# Fire protect simulation with Simu-Therm 8.0

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

# Contents

# About this tutorial

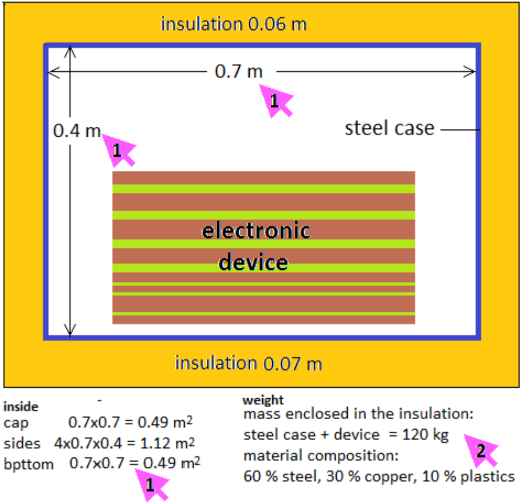
Simu-Therm 8.0 comes with a new expansion module for the simulation of fire protection which simplifies the procedure and extends the range of possible applications.
This tutorial enables you to perform simulation of fire protection independently. You learn the new procedure step by step in two example applications
1.) fire protection insulation of an electronic device in a box
2.) fire protection insulation of steel beam, e.g. in a building or on a ship

# Fire protect simulation with Simu-Therm 8.0

Fire protection of an object here means to surround the object with fire proof insulation 'walls' keeping heat from the object over a specified time period. The simulation is a mathematical model of the temperature change in the insulation and in the object to protect.
The simulation requires the following steps
1.) collect the sections of the insulation in a 'project file' (the data format of Simu-Therm)
2.) assign a contact area to every section of the insulation
3.) enter a curve of the outside temperature over the simulation period.
4.) optionally specify the heat capacity of the object to protect.

# Example 1: insulated box

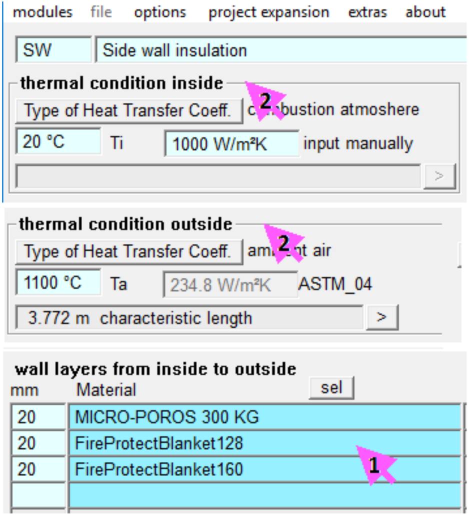
The first example is an electronic device enclosed in a rectangular steel case. The device has to be protected of a hydrocarbon fire with flame temperature 1100°C.
1. First we figure out the areas inside the insulation box
2. Next we need to know the total mass enclosed in the box. For the heat capacity of the mass it is important to know its components.



# Enter insulation section 'Side Wall'

We distinguish three insulation sections side wall, bottom, cap

1. layers of the side wall insulation Note that the **steel shell must not be included** as a layer, because it is considered as an internal mass.
2. we do not need to pay any attention to the heat transfer condition inside or outside. Simu-Therm will define standard heat transfer suitable for fire protect simulations

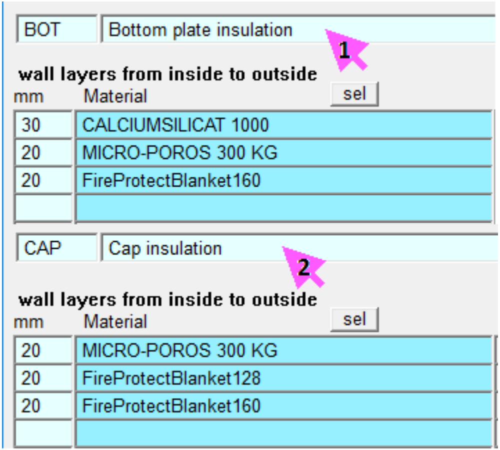


# Enter insulation sections 'Bottom' and 'Cap'

Enter the insulation section for the bottom and the cap of the box in the same way as before. Add descriptions in order not to mix up the sections.

