# SAFIR<sup>®</sup> training session – level 1

Johns Hopkins University, Baltimore

Example: 3D structural model of a beam

## "3D steel beam with thermal insulation"

T. Gernay & J.M. Franssen



## 1. General description

This example deals with a 3D beam of 6 m length. The beam is a W21x44 steel profile protected with 25.4 (1 inch) of sprayed fire resistive material (SFRM). It is exposed to the ASTM E119 fire on its four sides. The beam is simply supported. It is subjected to a uniformly distributed load of 50 kN/m.



Steel for W21x44:

- Yield strength 355 MPa
- Material model from Eurocode 3 part 1-2

Thermal insulation (SFRM)

- Thermal properties constant with temperature

## 2. Section for W21x44 steel beam

### 2.1. Create a project for 2D thermal analysis

#### From the pull down menu select:

Data -> Problem type -> SAFIR2016 -> Safir\_Thermal\_2d



To save the project select (or use icons on the left): *Files->Save or [Ctrl* + *s]* 

Enter a file name, e.g.: W21x44

GiD creates a directory with the name W21x44.gid

GiD creates a number of system files in this directory.

When you start the SAFIR calculation the Safir . IN, .OUT and .TEM files will be created in this directory.

#### 2.2. Create the geometry of the cross-section

The profile is a W21x44 with dimensions given here below.

Depth	525.8	mm
Width	165.1	mm
Web thickness	8.9	mm
Flange thickness	11.4	mm
SFRM thickness	25.4	mm

We will create the steel profile section first. The profile is centered on (0, 0). Select the option to create system lines:

or

*Geometry->Create->Straight Line* 

Enter in the command line (at the bottom of the widows) successively the coordinates of the 2 nodes that define the lines. After typing the coordinates of a node, click *[Enter]* to validate.

Enter name o Enter points	of the project to define line (ESC to leave)
Command:	-0.08255 -0.2629
2	Loom: 0.576x

Press [Enter]

Command:	0.08255 -0.2629
7	Coom: 0.576x

Press [Enter]

This creates a first line based on these two points, which is the lower face of the lower flange. Now adding a point to create the right face of the lower flange:

Command:	0.08255 -0.2515
7	Zoom: 0.576x

Press [Enter]. Then adding a point on the right side of the web:

Command:	0.00445 -0.2515
	Zoom: 1x

Press [Enter].

And so forth with the following points:

Command: 0.00445 0.2515	Command: 0.08255 0.2515	Command: 0.08255 0.2629
Zoom: 1x	Zoom: 1x	Zoom: 1x
Command: -0.08255 0.2629	Command: -0.08255 0.2515	Command: -0.00445 0.2515
Zoom: 1x	Zoom: 1x	Zoom: 1x
Command: -0.00445 -0.2515	Command: -0.08255 -0.2515	Command: -0.08255 -0.2629
Zoom: 1x	Zoom: 1x	Zoom: 1x

When entering this last point, select [Join] when asked by GiD to connect with the existing point at these coordinates. Then press [Esc] to exit the line creation command.

GiD displays the following geometry:

![](_page_4_Figure_6.jpeg)

Note: GiD offers a number of tools to create the geometry efficiently, such as the ones that can be accessed in *Utilities->Copy...* as well as the possibility to directly create objects. The objective here above was to show the process to create a profile line by line. Now, the *Copy* tool will be used to create the SFRM.

First, select *View->Label->All in->Points* to display the points numbers.

Select the option to copy existing model entities: *Utilities->Copy...* 

Select *Entities types: Points* and *Transformation: Translation*. As first point, leave the coordinates: 0.0, 0.0. As second point, enter: 0.0254, 0.0254 (i.e. the thickness of the SFRM) Then, select the top right point of the profile (no. 7). Click on *Finish* or *Esc* to validate. This creates a new point (13) that is at a distance of 25.4 mm of the profile.

The same translation is applied to nodes 3 and 4.

![](_page_5_Figure_4.jpeg)

Then, a new translation is defined with the second point being: 0.0254, -0.0254. This translation is applied to nodes 2, 5 and 6.

