000	0.950000
Fiberglass Batts	Solid	0.024845	0.549366	0.200631	1.210000	0.999000
OSB Board	Solid	0.080890	44.948141	0.334384	92.000000	0.620000
Plywood	Solid	0.104002	34.335386	0.334384	700.000000	0.500000
Softwoods (Coniferous)	Solid	0.080890	31.213987	0.382153	200.000000	0.730000
Hardwoods (Deciduous)	Solid	0.104002	43.699582	0.382153	4.300000	0.730000
Gypsum Board Exterior	Solid	0.115558	42.138882	0.203019	8.330000	0.710000
Gypsum Board Interior	Solid	0.115558	39.017484	0.203019	8.330000	0.730000
Cottaer Sandstone	Solid	1.040021	127.977346	0.203019		0.220000
3-ply cross-laminated panel	Solid	0.069335	28.342300	0.000029		0.560000
Frame Cavity	Cavity					

Figure 2-9. Material List View.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called “Default Name”. Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.

Import	*Currently disabled.*
Export	*Currently disabled.*
Report	*Currently disabled.*
Print	*Currently disabled.*
Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the material.
Type	The material type, from the options of ▪ Solid ▪ Frame Cavity ▪ Glazing Cavity ▪ External Radiation Enclosure ▪ Shading Material
k	Conductivity of the material. **Units:** Btu/h-ft-F (IP); W/m-K (SI)
ρ	Density of the material. **Units:** lb/ft3 (IP); kg/m3 (SI)
Cp	Specific heat of the material. **Units:** Btu/lb-F (IP);J/kg-K (SI)
μ	Water Vapor Diffusion Resistance Factor. **Units**: unitless
Φ	Porosity of the material. **Units:** unitless

2.3.1. Material Library Detail View

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each material.

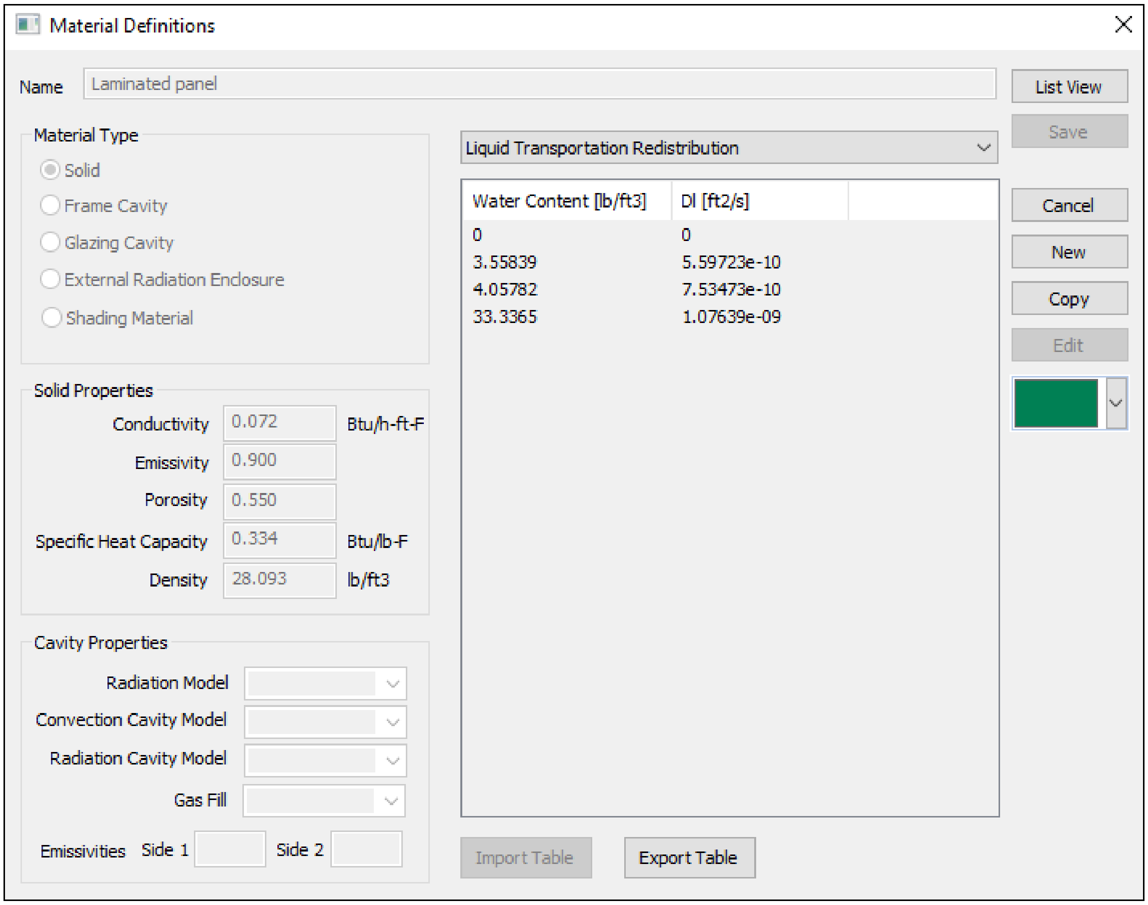
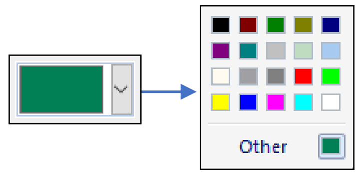


Figure 2-10. Material Detail View.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Used to access the List view of the Material Library.
Save	Saves any changes made to the Materials XML file.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit

	the new record as needed.
Edit	*Currently not implemented.*
Color	The color assigned to the material. It can be changed by clicking on the arrow to the right of the color box, selecting another color, and then clicking Save to save the changes to the record.  *Figure 2-11. Color Settings in the Material Library Detail View.*

Input Values

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the material.
Material Type	
	The material type, from the options of ▪ Solid ▪ Frame Cavity ▪ Glazing Cavity *(currently not implemented for HygroThermFEM)* ▪ External Radiation Enclosure *(currently not implemented for HygroThermFEM)* ▪ Shading Material *(currently not implemented for HygroThermFEM)*
Solid Properties *These input values are for Material Type = Solid*	
Conductivity	Conductivity of the material. **Units:** Btu/h-ft-F (IP); W/m-K (SI).
Emissivity	Emissivity of the material. **Units:** unitless
Porosity	Porosity of the material. **Units:** unitless
Specific Heat Capacity	Specific Heat Capacity of the material. **Units:** Btu/lb-F (IP); J/kg-K (SI).
Density	Density of the material.

	Units: lb/ft3 (IP); kg/m3 (SI).
Cavity Properties *These input values are for Material Type = Frame Cavity*	
Radiation Model	The Frame Cavity Radition Model, from the following choices: ▪ CEN ▪ ISO 15099
Convection Cavity Model	*Currently not implemented for HygroThermFEM.*
Radiation Cavity Model	*Currently not implemented for HygroThermFEM.*
Gas Fill	The gas fill in the cavity, from the following choices: ▪ Air ▪ Argon ▪ Krypton ▪ Xenon
Emissivites Side 1	*Currently not implemented for HygroThermFEM.*
Emissivities Side 2	*Currently not implemented for HygroThermFEM.*
Liquid Transportation Redistribution	

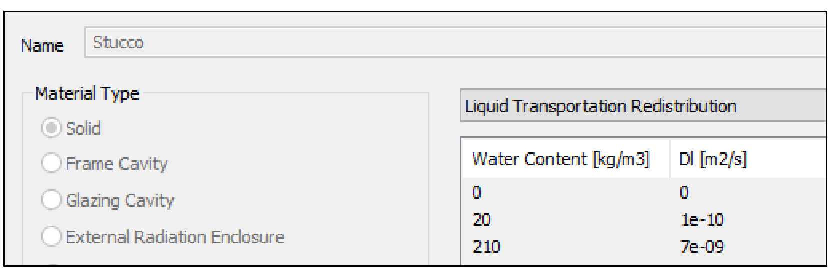


Figure 2-12. Liquid Transportation Redistribution variables for Stucco material.

Water Content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Dl	Liquid transportation coefficient. Units: ft2/s (IP); m2/s (SI).
Import File	To import data into this function, create a CSV file with the two values separated by a

Format

comma. The values (always in SI units) are:

- Water Content
- Dl (liquid transportation coefficient)

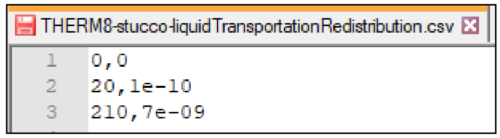


Figure 2-13. Import CSV file format for Liquid Transportation Redistribution variables, always in SI units.

Liquid Transportation Suction

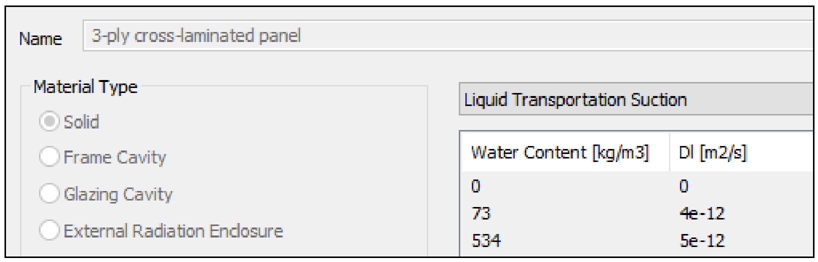


Figure 2-14. Liquid Transportation Suction variables for 3-ply cross-laminated panel material.

Water Content

Water content.

Units: lb/ft3 (IP); kg/m3 (SI).

Dl

Liquid transportation coefficient.

Units: ft2/s (IP); m2/s (SI).

Import File Format

To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are:

- Water Content (kg/m3)
- Dl (liquid transportation coefficient, m2/s)

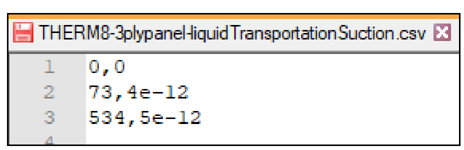


Figure 2-15. Import CSV file format for Liquid Transportation Suction variables for 3-ply cross-laminated panel material, always in SI units.