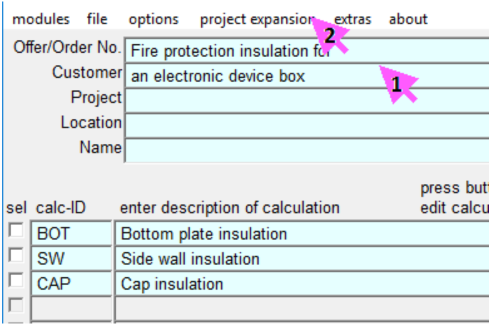
1. bottom plate insulation
2. cap insulation

Actually, the cap insulation is the same as the side wall, so it does not need to form a separate section. The extra section is just for demonstration.



# Add a project expansion

Switch to the project page. The project should look like on the screen shot.
1. enter project information lines
Up to now, our project has no expansion mode. At this point we will add the mode *'Fire protection inside of an insulated box'* to our project.
2. click on dropdown menu *project expansion*

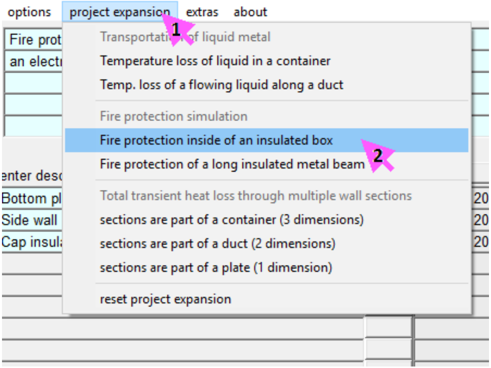


# Project expansion 'Fire Protection Box'

In the dropdown menu *project expansion* you ca choose between several expansions with very different features. Two expansions suit for fire protection.

1. select *'Fire protection inside of an insulated box'*

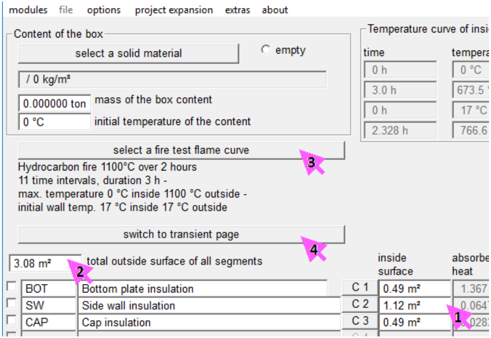
This is the expaansion for three dimensional objects like a box. For long thin objects like a beam can be treated as 2-dimensional profile with the second expansion. We will use it in the seond part of this tutorial.



# The fire protection page

The fire protection page controls working with the simulation. You can enter areas, select and modify a fire curve, define an internal mass, launch simulation and printing, view graphical results.

1. enter the inside areas of the insulation sections
2. enter the total outside area
3. click on *select a fire test flame curve* and select the file **FireCurveHydrocarbon1100Cels.STtime**
4. click on *switch to transient page* to view the transient time table



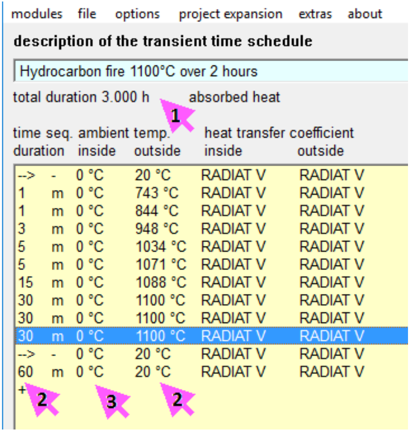
# The transient page for fire protection

For details about the transient page see tutorial *Tutorial_transient.pdf*
1. duration of the schedule is 3 hours

The ambient temperature raises rapidly from 20˚C to 1100˚C within 120 minutes. Then the ambient temperature drops back to 20˚C (fire stopped after 120 minutes). The simulation continues for 60 min to the temperature maximum in the box.