сору			,13
Enti	ities type: Points 👻		
First poir	nt		
Num:	х: 0.0		
	y: 0.0		
	z: 0.0		
Second	point 0.0254		
Num:	x: 0.0234		
	y0.02.34		
*	z: 0.0		
Collap Do Create Mainta Multip	o extrude: No 👻 e contacts ain layers ole copies: 1 ish Press 'Finish' to en		
		15	.14

A third translation is defined with the second point being: -0.0254, 0.0254. This translation is applied to nodes 8, 11 and 12.

Finally, translation -0.0254, -0.0254 is applied to nodes 1, 9 and 10.

This procedure has created 12 new points which define the contour of the SFRM. Now, select the option to create system lines:

*Geometry->Create->Straight Line* 

or 🔨

Use ctrl + A to pick an existing point with the mouse. Click successively on the points to create the contour of the SFRM. (For instance: 22 - 16 - 14 - 15 - 17 ...). When it is done, click *[Esc]*.

or 🏹

![](_page_7_Figure_2.jpeg)

There remains to create the surfaces.

Select Geometry->Create->NURBS surface -> By contour

Select the lines that form the contour of the steel profile and click *Esc* to validate.

![](_page_7_Figure_6.jpeg)

Then select the lines that form the contour of the SFRM (including the internal contour, i.e. actually all the line in the model). Click *Esc* to validate.

#### 2.3. Assign the thermal boundary conditions

In GiD, from the pull down menu select: *Data->Conditions* 

A window opens up in GiD. Select the button ("Line"). On the first pull down list, select: *Frontier constraints* 

Different time-temperature curves are predefined. Select ASTME119.

Conditions Selecting points, lines, surfaces, volumes, layers or groups only Frontier Conditions belonging to that level are displayed Temperature curve FISO	Conditions  Frontier constraints  Flux constraints  Vid constraints		× • •	Conditions	FISO FISO0 ASTME119 HYDROCARB
Assign Entities Draw Unassign	Assign	Entities Dra	w ∐nassign	Assign Entities	F1000THPS F20 F100 USER Draw Unassign

Click on the Assign button and assign it to the contour of the SFRM as shown below.

Press [Esc] or click on Finish to confirm.

Select *DRAW->Colors* in the Conditions dialog box to display the frontier constraints. Press *[Esc]* or click on *Finish* to leave this view mode.

![](_page_8_Figure_10.jpeg)

## 2.4. Assign a torsion constraint

The torsion constraint needs to be applied on a node that is on an axis of symmetry of the section. To create such a node, from the pull down menu select: *Geometry -> Edit -> Divide -> Lines -> Num divisions* 

Enter number of divisions: 2. Select the line at the top of the top flange of the profile. Validate with *Esc*. A node 25 is created.

![](_page_9_Figure_5.jpeg)

In GiD, from the pull down menu select: *Data->Conditions* 

Select the button On the pull down list: *Torsion constraints* Tick the box *Constraint* (only in GiD problem types versions prior to 1.4)

Select the node 25 on the vertical axis of symmetry of the steel profile. Validate with *Finish*.

## **2.5.** Assign the materials

From the pull down menu select: *Data->Materials* Select *STEEL* from the dialog box pull down list The *Thermal* tab is active. Then select: *STEELEC3EN* as Material Type A Convection Coeff hot of 25 A Convection Coeff cold of 4 A Relative Emission of 0.7

Materials			×
STEEL		~ 🐼 🚫	🗙 🗉 🥺 🦪
Thermal	Aechanical		
N Convectio Convectior Relati	MaterialType STE on Coeff hot 25 on Coeff cold 4 ve Emission 0.7	ELEC3EN -	
Assign	Draw	<u>U</u> nassign	Exchange
		<u>C</u> lose	

Then select the *Mechanical* tab. Input:

- A Young modulus of 210 000 MPa
- A Poison ratio of 0.3
- A Yield strength of 355 MPa

Materials			×
STEEL	`	- 🧭 🖒 🔰	🔨 💷 💦 🕗
Thermal	Mechanic	al	
E-M	odulus (You	ung) 2.1e11	
	Poisson r	atio 0.3	
Yield str	ength (N/m	1^2) 3.55e8	1
		-	_
Assign	Draw	Unassign	Exchange
		<u>C</u> lose	

Click on *Assign-> Surfaces* and assign it to the W21x44 surface Press *[Esc]* or *Finish* to confirm.