Moisture Dependent Thermal Conductivity

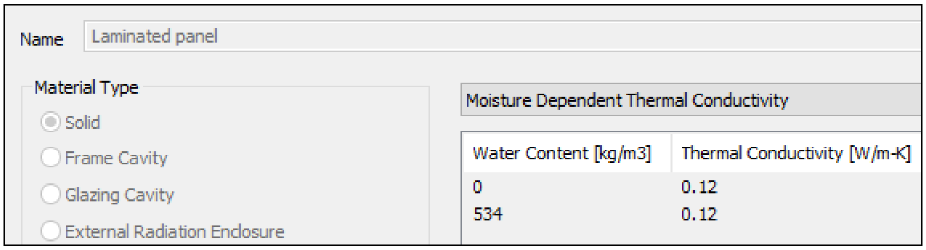


Figure 2-16. Moisture Dependent Thermal Conductivity variables for Laminated Panel material.

Water Content

Water content.

Units: lb/ft3 (IP); kg/m3 (SI).

Thermal Conductivity

Thermal conductivity.

Units: Btu/h-ft-F (IP); W/m-K (SI).

Import File Format

To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are:

- Water Content (kg/m3)
- Thermal Conductivity (W/m-K)

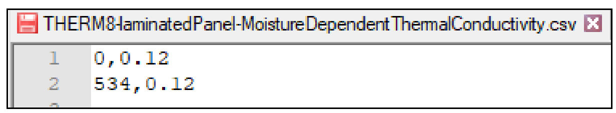


Figure 2-17. Import CSV file format for Moisture Dependent Thermal Conductivity variables for Lamianted Panel material, always in SI units.

Moisture Storage Function

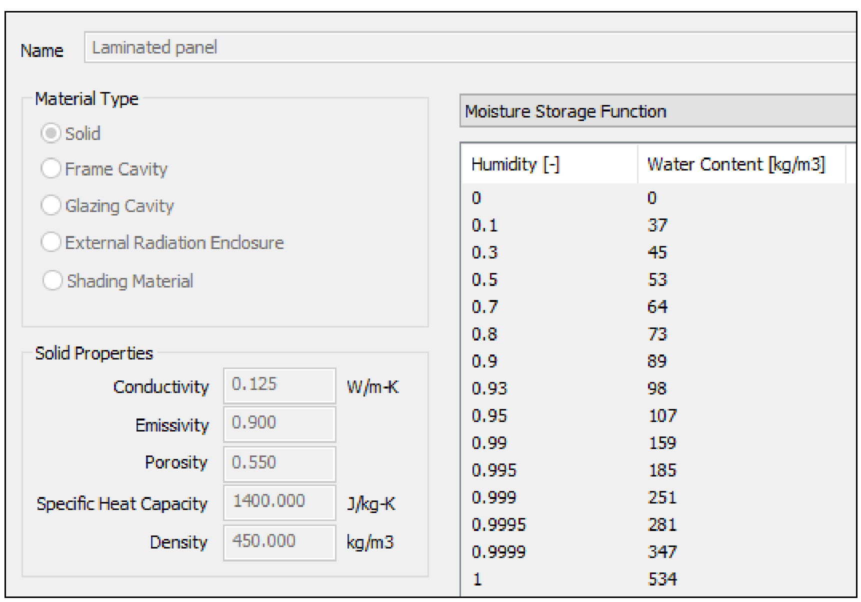


Figure 2-18. Moisture Storage Function variables for Laminated Panel material.

Humidity	Humidity fraction. Legal values: 0 – 1. Units: unitless.
Water content	Water content. Units: lb/ft3 (IP); kg/m3 (SI).
Import File Format	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Humidity (fraction, 0-1) ▪ Water Content (kg/m3)

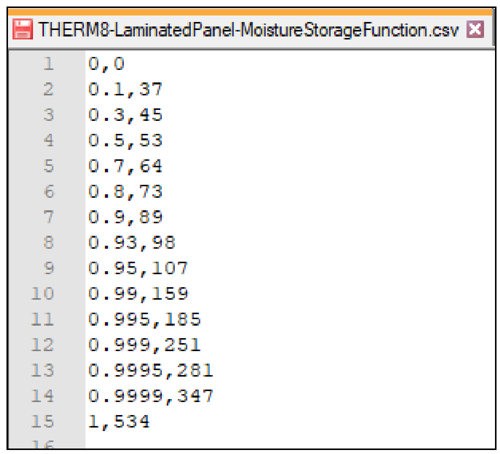


Figure 2-19. Import CSV file format for Moisture Storage Function variables for Lamianted Panel material, always in SI units.

Temperature Dependent Thermal Conductivity

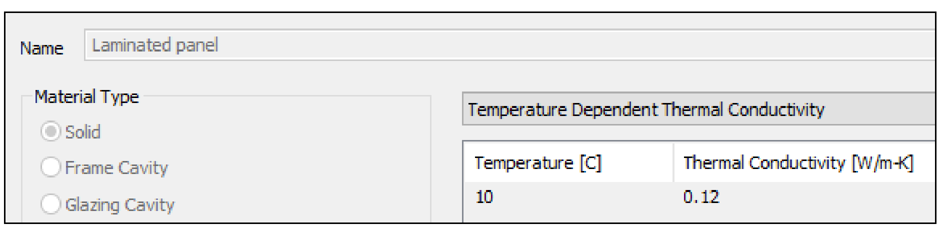


Figure 2-20. Temperature Dependent Thermal Conductivity variables for Laminated Panel material.

Temperature	Temperature. Units: F (IP); C (IP).
Thermal Conductivity	Thermal conductivity. **Units:** Btu/h-ft-F (IP); W/m-K (SI).

Import File Format	To import data into this function, create a CSV file with the two values separated by a comma. The values (always in SI units) are: ▪ Temperature (C) ▪ Thermal Conductivity (W/m-K)

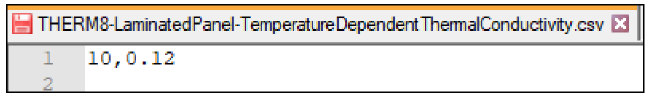


Figure 2-21. Import CSV file format for Temperature Dependent Thermal Conductivity variables for Lamianted Panel material, always in SI units.

File Format and Location

The Material Library is stored in an XML file called “Materials.xml” which is located in the “lib” subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.0\lib

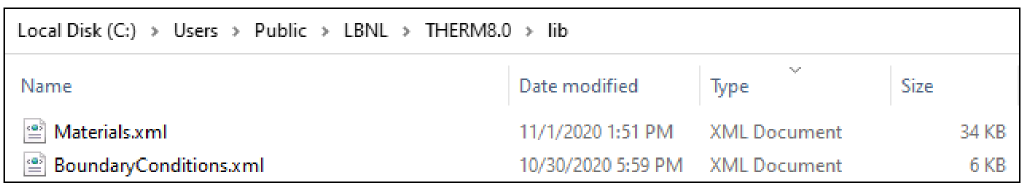


Figure 2-22. The Materials.xml file is located in the “lib” subfolder of the THERM working directory.

The definitions of all the materials displayed in the THERM Material Library are stored in this XML file.


```xml
<Materials>
  <SolidMaterial>
    <UUID>8dd145d0-5f30-11ea-bc55-0242ac130003</UUID>
    <Name>Laminated panel</Name>
    <Protected>true</Protected>
    <DefaultThickness>0.01</DefaultThickness>
    <MaterialInformation>NA</MaterialInformation>
    <BulkDensity>450</BulkDensity>
    <Porosity>0.55</Porosity>
    <SpecificHeatCapacityDry>1400</SpecificHeatCapacityDry>
    <ThermalConductivityDry>0.125</ThermalConductivityDry>
    <Emissivity>0.9</Emissivity>
    <WaterVaporDiffusionResistanceFactor>203</WaterVaporDiffusionResistanceFactor>
    <Color>0x008054</Color>
    <MoistureStorageFunction>
      <TableValue>
        <x>0</x>
        <y>0</y>
      </TableValue>
      <TableValue>
        <x>0.1</x>
        <y>37</y>
      </TableValue>
```

Figure 2-23. The Materials.xml file contains all the values for all the material records in the Material Library.

Making New Material Library Records

Using either the New or the Copy buttons in either the List or Detail View, it is possible to create new records. In the Detail View, the new or copied record value can be changes, and then those changes saved by clicking the Save button. Those changes are then made in the Material.xml file.

For example, if a new "solid" material record is created, and values added for the Solid Properties section as well as the different functions on the right side, all those changes are then saved in the Materials.xml file.