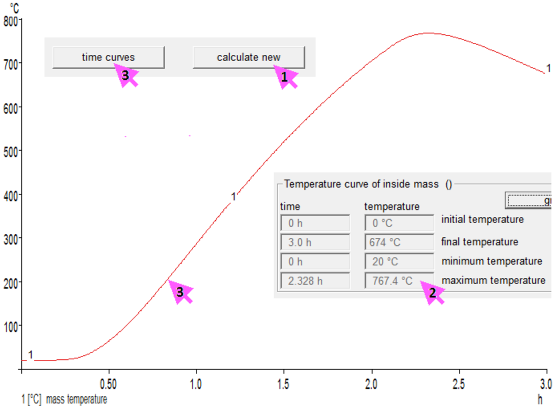
2. in case of fire protection, users can modify only the temperature curve, i.e. the columns time and temperature. To avoid mistakes, everything is else is automatically set by Simu-Therm.

3. e.g. the inside temperature is set to 0˚C.



# Simulate the empty insulation box

1. click on *calculate new* to launch the simulation of the empty box
2. the overview block shows that the maximum temperature in the box is 767°C, meaning that the insulation would not work if the box was empty.
3. click on *time curves* and in the time curve dialog on *OK* to get a graph of the 'mass temperature' (currently the mass is 0)

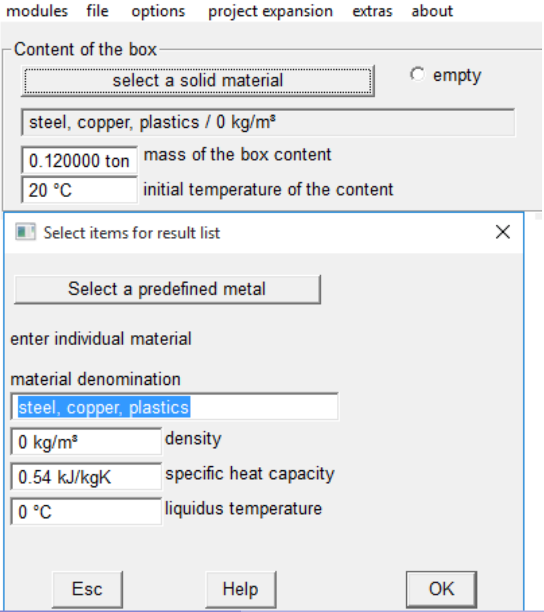


# Enter the enclosed mass

1. enter the weight **0.12 tons** (=120 kg)
2. enter the initial temperature **20°C**
3. click on  to open the material selection dialog

As we use a composed material, create an own material. The only relevant figure of the material is the specific heat capacity.

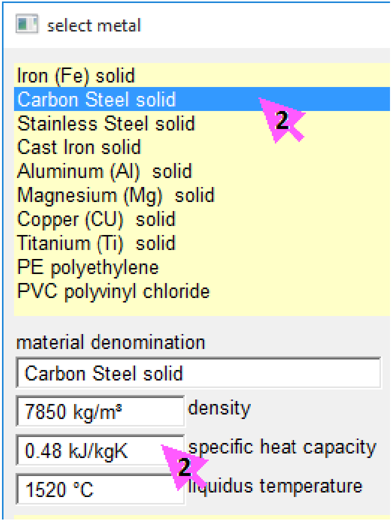
4. enter the specific heat capacity **0.54** $\frac{kJ}{kgK}$. The next slide shows how to get that value