Next, select *INSULATION* from the dialog box pull down list Fill in with the following properties:

Materials					
INSULATION	~	Ø	$\bigotimes$	🗙 🗉	2
MaterialType	INSULATION				
Thermal conductivity(W/mK)	0.12				
Specific heat(J/kgK)	1100				
Specific mass(kg/m3)	240				
Water content(kg/m3)	16.5				
Convection coeff hot surface	25				
Convection coeff cold surface	4				
Relative emissivity	0.8				
<u>A</u> ssign <u>D</u> raw	<u>U</u> nassigr	ו		Excha	nge
	<u>C</u> lose				

Click on *Assign-> Surfaces* and assign it to the SFRM surface. Press *[Esc]* or *Finish* to confirm.

Select *DRAW->all materials* in the Material dialog box to display Materials Press *[Esc]* or *Finish* to leave

Materials ×	
INSULATION V 🧭 😥 📉 🥶	
MaterialType INSULATION	
Thermal conductivity(W/mK) 0.12	
Specific heat(J/kgK) 1100	
Specific mass(kg/m3) 240	
Water content(kg/m3) 16.5	
Convection coeff cold surface 4	
Relative emissivity 0.8	
Finish Press 'Finish' to end selection Exchange	
Qlose	
у	
4	_
x x	
	· · · · · · · · · · · · · · · · · · ·

#### 2.6. Assign the general data

From the pull down menu select: *Data->Problem Data* 

In the Problem Data dialog mask enter: TIMESTEP, UPTIME, TIMEPRINT as needed Do not forget to tick the box *Autorun Torsion Analysis* Also tick the box *Consider reduction of torsional stiffness* and leave the value as 0.1

#### Click on the *Accept* data button

Problem data	×
	<b>k?</b> 🕘
Title 1	Safir_Thermal
Title 2	Mesh_from_G
TETA	0.9
TINITIAL	20.0
SOLVER	PARDISO -
NCORES	1
Type of calculation	MAKE.TEM +
Global center (Yo)	0
Global center (Zo)	0
Center of torsion(Yc)	0
Center of torsion(Zc)	0
NVOID	0
TIMESTEP	30
UPTIME	10800
TIMEPRINT	60
Auto run torsion anaysis a	nd insert result in Tem file
Consider reduction of tors	ional stiffness
reduction coeff	0.1
Accept	Close

### 2.7. Create the mesh

#### Select Mesh->Generate mesh or use [Ctrl + g]

Enter 0.015 as size of elements to be generated (note: a coarser mesh may be required with the demonstration version of SAFIR. If GiD does not run, try coming back to the mesh definition and using 0.018 as the size).

Validate with OK. Click on View mesh to visualize the mesh

Mesh generation ×	
Enter size of elements to be generated	
Get meshing parameters from model	
OK Cancel	

## 2.8. Start the calculation

From the pull down menu select: *Calculate->Calculate window* Click the *Start* button Click the *Output View* button

GiD creates a .IN file in the project directory and starts the calculation. In the output window you can see the calculation progress from SAFIR and the GiD interface program which generates GiD postprocessor files from the .OUT file.

Click on "Ok", save, and open the postprocessor Diamond to visualize the results.

![](_page_13_Picture_6.jpeg)

The .tem file of this model ('w21x44.tem') will need to be copy-pasted in the folder with the structural model.

## 3. Create model for the 3D structure

### **3.1.** Create a new project for structural **3D** analysis

#### From the pull down menu select:

Data->Problem type->SAFIR2016->Safir\_Structural\_3d

![](_page_14_Picture_6.jpeg)

To save the project select (or use icons on the left):

Files->Save  $or \bigtriangleup or [Ctrl + s]$ 

Note: If Caps lock is active on your keyboard, shortcut do not work

Enter a file name, e.g.: 3DBeam

GiD creates a directory with the name *3DBeam.gid* 

GiD creates a number of system files in this directory.

When you start the SAFIR calculation the Safir .*IN* and .*OUT* files will be created in this directory.

#### 3.2. Copy-Paste the section file in the structural analysis directory

GiD has created the directory 3DBeam.gid

The structural input file, which will be created in this directory, will require the information from the section files. Therefore, these sections files need to be located in the same directory.

