

Fire protect simulation with Simu-Therm 8.0

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

Contents

- About this tutorial
- Fire protect simulation with Simu-Therm 8.0
- Example 1: insulated box
- Enter the insulation sections in a project
- Project expansion 'Fire Protection Box'
- The fire protection page
- The transient page for fire protection
- Simulate the empty insulation box
- Enter the enclosed mass
- Simulate insulation box with mass
- Find the maximum temperature
- Example 2: insulated long steel beam
- Project expansion 'Fire Protection Beam'
- The fire protection page
- Enter the enclosed mass
- Simulation of the beam

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.

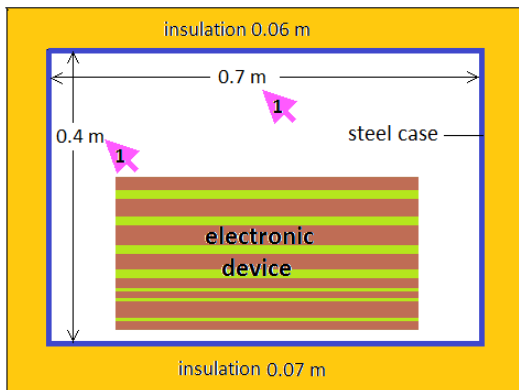
The simulation requires the following steps

- 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.



inside
cap $0.7 \times 0.7 = 0.49 \text{ m}^2$
sides $4 \times 0.7 \times 0.4 = 1.12 \text{ m}^2$
bottom $0.7 \times 0.7 = 0.49 \text{ m}^2$

weight
mass enclosed in the insulation:
steel case + device = 120 kg
material composition:
60 % steel, 30 % copper, 10 % plastics

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

modules file options project expansion extras about

SW | Side wall insulation

thermal condition inside

Type of Heat Transfer Coeff. c combustion atmosphere

20 °C Ti 1000 W/m²K input manually

thermal condition outside

Type of Heat Transfer Coeff. am ambient air

1100 °C Ta 234.8 W/m²K ASTM_04

3.772 m characteristic length

wall layers from inside to outside

mm	Material	sel
20	MICRO-POROS 300 KG	
20	FireProtectBlanket128	
20	FireProtectBlanket160	

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.

The screenshot shows the Simu-Therm 8.0 interface for defining insulation sections. It consists of two main panels, one for 'BOT' (Bottom) and one for 'CAP' (Cap).

Section 1: BOT (Bottom plate insulation)

wall layers from inside to outside

mm	Material
30	CALCIUMSILICAT 1000
20	MICRO-POROS 300 KG
20	FireProtectBlanket160

Section 2: CAP (Cap insulation)

wall layers from inside to outside

mm	Material
20	MICRO-POROS 300 KG
20	FireProtectBlanket128
20	FireProtectBlanket160

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

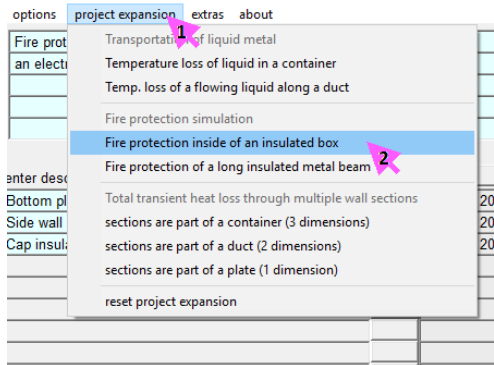
modules file options project expansion extras about					
Offer/Order No.		Fire protection insulation for			
Customer		an electronic device box			
Project					
Location					
Name					
sel	calc-ID	enter description of calculation		press but edit calcu	
<input type="checkbox"/>	BOT	Bottom plate insulation			
<input type="checkbox"/>	SW	Side wall insulation			
<input type="checkbox"/>	CAP	Cap insulation			
<input type="checkbox"/>					

Project expansion 'Fire Protection Box'

In the dropdown menu *project expansion* you can 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 expansion 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 second 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 screenshot shows the 'Fire protection' page in Simu-Therm 8.0. It includes a menu bar (modules, file, options, project expansion, extras, about) and several input fields and tables. Numbered callouts point to specific features: 1 points to the 'absorbed heat' column in the insulation table, 2 points to the 'total outside surface of all segments' input, 3 points to the 'select a fire test flame curve' button, and 4 points to the 'switch to transient page' button.

Content of the box

select a solid material ☐ empty

/ 0 kg/m³

0.000000 ton mass of the box content

0 °C initial temperature of the content

select a fire test flame curve

Hydrocarbon fire 1100°C over 2 hours
11 time intervals, duration 3 h -
max. temperature 0 °C inside 1100 °C outside -
initial wall temp. 17 °C inside 17 °C outside

switch to transient page

3.08 m² total outside surface of all segments

			inside surface	absorbed heat
<input type="checkbox"/>	BOT	Bottom plate insulation	C 1 0.49 m²	1.367
<input type="checkbox"/>	SW	Side wall insulation	C 2 1.12 m²	0.064
<input type="checkbox"/>	CAP	Cap insulation	C 3 0.49 m²	0.028

Temperature curve of inside

time	temperature
0 h	0 °C
3.0 h	673.5 °C
0 h	17 °C
2.328 h	766.6 °C

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.