# Calculate the specific heat of composed material

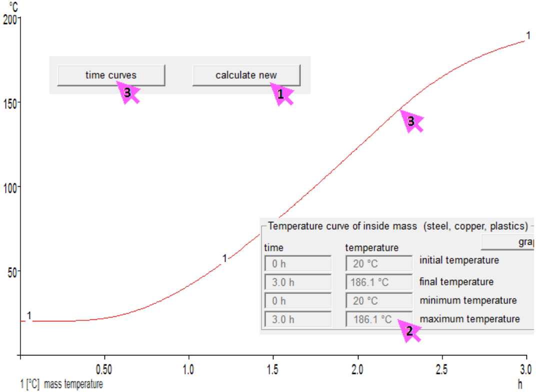
1. in the material selection dialog click on *select a solid material*
2. in order to look up the specific heat capacity of a material, select it from the list.

E.g you find 0.48 $\frac{kJ}{kgK}$ for carbon steel and 0.385 $\frac{kJ}{kgK}$ for copper.
1.4 $\frac{kJ}{kgK}$ is a typical value for plastics
For our composed material we can use the weighted average: $0.6 \times 0.48 + 0.3 \times 0.385 + 0.1 \times 1.4 = 0.5435 \; \frac{kJ}{kgK}$



# Simulate insulation box with mass

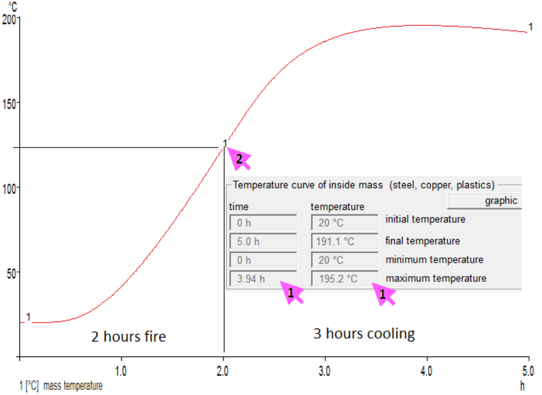
1. click on *calculate new* to launch the simulation again
2. this time maximum temperature in the box is 186°C, much better than for an empty box, but still too hot.
3. click on *time curves* for graph of the inside temperature. To find the maximum, we add 2 more hours of cooling time (1 hour fire, 3 hours cooling).



# Find the maximum temperature

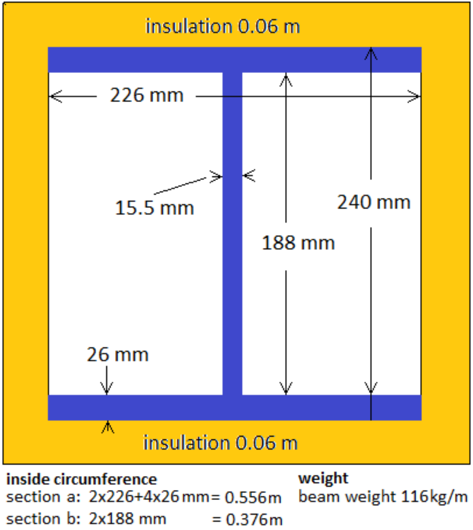
Simulation over 5 hours: 2 hours firing, 3 hours cooling

1. the maximum temperature of 195°C occurs almost 2 hours after the fire stopped.
2. be aware that the maximum temperature can be much higher than the temperature at the end of firing. (in this case 195°C vs 125°C)



# Example 2: insulated long steel beam

The second example is a long steel beam, protected against fire with a 60 mm fiber insulation.
1. First we enter the circumference inside the insulation. To stay flexible, we distinct 2 sections: a. in contact with the beam, b. not in contact.
2. The mass enclosed in the insulation (weight of the beam) is 116 kg/m.