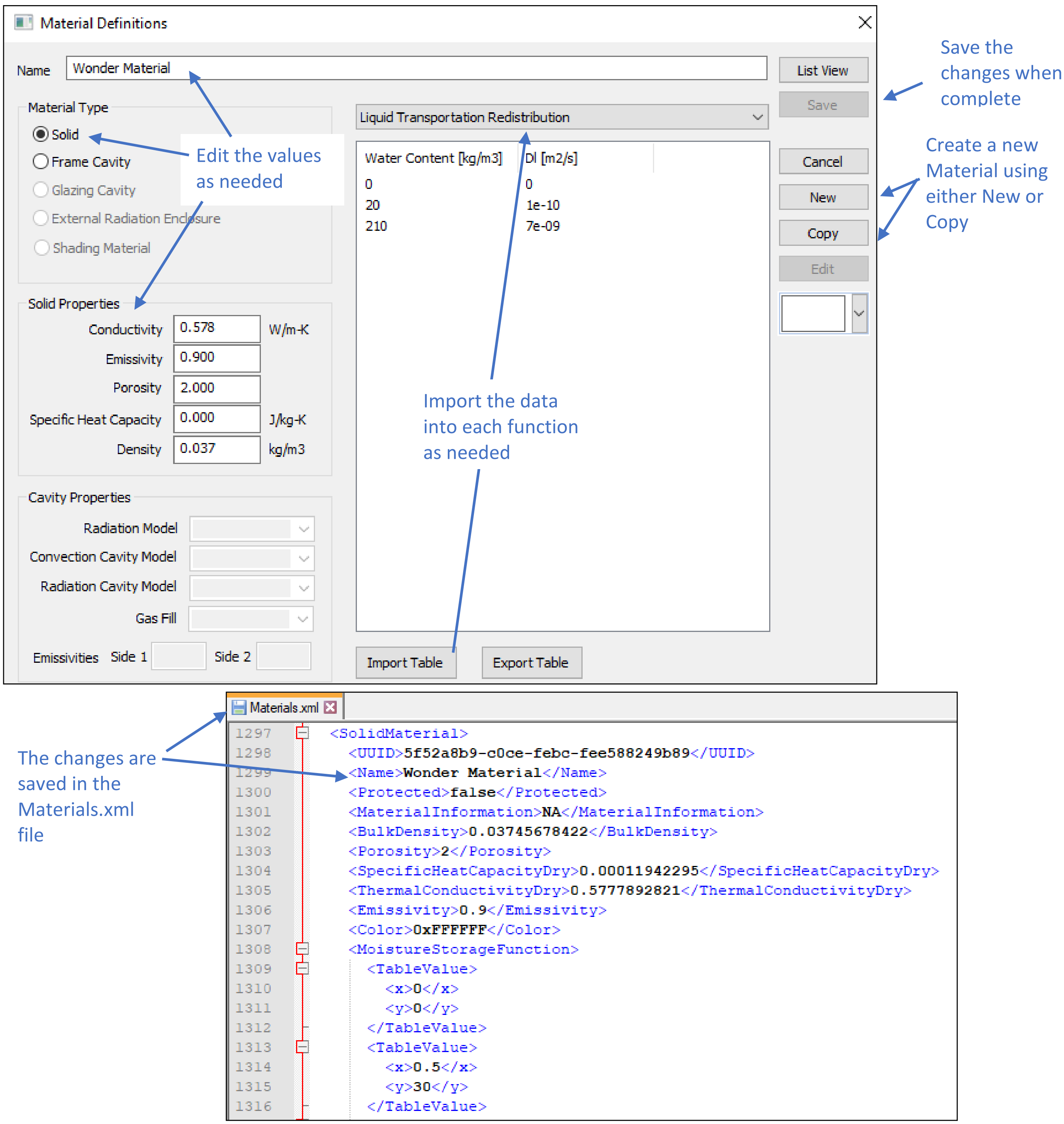


Figure 2-24. Make a new material using either New or Copy and change the input values as needed.

2.4. Boundary Condition Library

The Boundary Condition Library is different depending on the Simulation Engine selected.

- ConRad: If the ConRad simulation engine is selected, the Material library is the same as in previous versions of THERM.
- HygroThermFEM: If the HygroThermFEM simulation engine is selected, the Material library is much different.

2.4.1. Boundary Condition Library List View

Selecting the Boundary Condition choice from the Libraries menu when the HygroThermFEM simulation model is selected displays the Boundary Condition records in "List View".

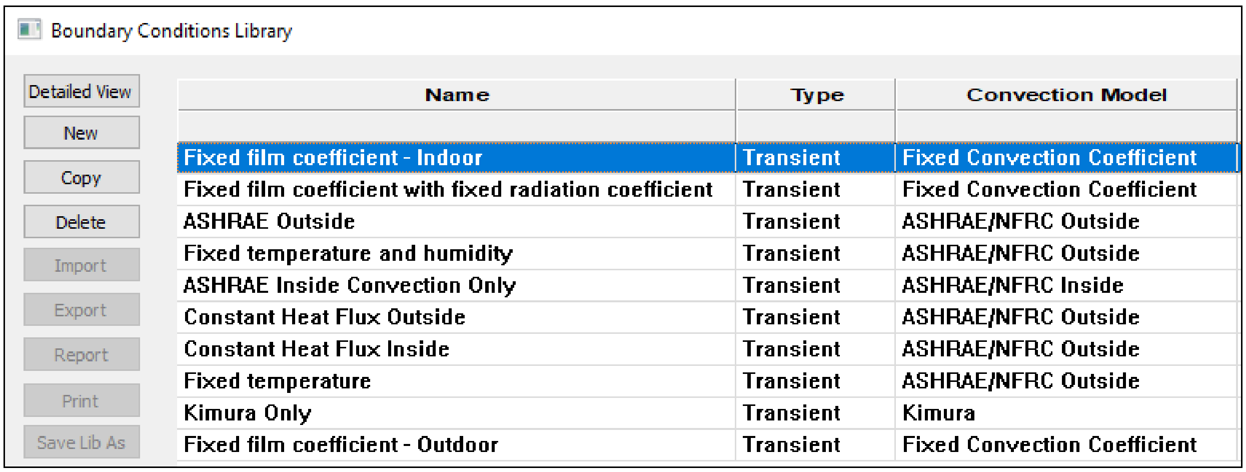


Figure 2-25. Boundary Condition List View.

Buttons

The buttons on the left hand side of the List View have the following functions:

Detailed View	Used to access the detailed view of the record, where the input values can be edited.
New	Used to make a new record, based on a default record. Click the New button, and the program will create a new record at the bottom of the list called "Default Name". Highlight that record and click the Detailed View button to edit the new record as needed.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Delete	Used to delete a record. Highlight the record to delete and click the Delete button.
Import	*Currently disabled.*
Export	*Currently disabled.*

Report	*Currently disabled.*
Print	*Currently disabled.*
Save Lib As	*Currently disabled.*
Load Lib	*Currently disabled.*

Column Headings

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the boundary condition.
Type	The boundary condition type, from the options of ▪ Transient ▪ Steady State (this is for the case where the transient moisture model is used with steady state algorithms, not currently implemented)
Convection Model	Options are: ▪ ASHRAE/NFRC Outside ▪ ASHRAE/NFRC Inside ▪ Fixed Convection Coefficient ▪ TARP ▪ Yazdanian Klems ▪ Kimura ▪ Montazeri
Temperature	For Type = Steady State, Convection Model Air Temperature. *Steady State is not currently implemented for HygroThermFEM.* **Units:** F (IP); C (SI)
Film Coefficient	For Type = Steady State, Convection Model Film Coefficient. *Steady State is not currently implemented for HygroThermFEM.* **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Heat Flux	For Type = Steady State, Convection Model Heat Flux. *Steady State is not currently implemented for HygroThermFEM.* **Units:** Btu/h-ft2 (IP); W/m2 (SI)

Solar Radiation	For Type = Steady State, Convection Model Solar Radiation. *Steady State is not currently implemented for HygroThermFEM.* **Units:** Btu/h-ft2 (IP); W/m2 (SI)
Radiation Model	For the Neumann Boundary Condition Type, if the Radiation Checkbox is checked, the Radiation model will be one of the following: ▪ Automatic Enclosure Model ▪ Black Body Radiation ▪ Fixed Radiation Coefficient
Film Coefficient	For Type = Steady State, Radiation Model Film Coefficient. *Steady State is not currently implemented for HygroThermFEM.* **Units:** Btu/h-ft2-F (IP); W/m2-K (SI)
Temperature	For Type = Steady State, Radiation Model Radiation Surface Temperature. *Steady State is not currently implemented for HygroThermFEM.* **Units:** F (IP); C (SI)
Humidity	For Type = Steady State, Radiation Model Humidity fraction. *Steady State is not currently implemented for HygroThermFEM.* **Legal Values: 0-1** **Units:** unitless

2.4.1. Boundary Condition Library Detail View

In the List View, highlight a record and click the Detailed View button to see the detailed input values for each boundary condition.

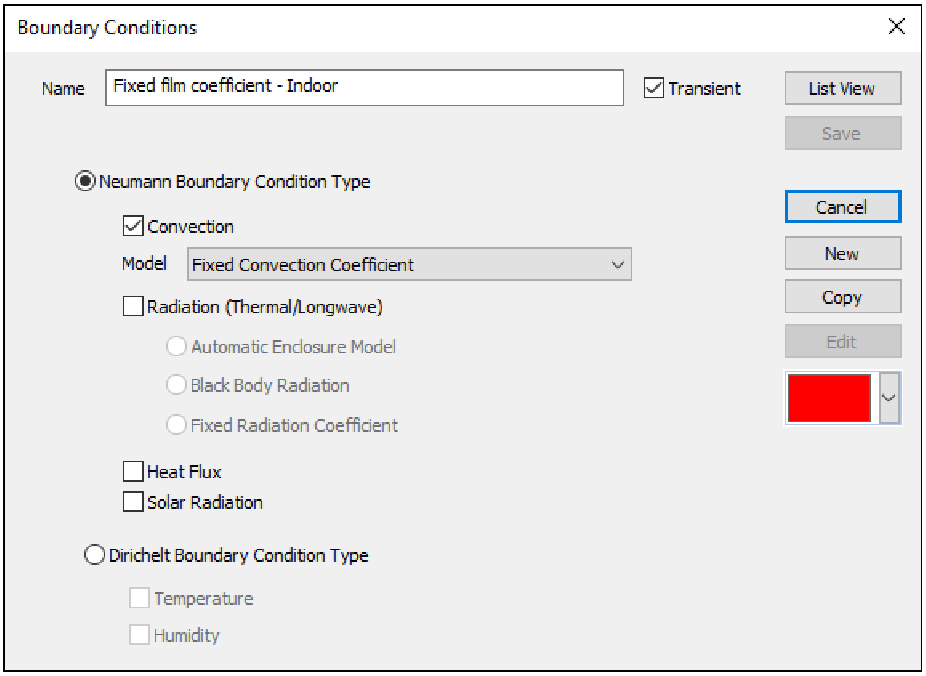
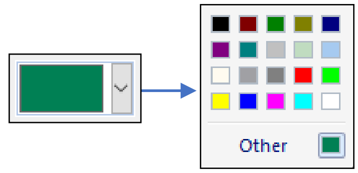


Figure 2-26. Boundary Condition Detail View.

Buttons

The buttons on the right hand side of the Detail View have the following functions:

List View	Used to access the List view of the Boundary Condition Library.
Save	Saves any changes made to the BoundaryConditions XML file. If changes are made to a field, move off the field in order for the Save button to become activated.
Cancel	Cancels any changes and returns to the List View.
New	Used to make a new record.
Copy	Used to copy an existing record (including name and all associated input values) into a new record. Highlight the record to be copied and click the Copy button. The new record will be added to the bottom of the list. Highlight that record and click the Detailed View button to edit the new record as needed.
Edit	*Currently not implemented.*

Color	The color assigned to the boundary condition. It can be changed by clicking on the arrow to the right of the color box, selecting another color, and then clicking Save to save the changes to the record.  *Figure 2-27. Color Settings in the Boundary Conditions Library Detail View.* One convention is to make the interior (warm) boundary conditions in the "red" colorway, and the exterior (cold) boundary conditions in the "blue" colorway.

Input Values

The List view contains columns with selected results (from the Detail view) for each record, as described below.

Name	The name of the boundary condition.
Transient	This is selected by default, and not editable in this version. The non-transient boundary conditions would be for the Steady State mode for the HygroThermFEM simulation engine, which has not been implemented yet.
Neumann Boundary Condition Type	
Convection	This option is checked if the boundary condition will include convection modeling.

