Transient heat flow with Simu-Therm 8.0

Hilger & Daniel Software GmbH
www.simu-therm.de

Contents

Structure of this tutorial

This tutorial gives an introduction to the basic functionality of the Simu-Therm 8.0 transient heat flow simulation.

In two step by step tutorials you will learn how to enter a transient time schedule, perform the simulation, interpret the results and make printouts.

The two examples:

1. Heating schedule
2. Wall of a metallurgic furnace in cyclic operation

You can easily perform the described calculation examples on on your computer. We strongly recommend to do so.

After working through this tutorial, you will be able to to perform typical transient heat flow simulations.

What is a transient simulation ?

A **steady state heat flow** calculation is based on the assumption that the **temperatures stay constant** inside and outside over an infinite period of time. So do other parameters.
In a **transient simulation the temperature** inside and/or outside **is time dependent**.
The change of temperature on the surface induces a change of the temperature in the wall. The transient simulation calculates the resulting temperature in the wall over the simulation time period, starting from a defined initial temperature and ending with the temperature at the end of the simulation time.
During the simulation other important parameters, e.g. heat consumption and heat content, are recorded and available for further processing.

The steps to perform a transient simulation

1. Enter a steady state calculation (or select one in a project)

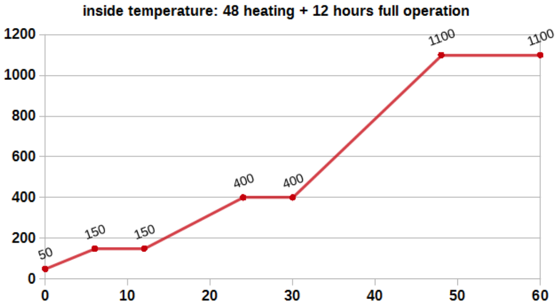
2. Define a time schedule of ambient conditions **on the wall surfaces**. In typical applications like heating simulation the inside temperature is the only figure which varies over time.
At least 2 points of the schedule need to be defined: Start ($time = 0$) and end ($time = t > 0$). Between the points of the schedule the temperature and other parameters are linearly interpolated.

3. Determine the initial temperature **in the wall**

4. Launch the simulation and visualize the results

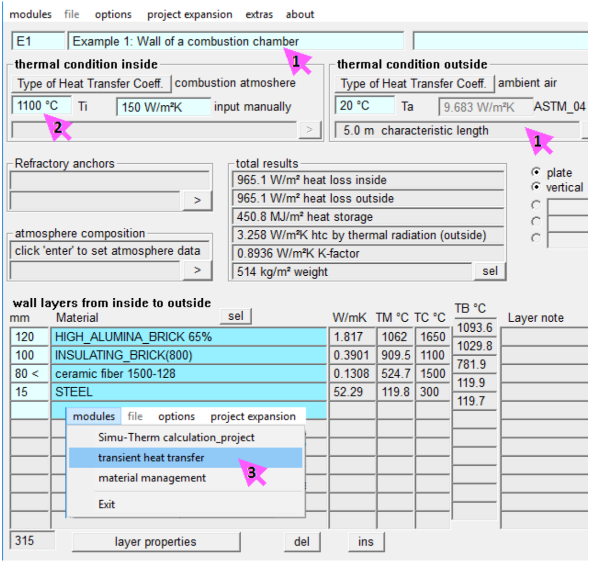
Example 1: Heating schedule 48 hours

In our sample project we collect the heat loss calculations for the lining of a cowper.
Combustion chamber and the checkerwork region require tube wall calculations, whereas the dome is a spherical wall.



Example 1: Enter the wall of the combustion chamber

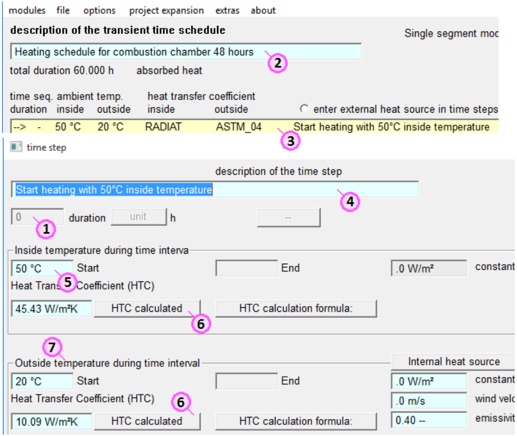
1. Our example is the plane wall of a combustion chamber, height 5 meters
2. The inside temperature will be overwritten in the transient simulation. We just choose the operation temperature 1100°C
3. In order to switch to the transient page, open the dropdown menu *'modules'* and select *'transient heat transfer'*