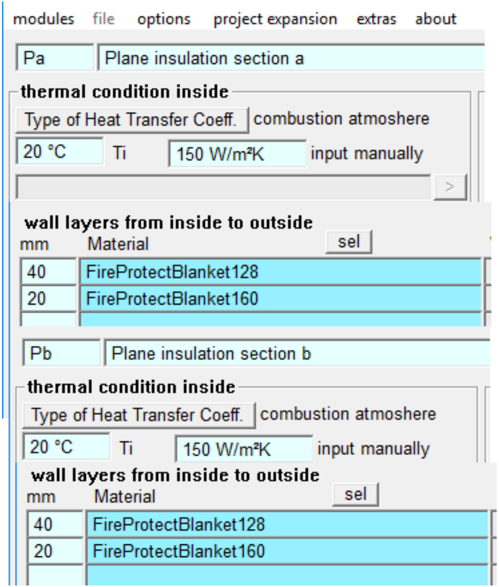


# Enter insulation sections a and b

Enter the two insulation sections in the same way as before. Add descriptions in order not to mix up the sections. For now, use the same layers in both sections.

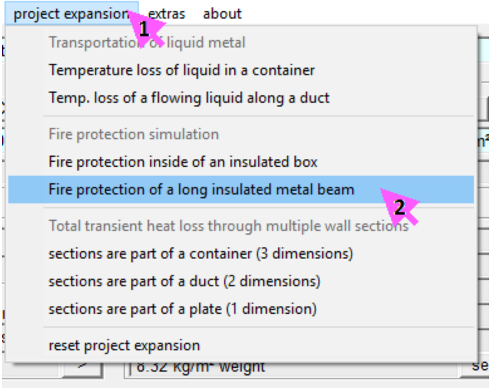
1. enter insulation section a
2. duplicate section a and change the description

Later we might add some insulation wool filled in the cavitations



# Project expansion 'Fire Protection Beam'

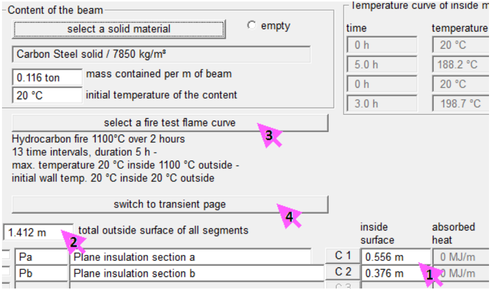
1. In the dropdown menu *project expansion* you ca choose two expansions suitable for fire protection.
2. select *'Fire protection of a long insulated metal beam'*
This is the expansion for two dimensional objects like a long beam or other long thin prismatic bodies.



# The fire protection page

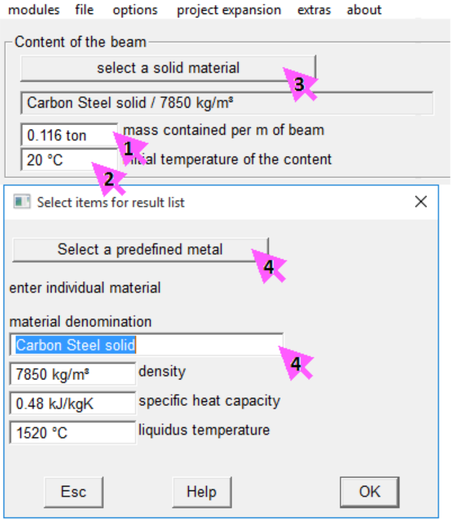
The fire protection page controls working with the simulation. You can enter areas, select and modify a fire curve, define an internal mass, launch simulation and printing, view graphical results.

1. enter the inside areas of the insulation sections
2. enter the total outside area
3. click on *select a fire test flame curve* and select the file **FireCurveHydrocarbon1100Cels.STtime**
4. click on *switch to transient page* to view the transient time table



# Enter the enclosed mass

1. enter the weight **0.116 tons** (=116 kg)
2. enter the initial temperature **20°C**
3. click on *select a solid material* to open the material selection dialog
4. select the predefined material *Carbon Steel solid*



# Simulation of the beam

1. click on *calculate new* to launch the simulation
2. the maximum temperature of 199°C in the beam occurs after 3 hours
3. click on *time curves* for graph of the inside temperature.
4. note that the maximum temperature 199°C is higher than the temperature after 2 hours, when the fire stops.