Copy and paste the files 'W21x44.tem' in the directory 3DBeam.gid

#### 3.3. Create the system geometry (3D beam)

The view is by default in the x-y plane. Here, the plane of the frame will be defined in the x-z plane. To change to the 3d isometric view select from the pull down menu: *View->Rotate->isometric* 

Or if you want to define a point of view by your own use:

 View->Rotate->Trackball
 or [F7]
 or

 Create the system lines:
 Geometry->Create->Straight Line
 or

Enter in the command line (at the bottom of the widows) successively the coordinates of the 2 nodes that define the beam. After typing the coordinates of a node, click *[Enter]* to validate.

Leaving line creation. 0 new lines Enter points to define line (ESC to leave)					
Command: 000					
	7	0.001			

Press [Enter]

Command:	600	
	Zoom: 1x	

Press [Enter]

Then press [Esc] to leave the line creation menu. You should see this in GiD:

![](_page_15_Figure_12.jpeg)

To see nodes and beams numbers select: *View->Label->All* 

## **3.4.** Define constraints for the supports

From the pull down menu select

Data->Constraints

Select Point Constraints

Tick the boxes for X-constraint, Y-constraint, Z-constraint, and ROTX constraint. Assign these constraints to *POINT 1* and press *[Esc]*.

Safir_Structural_3D x64 Project: 3Dbeam (SAFIR2016\Safir_Structural_3d)
Files View Geometry Utilities Data Mesh Calculate Structure About Problem Type Help
Constraints Priet-Constraints Priet-Constraints Priet-Constraints Priet-Constraints Press Finish' to end selection Dose 1 2 2 2 2

Then, tick the boxes for Y-constraint and Z-constraint and assign these constraints to *POINT 2* and press *[Esc]*.

Constraints     ×       • .     .       Point-Constraints     ~       V-Constraint:     .       Y-Constraint:     .       Z-Constraint:     .       ROTX.Constraint:     .       ROTY.Constraint:     .       ROTZ.Constraint:	Constraints	
Assign Entities Draw Unassign		
Close	2	

In the dial box, with *Draw->Colors* you can display the constraints. Press *Finish* or *[Esc]* to leave this view mode.

## **3.5.** Assign the loads

From the pull down menu select *Data->Loads* 

Assign a distributed load of -50 000 N/m in Z direction on the beam.

Use the function FLOAD.

Press Finish or [Esc] to validate.

Loads	×
• 🔨 🟹	
Beam-Load	- 💦 🕘
X Pressure 0.0	
Y Pressure 0.0	
Z Pressure -50000.0	
LOAD FUNCTION FLOAD	•
Assign Entities Draw	Unassign
2	j
<u>C</u> lose	

### 4.6. Create Local Axes

*Local Axes:* The orientation of the cross-section is controlled by defining a local axes X'Y'Z' –system.

![](_page_17_Figure_10.jpeg)

Unlike SAFIR which needs a 4th node to describe the orientation of a cross section on a beam, the GiD-SAFIR interface uses a local X'Y'Z' axes system. When you start the SAFIR calculation the GiD-SAFIR Interface creates the 4th node in the X'Y' plane. If the center of the local axes is not located on the system line of the beam, the direction vector of the Y'-axis is used together with the starting point of the beam to define the 4th node. However the GiD-SAFIR interface will issue a warning message in the Viewoutput window of the calculation run.

#### From the pull down menu select:

Data->Local Axes->Define

Enter the name of the local axis *LAX* Select *3 points XZ* 

Dialog window		×
	e definition mo	de or delete
<u>3</u> Points XZ	Xand Angle	<u>D</u> elete
	<u>C</u> ancel	

Select **Point 1** as the local axis center. Note: Press "CLTR + A" to allow the selection of an existing point with the mouse.

Select *Point 2* as the point in positive x axis.

The third node points to the positive direction along the z axis. There is no such node available in the model. You have to enter coordinates manually.