Model	The Model pulldown list pertains to the type of convection model to be used, from the following choices:
	▪ ASHRAE/NFRC Outside
	▪ ASHRAE/NFRC Inside
	▪ Fixed Convection Coefficient
	▪ TARP
	▪ Yazdanian Klems
	▪ Kimura
	▪ Montazeri
	  ▪ For the Montazeri Convection Model, extra input values will be entered when assigning those to a specific boundary condition segment, which are
	    ▪ Building Width
	    ▪ Building Height

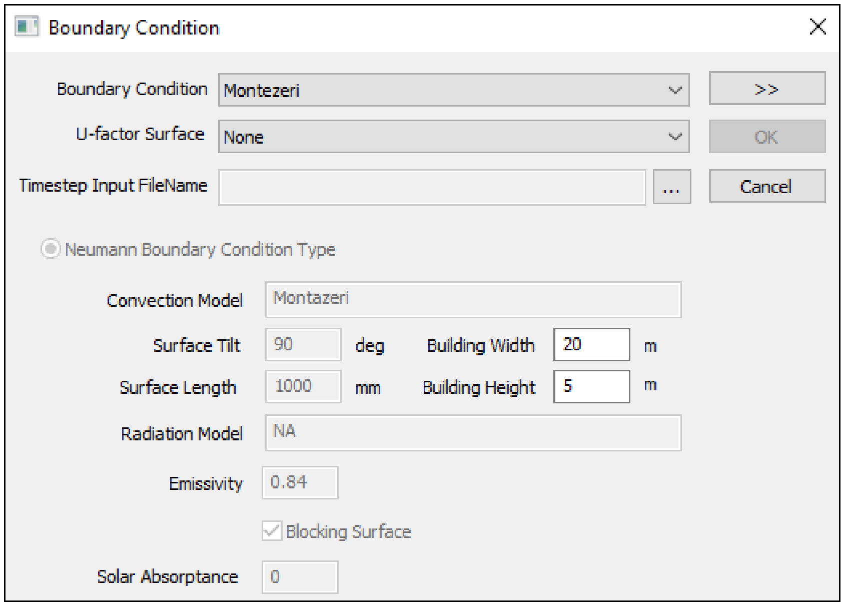
<table>
<tr><td></td><td>

Figure 2-28. Building Width and Building Height are entered for specific boundary condition segments when they are assigned the Boundary Condition Type = Montezeri.</td></tr>
<tr><td>Radiation</td><td>This option is checked if the boundary condition will include Radiation modeling

The Radiation model options are:
Automatic Enclosure Model
Black Body Radiation
Fixed Radiation Coefficient</td></tr>
<tr><td>Heat Flux</td><td>This option is checked if the boundary condition will include Heat Flux modeling</td></tr>
<tr><td>Solar Radiation</td><td>This option is checked if the boundary condition will include Solar Radiation modeling</td></tr>
<tr><td colspan="2">Dirichelt Boundary Condition Type</td></tr>
<tr><td>Temperature</td><td>Check this option to include Temperature in this Boundary Condition Type</td></tr>
<tr><td>Humidity</td><td>Check this option to include Humidity in this Boundary Condition Type</td></tr>
</table>

File Format and Location

The Boundary Condition Library is stored in an XML file called "BoundaryConditions.xml" which is located in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.0\lib

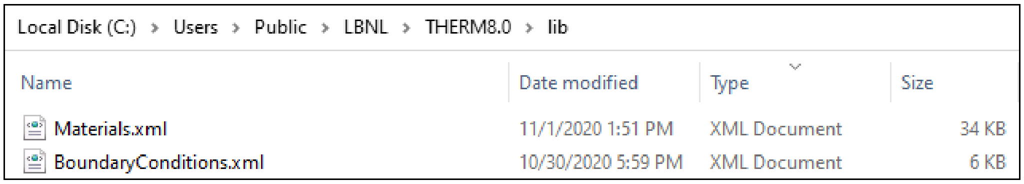
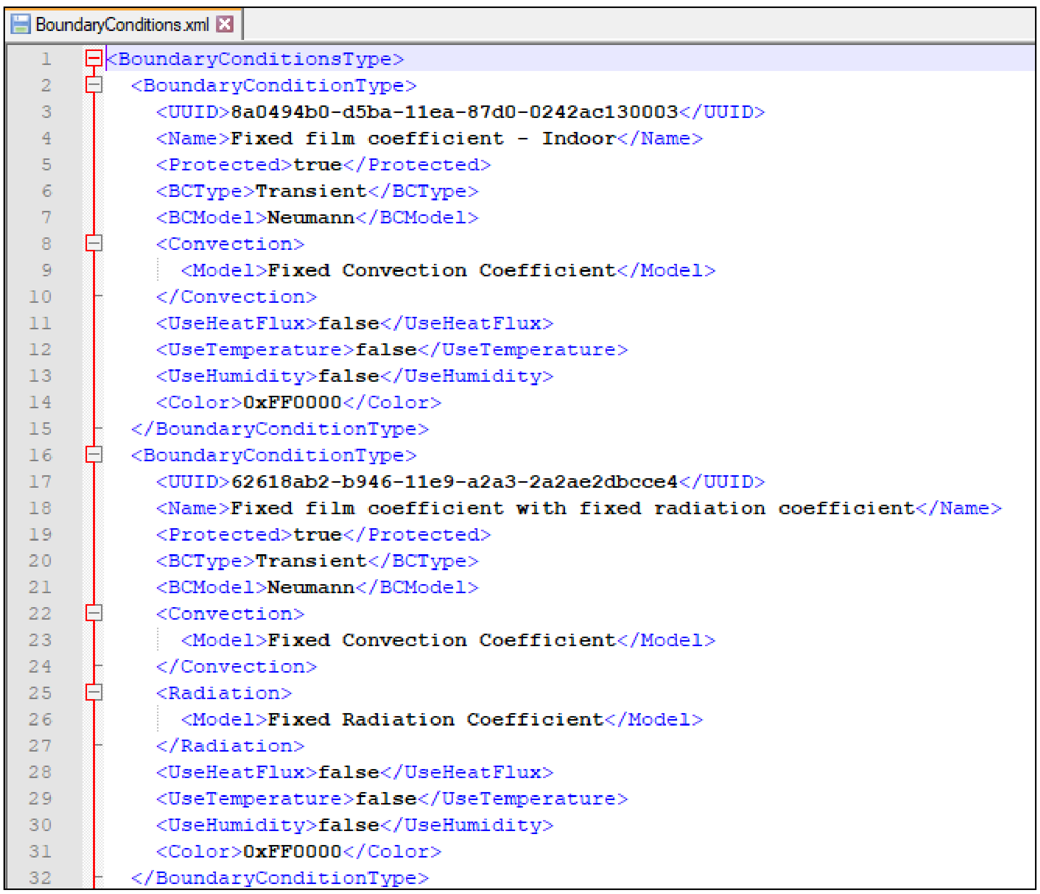


Figure 2-29. The BoundaryConditions.xml file is located in the "lib" subfolder of the THERM working directory.

The definitions of all the boundary conditions displayed in the THERM Boundary Condition Library are stored in this XML file.




```xml
<BoundaryConditionsType>
  <BoundaryConditionType>
    <UUID>8a0494b0-d5ba-11ea-87d0-0242ac130003</UUID>
    <Name>Fixed film coefficient - Indoor</Name>
    <Protected>true</Protected>
    <BCType>Transient</BCType>
    <BCModel>Neumann</BCModel>
    <Convection>
      <Model>Fixed Convection Coefficient</Model>
    </Convection>
    <UseHeatFlux>false</UseHeatFlux>
    <UseTemperature>false</UseTemperature>
    <UseHumidity>false</UseHumidity>
    <Color>0xFF0000</Color>
  </BoundaryConditionType>
  <BoundaryConditionType>
    <UUID>62618ab2-b946-11e9-a2a3-2a2ae2dbcce4</UUID>
    <Name>Fixed film coefficient with fixed radiation coefficient</Name>
    <Protected>true</Protected>
    <BCType>Transient</BCType>
    <BCModel>Neumann</BCModel>
    <Convection>
      <Model>Fixed Convection Coefficient</Model>
    </Convection>
    <Radiation>
      <Model>Fixed Radiation Coefficient</Model>
    </Radiation>
    <UseHeatFlux>false</UseHeatFlux>
    <UseTemperature>false</UseTemperature>
    <UseHumidity>false</UseHumidity>
    <Color>0xFF0000</Color>
  </BoundaryConditionType>
```

Figure 2-30. The BoundaryConditions.xml file contains all the values for all the material records in the Material Library.

Making New Boundary Condition Library Records

Using either the New or the Copy buttons in either the List or Detail View, it is possible to create new records. In the Detail View, the new or copied record value can be changes, and then those changes saved by clicking the Save button. Those changes are then made in the BoundaryCondition.xml file.

For example, if a new boundary condition record is created, all the settings for that boundary condition are saved in the BoundaryConditions.xml file.