Enter step 1 of the heating schedule

1. Note that the field for the duration is greyed out for the first step.

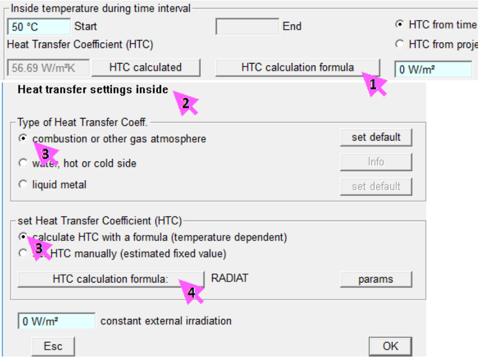
2. Enter a description of the schedule
3. Click on line 1 to switch to the time step dialog
4. Enter a description of the time step
5. Enter inside temperature **50 °C**
6. Click on *'HTC calculated'*
7. Heat transfer outside: Again select *'HTC calculated'*



Determine the type of inside heat transfer

First select the type of the HTC inside. Assume that the vessel is heated by a natural gas burner.

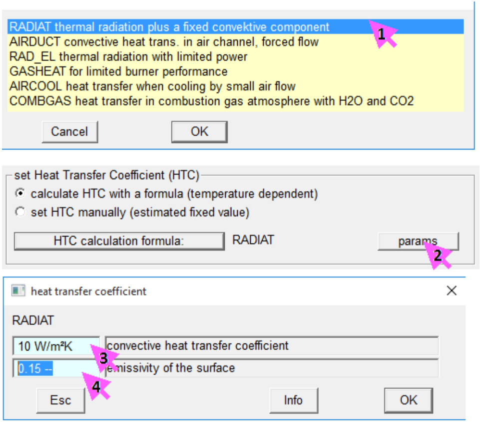
1. click on *'HTC calculation formula'* in order to open the dialog for the type of the inside heat transfer (2)
3. click on *'combustion or other gas atmosphere'* and on*'calculate HTC with a formula'*, because we will use a temperature dependent heat transfer
4. click on *'HTC calculation formula'* to select a suitable formula in the following dialog



Select the RADIAT formula

Depending on the HTC type you get a selection of applicable calculation formulas for the HTC.

1. select the formula *'RADIAT'*
2. click on *'params'* to enter the parameters of the formula.
3. enter $10\frac{W}{m^2K}$, a typical value for the convective heat teansfer to slow flowing air.
4. enter emissivity **0.15**. This is a typical emissivity for combustion atmospheres containing the radiating components CO2 and H2O.

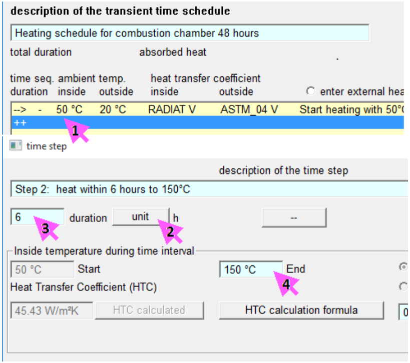


Enter step 2 of the heating schedule

Back to the transient main page, the first step (=first line) should look like in the image below.

Note that duration and final temperature are the only editable fields. Type and calculation formula cannot be changed during a time step with duration > 0

1. click on line **'++'** to open the time step dialog for the second step.
2. select time unit **'h'** (hours)
3. enter duration **'6 hours'**
4. enter temperature at the end of the time step: **150°C**

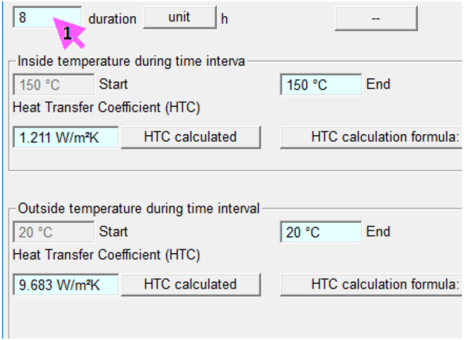


Enter step 3 of the heating schedule

Back to the transient main page, click on **'++'** again to enter another time step

