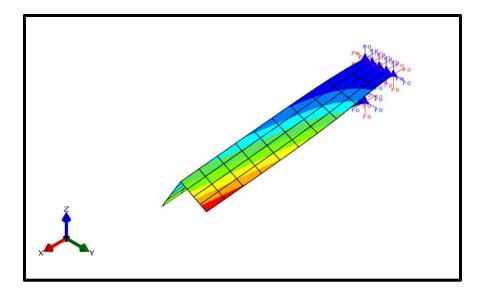
SAFIR[®] training session – level 1

Johns Hopkins University, Baltimore

Example: 3D structural model with shell

"Steel angle in cantilever modeled with shell finite elements"

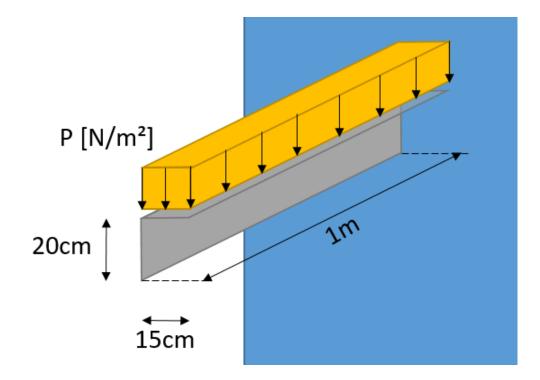
T. Gernay & J.M. Franssen



1. General description

This example deals with a cantilever beam made of an angle profile. The beam is 1 m length and is subjected to a uniformly distributed load. The plates are 6 mm thick. The steel has a yield strength of 275 MPa at ambient temperature. The steel beam is at uniform temperature of 600°C and the objective is to determine the failure load P.

The beam is modeled with shell finite elements.



2. Section for the plates

2.1. Create a project in 2D for TSH thermal analysis

From the pull down menu select: Data -> Problem type -> SAFIR2016 -> Safir_Thermal_tsh

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

Enter a file name, e.g.: *plate* GiD creates a directory with the name *plate.gid* GiD creates a number of system files in this directory. When you start the SAFIR calculation the Safir . IN, .OUT and .TSH files will be created in this directory.

2.2. Create the geometry of the cross-section

GiD will open automatically a new window.

Put 0.006 m as shell thickness and 6 as number of elements, as shown below:

Create Shell-Section ×			
Shell_Thickness(t) in meter 0.006 Number_of_Elements(n) 6			
<u>Apply</u> <u>Close</u>			

Click on Apply

G Safir_The		UNNAMED (SAFIR2016\Safir_Thermal_tsh)			
		esh Calculate About Problem Type Help	ALC 1 0		
000	} @@@@ # @#	🖶 💊 🕸 Layer0 🔍 📚	19 3 1		
ی 🖉					
الا 🖉					
2 🗖					
Ş 🚆					
🍤 :=:			2	1	,
<u> / "</u>					
 					
) P					
20					
Š) 🕣					
- 7					
<u>}</u>			2	1	4
<u> </u>					
0.12					
🔊 😽					
<u>_ 1</u>					
S. 20					
~			3	3	A
1	у				
×	4				
◎ ◎ ◎	×				
*					
Leaving ger	retion				
	transparent syntax without ' prefix:	use 'Zoom instead Zoom			
Command:					
	Zoom: 1x	Nodes: 0, Elements: 0		Render: Normal	Løyers: 1

Note that the section is centered with respect to the y-axis.

It means that in the structural model, the nodeline of the shell elements is located at midheight of the section (if no modification of the nodeline is made by the user).

2.3. Assign the thermal boundary conditions

No thermal boundary conditions will be assigned in this model. The temperatures will be modified manually in the .tsh file later on to impose a uniform temperature of 600°C constant over time.