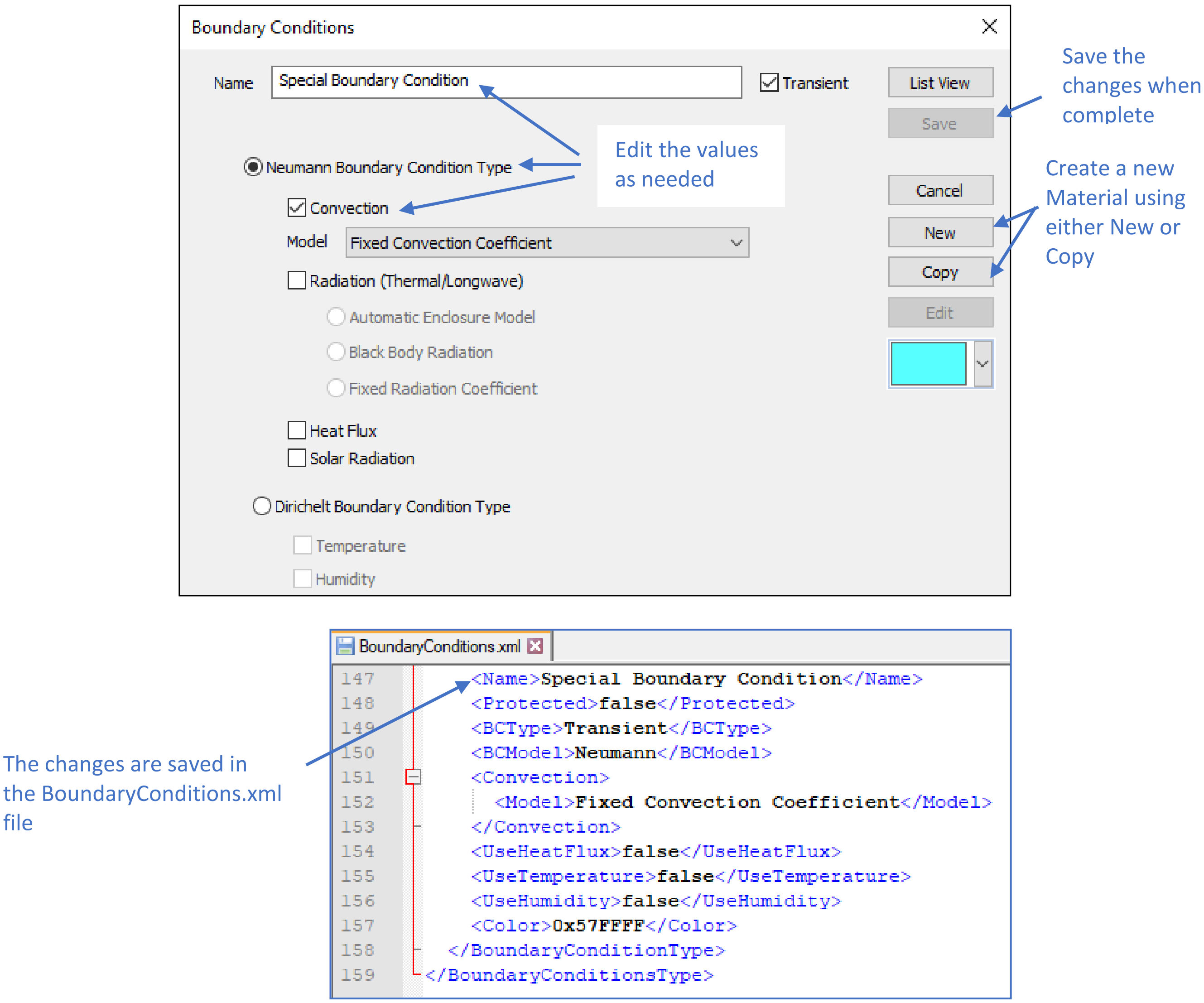


Figure 2-31. Make a new boundary condition using either New or Copy and change the input values as needed. The changes will be saved into the BoundaryConditions.xml file.

Time Step Boundary Condition Files

In addition to specifying a standard Boundary Condition for a boundary condition segment when using the transient simulation engine (HygroThermFEM), it is also necessary to specify a Time Step Boundary Condition, which are stored in separate XML files.

There are several examples of Time Step files that are installed with THERM 8, and they are found in the "lib" subfolder of the THERM working directory.

C:\Users\Public\LBNL\THERM8.0\lib

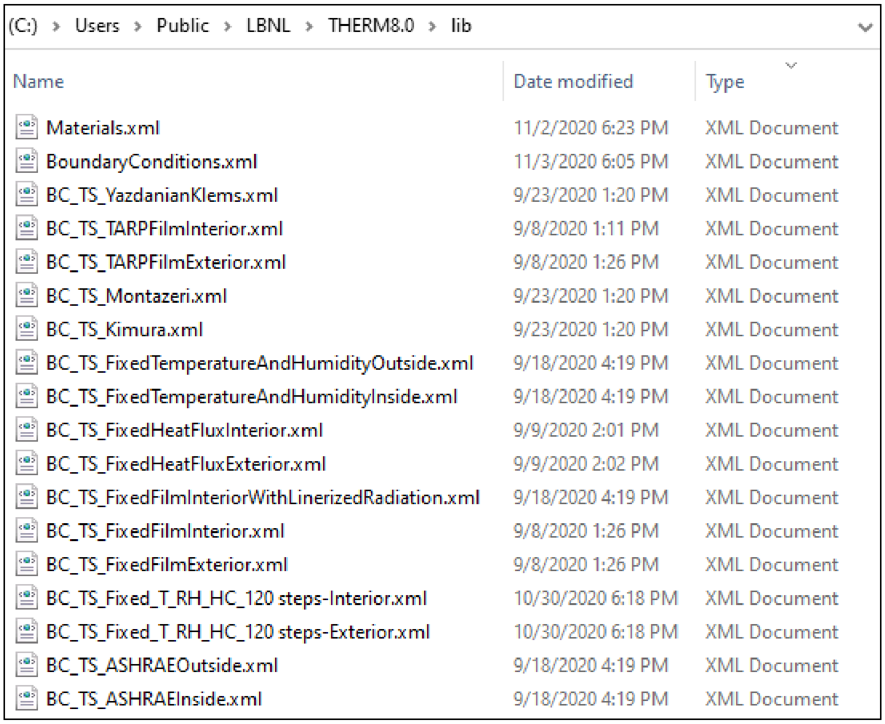


Figure 2-32. The Boundary Condition Time Step files are located in the "lib" subfolder of the THERM working directory.

These example files have a common naming convention of "BC_TS_" to indicate that they are **B**oundary **C**ondition **T**ime **S**tep files, but the files can have any name desired.

The Time Step files contain values for as many time steps as are being defined in each.

For example, the example file “BS_TS_FixedFilmInterior.xml” file has 5 timestep sections, with definitions for Temperature, Humidity and Film Coefficient for each time step.

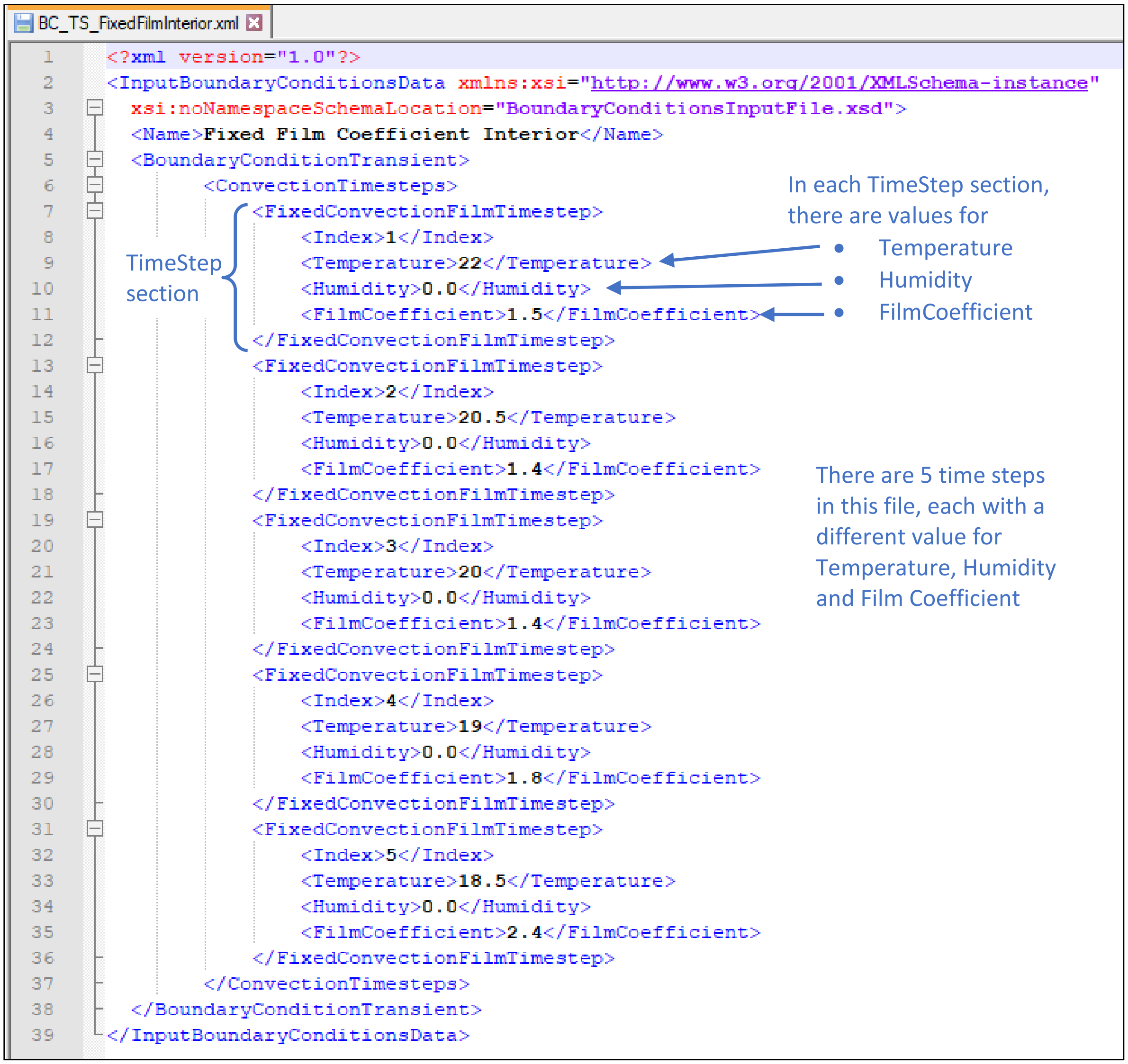


Figure 2-33. The Boundary Condition Time Step file format.

Applying Boundary Conditions to a Model

Applying boundary conditions to a model is similar to how they are applied in previous versions of THERM, with a few exceptions.

- After the model is completed, click the BC toolbar button to assign the boundary conditions. Intially, all the boundary conditions for the model will be set to “Adiabatic”.