In the third we keep the temperature **150 Cels** constant over **8 hours**

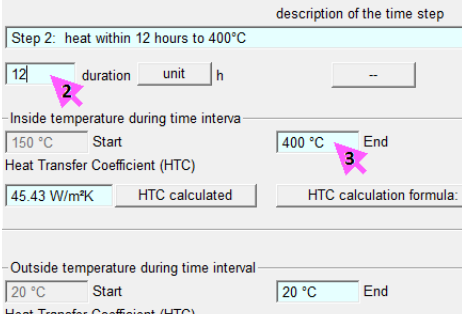
1. click on line **'++'** to open the time step dialog
2. enter duration **'8 hours'**
3. confirm the step with **OK**



Enter step 4 of the heating schedule

In step 4 we increase the temperature from 150°C to **400°C** within **12 hours**

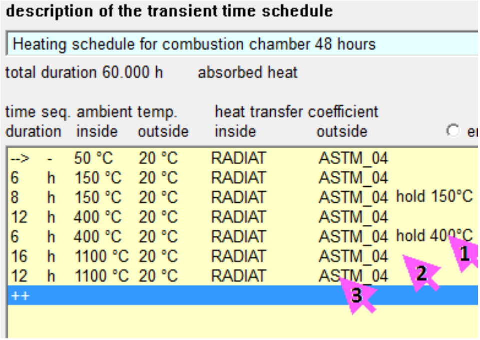
1. click on line **'++'** to open the time step dialog
2. enter duration **12 hours**
3. enter final temperature **400°C**
4. confirm the step with **OK**



Finish the heating schedule

Enter 3 time steps more to finish the heating schedule. Please procede like in the first 4 steps.

1. step 5: hold the temperature 400°C over 6 hours
2. step 6: raise the temperature from 400°C to the final temperature 1100°C within 12 hours. Although heating is finished, we add a time step to get closer to the steady state.
3. step 7: hold the temperature 1100°C over 12 hours

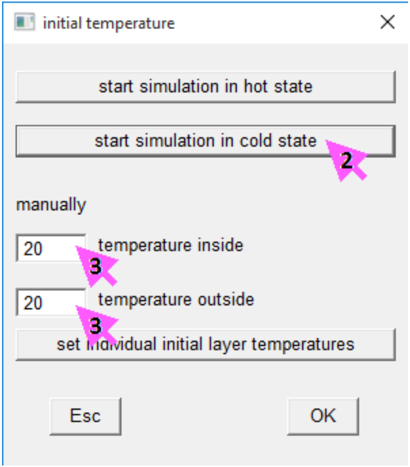


Define initial temperature in the wall

Before starting the simulation, make sure that the initial temperature of the wall is correct.

As we are simulating the heating of the wall, the inital wall temperature might be the ambient temperature 20°C.

1. in the main dialog click on **’initial state’** to open the dialog for the initial wall temperature.
2. click on **’start simulation in cold state’** to set the wall temperature to 20°C
3. we could start from any other steady state, determined by the inside and outside temperature.

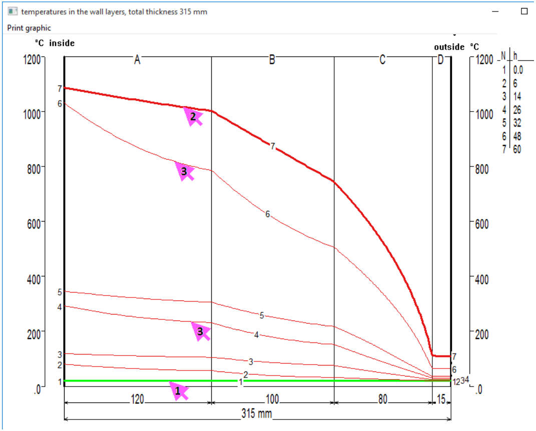


Launch the simulation

Launch the simulation **'calculate new'** - the following graph pops up

The graph shows the temperature in the wall at every tim step

1. green line = initial temperature (20°C)
2. fat red line = final temperature (close to steady state)
3. thin red lines = temperature curves at the end of the other time intervals, see table