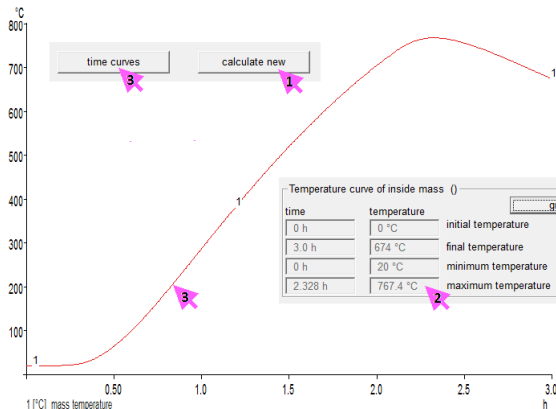
2. in case of fire protection, users can modify only the temperature curve, i.e. the columns time and temperature. To avoid mistakes, everything else is automatically set by Simu-Therm.

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


modules file options project expansion extras about					
description of the transient time schedule					
Hydrocarbon fire 1100°C over 2 hours					
total duration 3.000 h			absorbed heat		
time	seq.	ambient temp.		heat transfer coefficient	
duration		inside	outside	inside	outside
-->	-	0 °C	20 °C	RADIAT V	RADIAT V
1	m	0 °C	743 °C	RADIAT V	RADIAT V
1	m	0 °C	844 °C	RADIAT V	RADIAT V
3	m	0 °C	948 °C	RADIAT V	RADIAT V
5	m	0 °C	1034 °C	RADIAT V	RADIAT V
5	m	0 °C	1071 °C	RADIAT V	RADIAT V
15	m	0 °C	1088 °C	RADIAT V	RADIAT V
30	m	0 °C	1100 °C	RADIAT V	RADIAT V
30	m	0 °C	1100 °C	RADIAT V	RADIAT V
30	m	0 °C	1100 °C	RADIAT V	RADIAT V
-->	-	0 °C	20 °C	RADIAT V	RADIAT V
60	m	0 °C	20 °C	RADIAT V	RADIAT V
+					

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


modules file options project expansion extras about

Content of the box

☐ empty

mass of the box content

initial temperature of the content

 Select items for result list ×

enter individual material

material denomination

density

specific heat capacity

liquidus temperature

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 $0.48 \frac{\text{kJ}}{\text{kgK}}$ for carbon steel and $0.385 \frac{\text{kJ}}{\text{kgK}}$ for copper.

$1.4 \frac{\text{kJ}}{\text{kgK}}$ is a typical value for plastics
For our composed material we can use the weighted average: $0.6 \times 0.48 + 0.3 \times 0.385 + 0.1 \times 1.4 = 0.5435 \frac{\text{kJ}}{\text{kgK}}$

select metal

- Iron (Fe) solid
- Carbon Steel solid**
- Stainless Steel solid
- Cast Iron solid
- Aluminum (Al) solid
- Magnesium (Mg) solid
- Copper (Cu) solid
- Titanium (Ti) solid
- PE polyethylene
- PVC polyvinyl chloride

material denomination

Carbon Steel solid

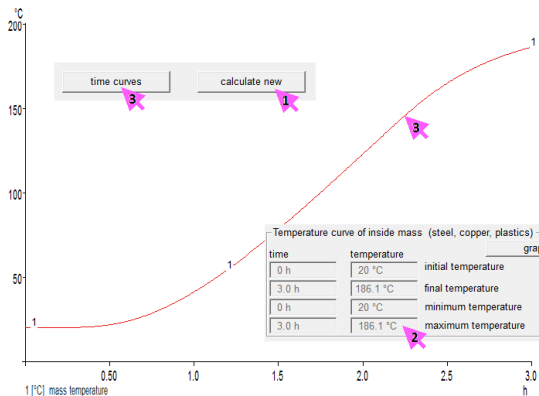
7850 kg/m³ density

0.48 kJ/kgK specific heat capacity

1520 °C liquidus temperature

Simulate insulation box with mass

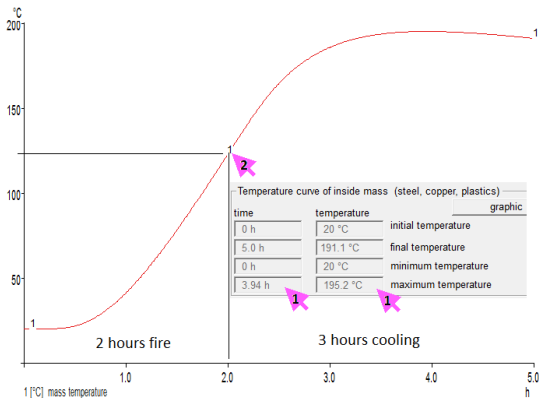
1. click on *calculate new* to launch the simulation again
2. this time maximum temperature in the box is 186°C, much better than for an empty box, but still too hot.
3. click on *time curves* for graph of the inside temperature. To find the maximum, we add 2 more hours of cooling time (1 hour fire,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