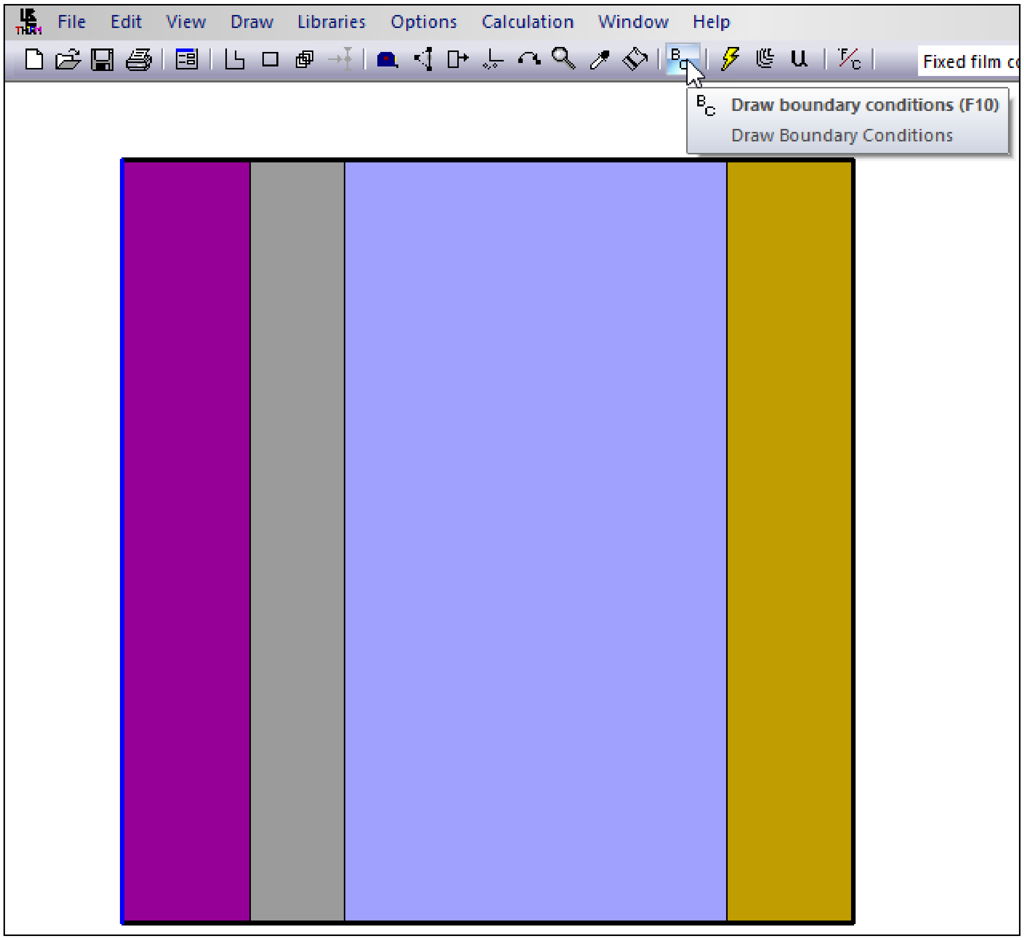


Figure 2-34. Click the BC toolbar icon to generate the intial boundary conditions for the model, which will initially all be set to Adiabatic (with a black color0.

- Select the desired boundary condition segments to assign boundary conditions to and either double click the left mouse button or press the Enter key, in order to open the Boundary Condition selection dialog box.
- Select the Boundary Condition from the pulldown list which displays the records from the Boundary Condition Library
- Select the U-factor Surface tag as applicable
- Select the Timestep Input Filename as applicable. The OK button will not be activated until a Timestep Input Filename has been specified

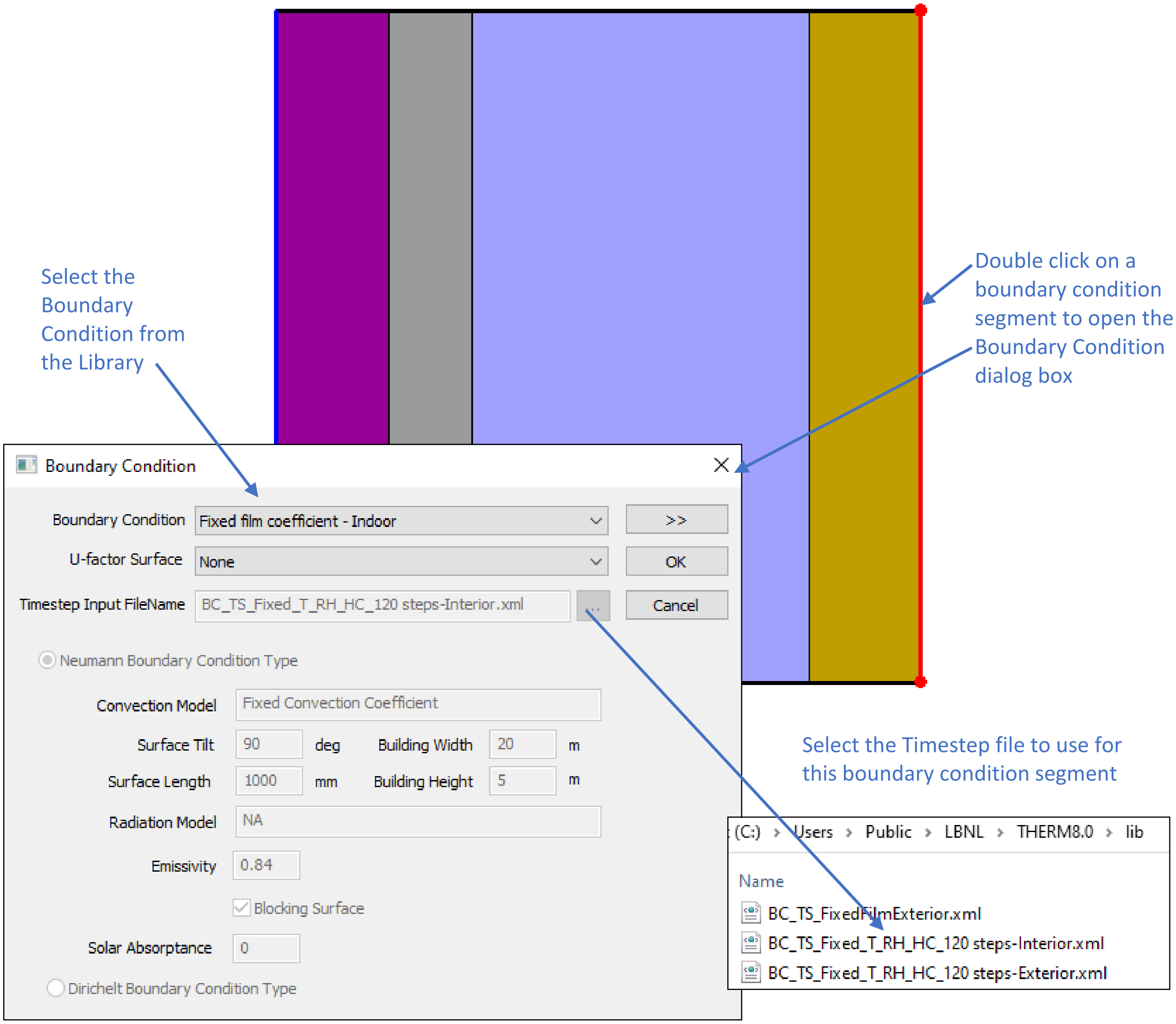


Figure 2-35. Specify both a Boundary Condition and a Timestep File for each boundary segment in a model.

2.5. Simulate the Model

When the model has been drawn, and materials and boundary conditions assigned, the model can be simulated by clicking the Calc toolbar icon, or selecting the Calculation/Calculation (F9) menu option.

In the example file below (from the sample file called "Stucco Wall – Moisture.thm"), the **File/Properties Time Step** is set to **3600 seconds** (1 hour), the **Number of Time Steps** is set to **100**, and the **QuadTree Mesh Parameter** is set to **4**.

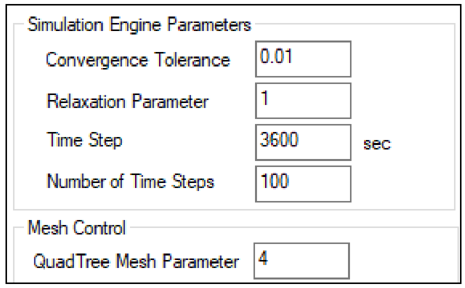


Figure 2-36. File/Properties setting for the sample file called "Stucco Wall – Moisture.thm".

The sample file Stucco Wall – Moisture.thm consists of five polygons.

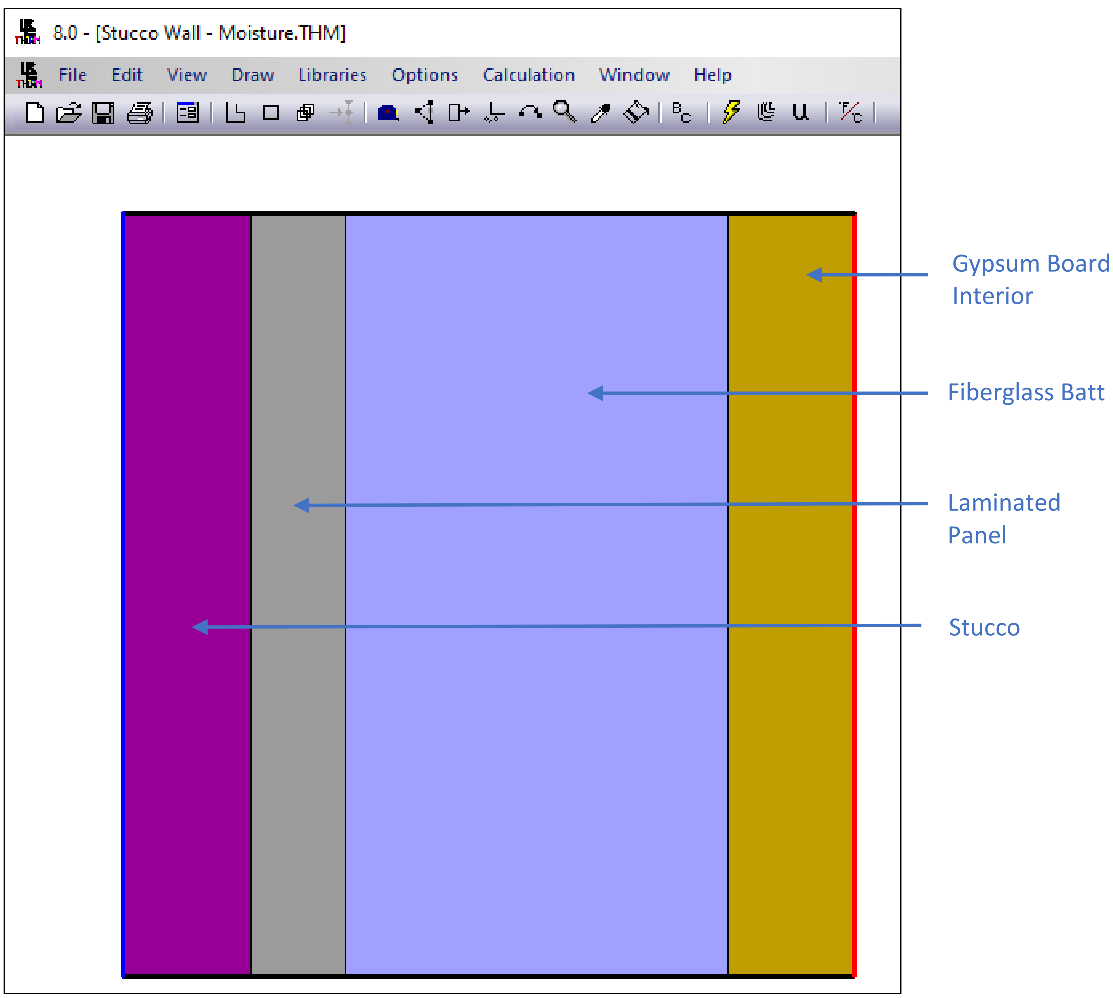


Figure 2-37. Stucco Wall – Moisture.thm.