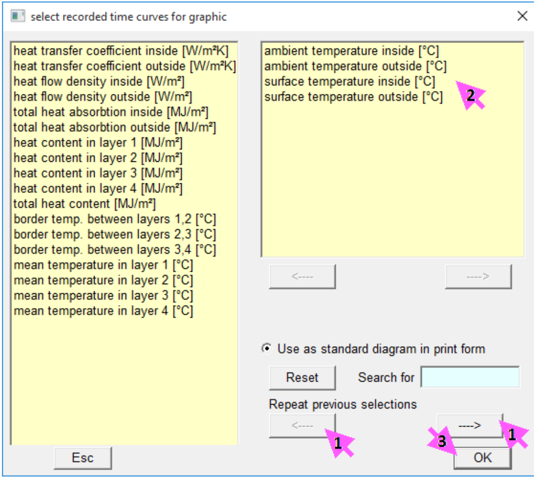


Dialog to select recorded curves for a time diagram

During the simulation SIMU-THERM records the values of important data over the simulation time.

You can select arbitrary groups of the data to visualize them in a time diagram.

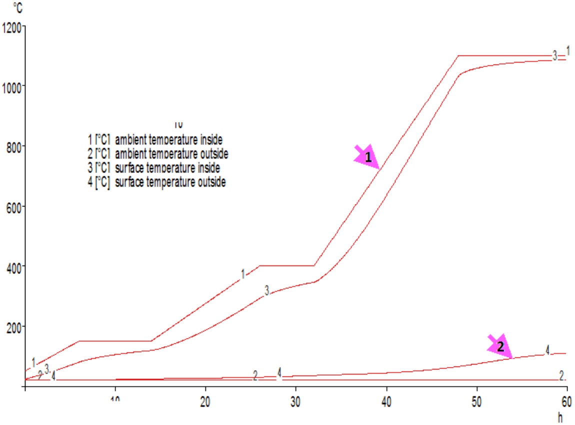
1. select a predefined data group with the arrows.
2. first of the predefined data groups: Ambient temperatures (user-defined) and surface temperatures (calculated)
3. click on **'OK'** to show the time diagram.



Ambient temperatures and surface temperatures

1. ambient temperature inside (= hot face temperature). This curve is useful to check if the heating schedule was entered correctly.

2. surface temperature outside (= shell temperature).

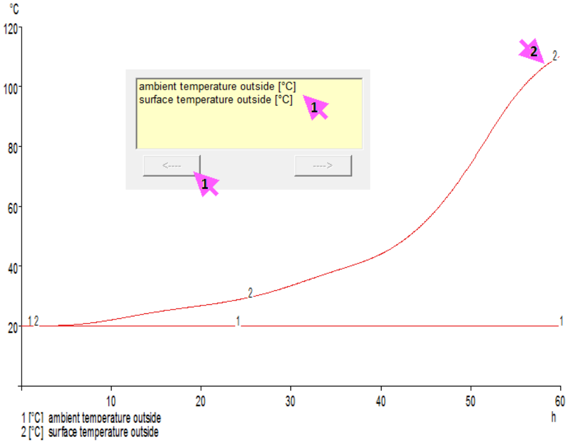


Shell temperature over time

In order to obtain a better scale for the shell temperature, we remove the inside temperatures from the graph.

1. mark the inside ambient temperature and remove it with the arrow. Then do the same with the inside surface temperature

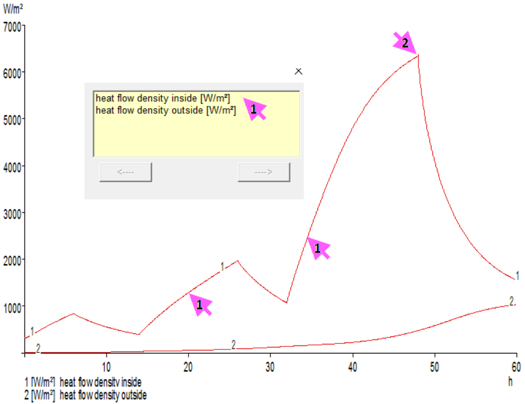
2. note that the shell temperature is still increasing after 60 hours



Heat flow density

The heat flow density is the amount of heat that passes the wall surface per m2. Unit is Watt/m2 = Joule/m2*s

1. the heat flow density provides information about the heater performance needed.
2. if the heater capacity was below 7 kW per m2, the temperature should be raised slower in the last heating interval



Heat absorption and heat content

The heat absorption per m2 is the accumulated heat absorbed by the wall. Unit is MJ/m2

1. the heat absorption provides information about the total amount of energy needed to heat up a furnace.
2. the heat content is close to the absorbed heat, so most of the absorbed energy is stored in the wall.