End joining	. Enter new Points			
Enter point in positive Z axis (the point will be exactly inside the axes)				
Command:	0 -1 0			

## Press Enter

![](_page_18_Picture_11.jpeg)

To draw local axes select: Data->Local Axes->Draw all

## 4.7. Define the global materials

From the pull down menu select *Data->Material* 

There are 2 materials in the model:

- The steel for the profile W21x44: STEELEC3EN
- The insulation for the SFRM: INSULATION

Material ×
Temperatures General Material1 Material2 Number of materials 2 🗸
<u>A</u> ccept <u>C</u> lose
Material       ×         Temperatures       General       Material1       Material2       Material3         Material1       STEELEC3EN       ×         Mat1       Poisson ratio       0.3         Mat1       Nast remperature       1200.         Mat1       Nast remperature       1200.         Mat1       Rate Decrease Yield Strength       0.
Material ×

<u>A</u>ccept

<u>C</u>lose

#### 4.8. Define the properties (i.e. assign temperature files)

From the pull down menu select *Data->Properties* 

The objective is to assign the .tem file named *w21x44* to the model line. In the dial box of *Data->Properties*, change the File-Name: *safir.tem* to the temperature file (.TEM file) of the cross-section, in this case *w21x44.tem*.

Change *Local-Axes* from *-Automatic-* to *LAX* Change the number of materials to 2. Assign the *w21x44.tem* section to the beam.

Properties ×	
<u>_</u> 7 @	
BEAM Cross Section 🗸 😽 🖉	
File-Name w21x44 tem	
Local-Axes LAX 🗸	
Number of materials 2 🔹	
Mat1 Global Nr 1	
Mat2 Global Nr 2	
Mat3 Global Nr 3	
Mat4 Global Nr 4	
Assign Entities Draw Unassign	
<u>C</u> lose	2

You can draw the local axes of the beams to check the model. Select *Draw -> All Conditions -> Include Local Axes* 

### **4.9.** Assign the mass

To define the mass for dynamic calculation, select from the pull down menu:

Data-> Mass

Select *Mass on Beam* and put 100 kg/m as Distributed-Beam-Mass and 2 as Rotational-Inertia. Assign to the beam.

Mass for Dynamic	Calculation			×	
•					
Mass on Beam			~	<b>k?</b> 🥏	
Distributed-Beam-Mass 100.0					
Rotational-I					
Assign Entities Draw Unassign					
<u>C</u> lose					

#### Define the general problem data 4.10.

Select from the pull down menu: Data->Problem Data And fill as shown below

General	×	General	×
N? 4	2		<b>k</b> ? (2
Calculation parameters Output optional results		Calculation parameters	Output optional results
Title 1 Safir_Static_31 Title 2 Mesh_from_G SOLVER PARDISO NCORES 1 Loads DYNAMIC PURE NR Convergence COMEBACK TIMESTEPMIN 0.002 Consider max displacement PRECISION 1.0e-3 NGEOBEAM 1 NG 2 NFIBERBEAM 584 NGEOTRUSS 0 NGEOSHELL 0 NGSHELLTHICK 3 NREBARS 0 TIMESTEP 2 UPTIME 10800 TIMESTEP 2 UPTIME 10800 TIMESTEPMAX 36. TIMEPRINT 20		PRINTTMPRT Print      PRINTVELAC Print v      PRINTREACT Print r      PRINTMN Print inte      PRNEIBEAM Print st      PRNNXSHELL Print      PRNMXSHELL Print      PRNEASHELL Print f      PRNEISHELL PRINT f      PRINT f      PRNEISHELL PRINT	temperatures in the fibers relocity and acceleration eactions rnal forces of beams iffness in beams tresses in shell elements membrane forces in shell element bending moments in shell element membrane stiffness in shell elements rains and stresses in shell element
		•	•
Accept <u>C</u> lose		Accept	Close

#### **Define the mesh** 4.11.

Select Mesh -> Structured -> Lines -> Assign number of cells

![](_page_21_Figure_7.jpeg)

Assign 10 elements. Select the line. Press [Esc] to validate.

Enter valu	ue window	×		
0	Enter number of cells to assign to lines			
	10			
[	Assign Close			

Select Generate Mesh and then View Mesh

#### 4.12. Start the calculation

From the pull down menu select: Calculate->Calculate window Click the Start button

You can follow the progress of the calculation by clicking on *Output view* or by selecting *Calculate->View process info* 

#### 4.13. Check the results

Open the .XML file in Diamond to check the model. Plot the support conditions, applied loads, deflected shape, membrane forces, etc.

![](_page_22_Picture_10.jpeg)

![](_page_22_Picture_11.jpeg)