The boundary condition definitions are shown below. The Timestep Input files have a timestep of 120, and the File/Properties Time Step value is set to 100, so the lower of the two timestep settings will determine the time step for the simulation, in this case 100 timesteps.

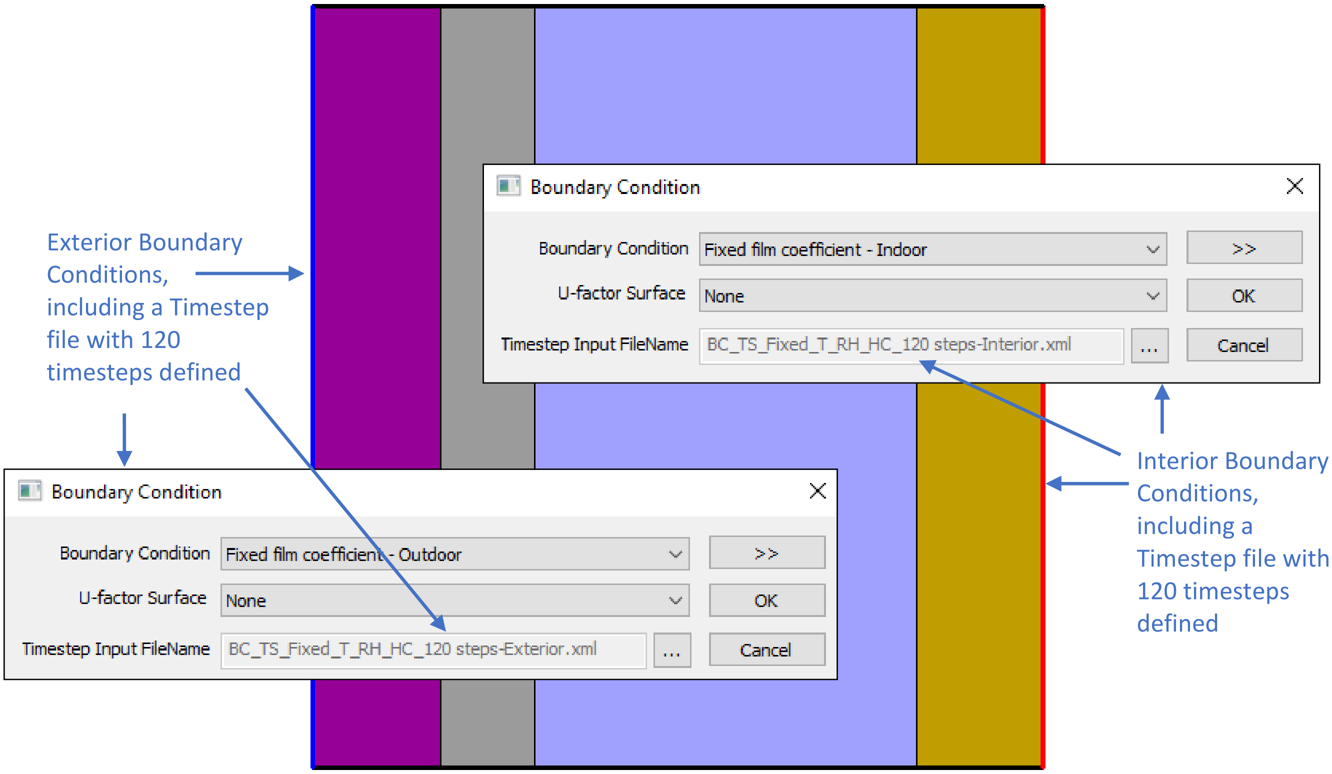


Figure 2-38. Boundary Condition definitions.

During the transient simulation, the program will display a small progress window showing the current timestep.

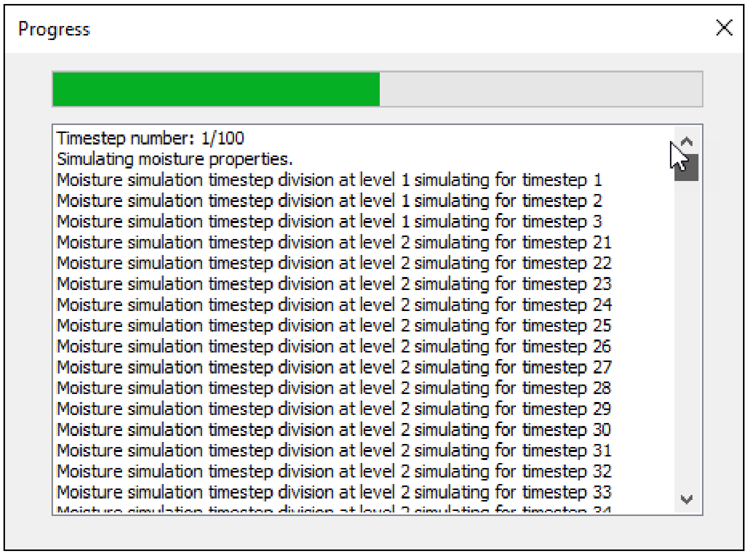


Figure 2-39. A progress window shows the status of the transient simulation.

2.6. View the Results

When the simulation has finished, the progress window will close.

To view the results, click the Calculation / Show Results menu option to open the THERM Visualization program. It may take a few seconds to appear.

This results visualization tool shows a color flooded image on the left and a graph of the results on the right.

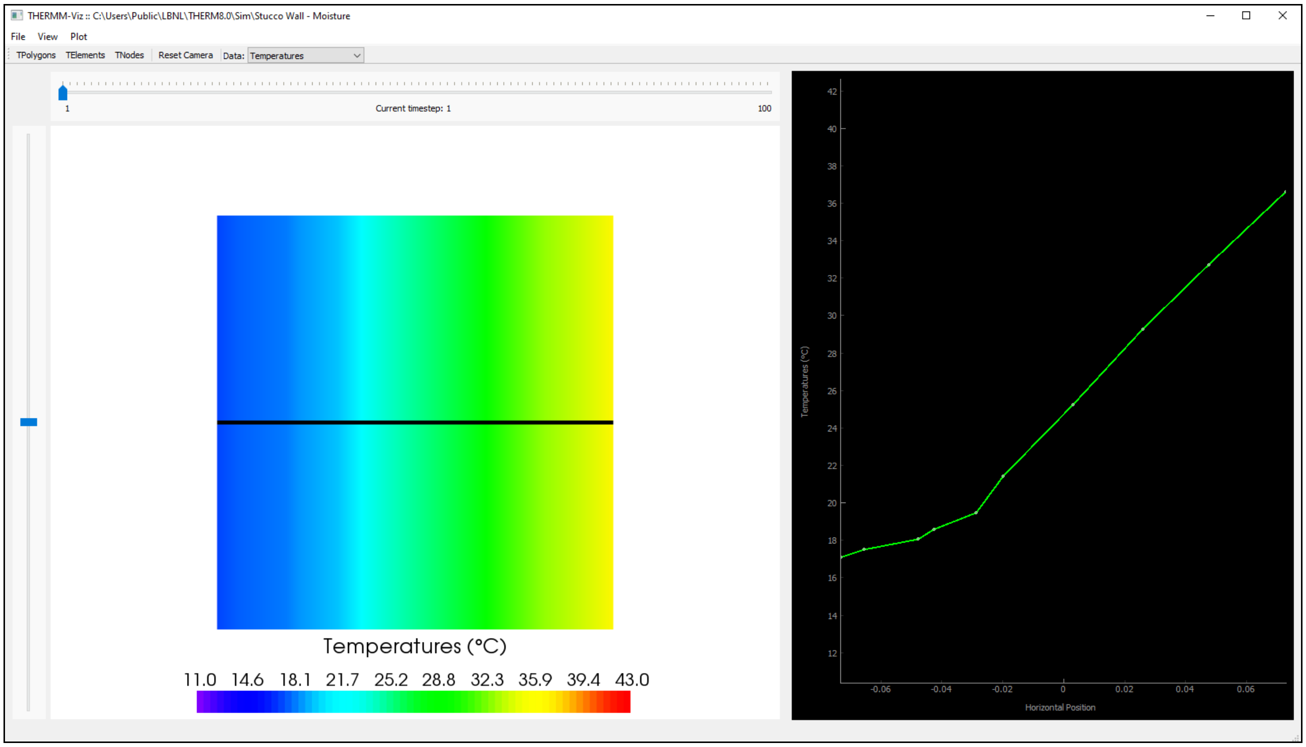


Figure 2-40. The THERM Visualization tool.

The slider at the top of the visualization tool allows results to be viewed at each timestep. There are also three different values that can be displayed, chosen from the Data pulldown list:

- Temperatures
- Humidities
- Water Content

The slider on the left side allows placement of the horiztonal location line, which determines the values displayed in the graph on the right hand side. This is useful when viewing results for a non-symmetrical model.