Find the maximum temperature

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

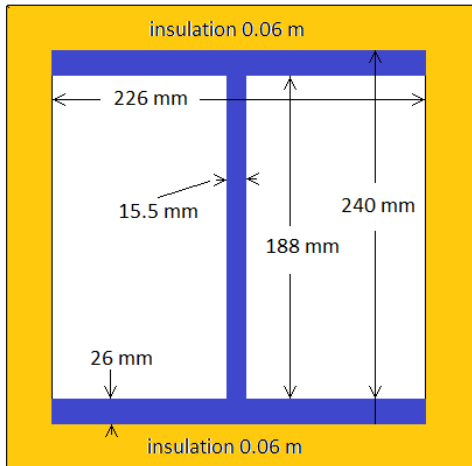
1. the maximum temperature of 195°C occurs almost 2 hours after the fire stopped.
2. be aware that the maximum temperature can be much higher than the temperature at the end of firing. (in this case 195°C vs 125°C)



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.



inside circumference

section a: $2 \times 226 + 4 \times 26 \text{ mm} = 0.556 \text{ m}$

section b: $2 \times 188 \text{ mm} = 0.376 \text{ m}$

weight

beam weight 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

The screenshot displays the Simu-Therm 8.0 software interface. At the top, a menu bar includes 'modules', 'file', 'options', 'project expansion', 'extras', and 'about'. The main window is divided into two sections, 'a' and 'b', each representing a 'Plane insulation section'.

Section a: Plane insulation section a

- thermal condition inside:**
 - Type of Heat Transfer Coeff.: combustion atmosphere
 - 20 °C Ti 150 W/m²K input manually
- wall layers from inside to outside:**

mm	Material
40	FireProtectBlanket128
20	FireProtectBlanket160

Section b: Plane insulation section b

- thermal condition inside:**
 - Type of Heat Transfer Coeff.: combustion atmosphere
 - 20 °C Ti 150 W/m²K input manually
- wall layers from inside to outside:**

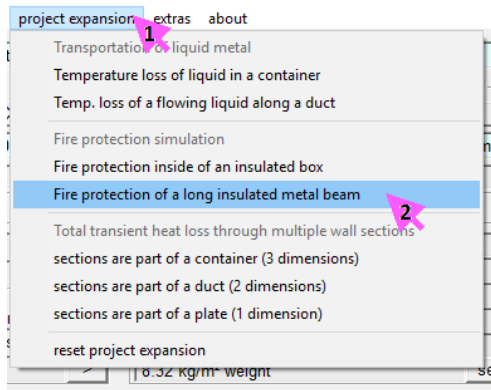
mm	Material
40	FireProtectBlanket128
20	FireProtectBlanket160

Project expansion 'Fire Protection Beam'

1. In the dropdown menu *project expansion* you can 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

The screenshot shows the 'Content of the beam' and 'temperature curve of inside m' sections of a software interface. Numbered callouts point to specific fields:

- 1: Points to the 'absorbed heat' column in the 'temperature curve of inside m' table.
- 2: Points to the 'total outside surface of all segments' field.
- 3: Points to the 'select a fire test flame curve' button.
- 4: Points to the 'switch to transient page' button.

Content of the beam

select a solid material ☐ empty

Carbon Steel solid / 7850 kg/m³

0.116 ton mass contained per m of beam

20 °C initial temperature of the content

select a fire test flame curve

Hydrocarbon fire 1100°C over 2 hours
13 time intervals, duration 5 h -
max. temperature 20 °C inside 1100 °C outside -
initial wall temp. 20 °C inside 20 °C outside

switch to transient page

1.412 m total outside surface of all segments

		inside surface	absorbed heat
Pa	Plane insulation section a	C 1 0.556 m	0 MJ/m
Pb	Plane insulation section b	C 2 0.376 m	0 MJ/m

temperature curve of inside m


time	temperature
0 h	20 °C
5.0 h	188.2 °C
0 h	20 °C
3.0 h	198.7 °C

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*


modules file options project expansion extras about

Content of the beam


select a solid material 

Carbon Steel solid / 7850 kg/m³

0.116 ton  mass contained per m of beam


20 °C  initial temperature of the content

Select items for result list

Select a predefined metal 

enter individual material

material denomination

Carbon Steel solid 

7850 kg/m³ density

0.48 kJ/kgK specific heat capacity

1520 °C liquidus temperature

Esc Help OK

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.