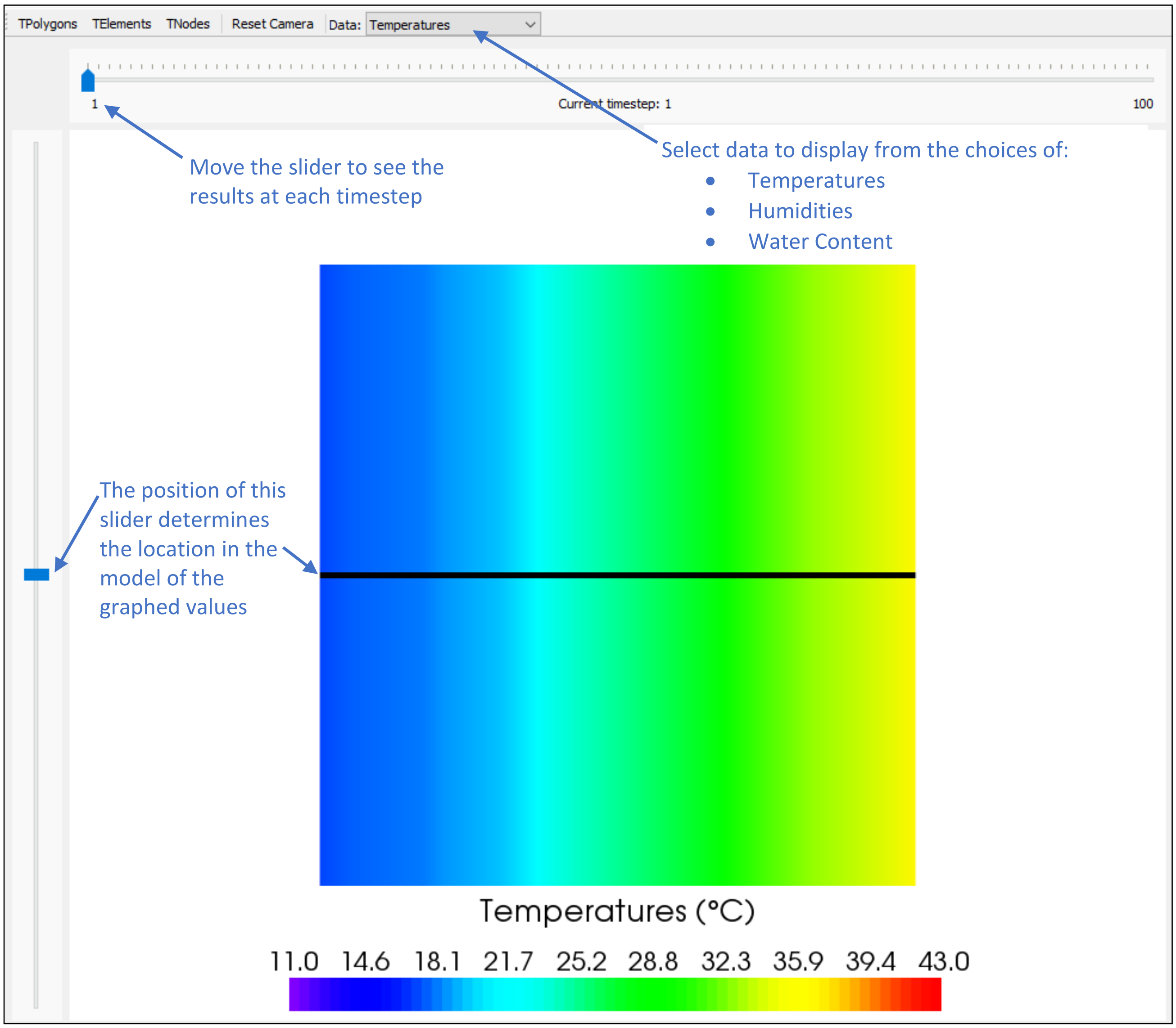


Figure 2-41. The THERM Visualization tool control options

.

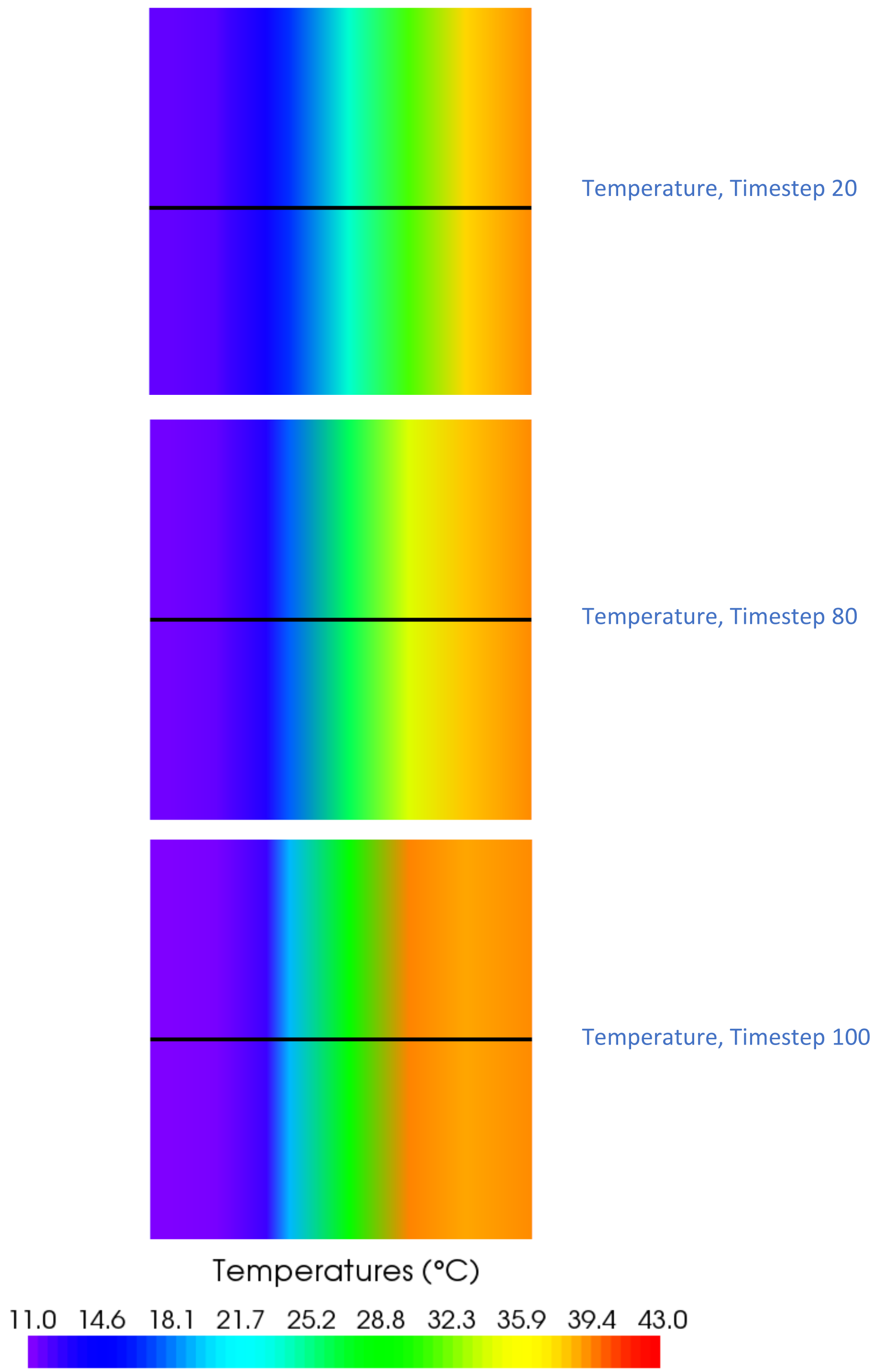


Figure 2-42. Temperatures over the timesteps

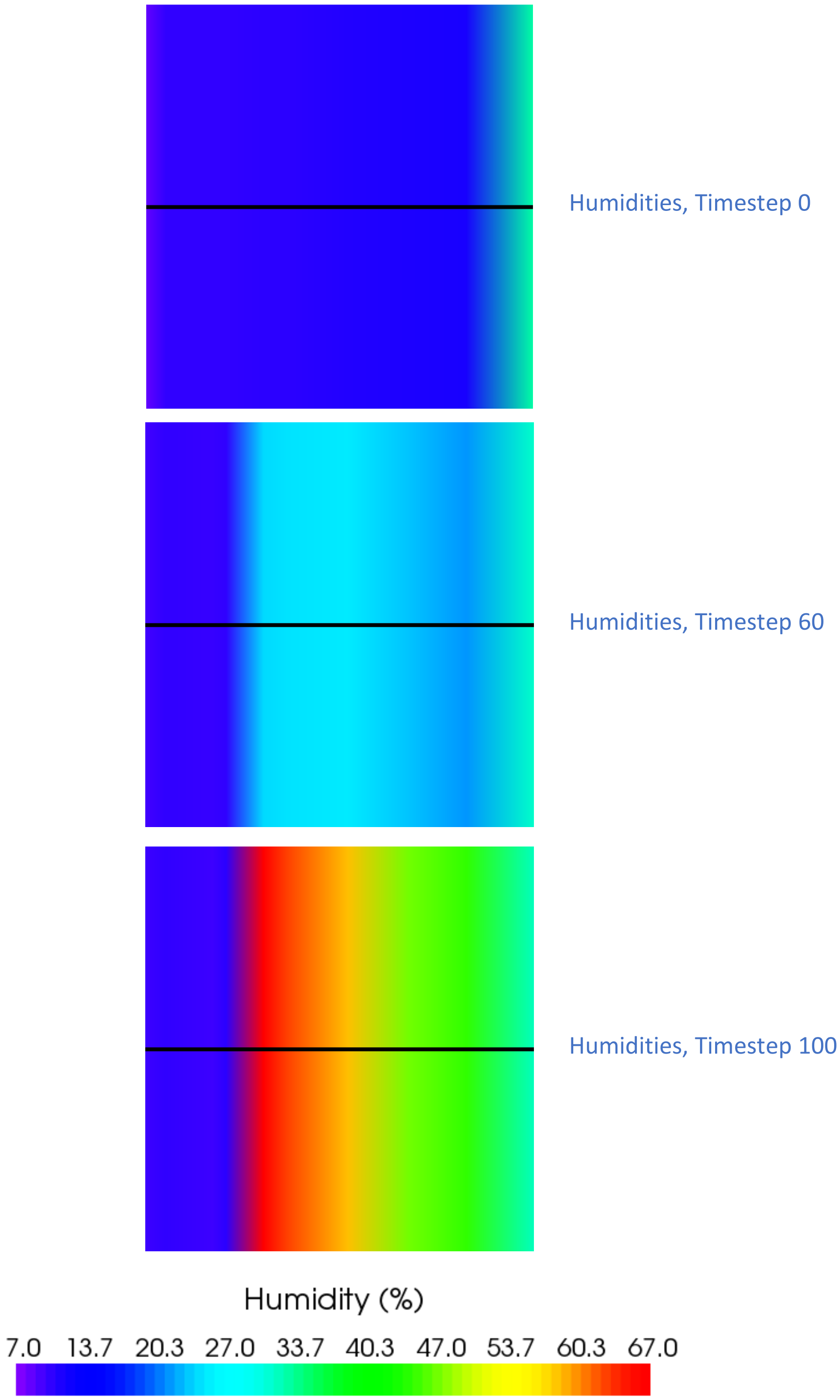


Figure 2-43. Humidities over the timesteps