2.4. 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	- 🧭 🚫 🗙 🖭 <table-cell> 🖉</table-cell>
MaterialTyp	e STEELEC3EN 👻
Convection Coeff h	ot 25
Convection Coeff co	d 4
Relative Emissio	n 0.7
<u>A</u> ssign <u>D</u> ra	v <u>U</u> nassign Exchange
	Close

Click on Assign-> Surfaces and assign it to the 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

2.5. 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. Click on the *Accept* data button

Problem data		×
	k ?	2
General Rel	bar-Layer-Local Rebar-Layer-Global	
	Title 1 Safir_Shell_Te	
	Title 2 Mesh_from_G	
	TETA 0.9	
т	INITIAL 20.0	
5	SOLVER PARDISO -	
N	ICORES 1	
Type of calc	ulation MAKE.TSH 👻	
TIN	MESTEP 12	
l	JPTIME 1000	
TIM	EPRINT 500	
	Accept Close	

2.6. Create the mesh

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

The size of elements is irrelevant since the number of elements has been chosen in 2.2. Just validate with *OK*. Click on *View mesh* to visualize the mesh

2.7. 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.

PlateJOHL Ile Edit Display Plot Results Options Window Dia @ @ @ @ @ @ @ @ @	Contour ⊞Mesh	- c ×
		Diamond 2016 for SAFIR
		FILE : plate NODES : 14 SOLIDS : 6
		MESH PLOT TEMPERATURE PLOT
		TEMPERATURE :
		20°C to 20°C 20°C to 20°C
Žz		20°C to 20°C

2.8. Modify the .tsh file

The .tsh file of this model (*'plate.tsh'*) will need to be copy-pasted in the folder with the structural model. But first it needs to be modified. Go into the folder *plate.gid* and open the file *plate.tsh*.

```
D:\SAFIR\Courses_SAFIR\Formation-20180621\GiD-Examples\SAFIR-training-3-Shell_profile\plate.gid\plate.TSH - Notepad++
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
🔚 frame.in 🗵 🔚 plate.TSH 🗙
       Safir_Shell_Temperature_Analysis
       Mesh_from_GID-Mesher
        THICKNESS 0.006
         MATERIAL 1
REBARS 0
        HOT
POSITIONS OF THE NODES.
 10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
        NUMBER OF FOSITIONS: 7
-0.3000E-02 -0.2000E-02 -0.1000E-02 0.0000E+00 0.1000E-02 0.2000E-02 0.3000E-02
         TIME= 1000.0000 SECONDS OR 16 MIN. 40 SEC.
           -0.0030
-0.0020
-0.0010
0.0000
0.0010
0.0020
                            20.00
20.00
20.00
20.00
20.00
20.00
            0.0030
                             20.00
         TIME= 1000.0000 SECONDS OR 16 MIN. 40 SEC.
           -0.0030
                             20.00
           -0.0030
-0.0020
-0.0010
0.0000
0.0010
0.0020
                             20.00
20.00
20.00
20.00
                             20.00
            0.0030
                             20.00
```

The temperature will uniformly be increased to 600°C in 100 sec, then it will be maintained constant. To do that, modify the file as shown below.

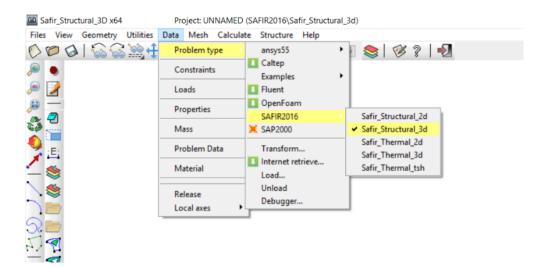
📝 D:\S/	AFIR\Courses_SAFIR\Fc	rmation-20180621\GiD-Examples\SAFIR-training-3-Shell_profile\shell.gid\plate.TSH - Notepad++					
	ile Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?						
		₭ № № ⊅ ⊂ # ₩ ₩ ≪ < 12 23 1					
	.in 🗵 🔚 plate.TSH 🗙						
3	THICKNESS	0.006					
5	MATERIAL	1					
67	REBARS 0						
8	HOT						
9 10	POSITIONS OF						
11	NUMBER OF PO						
12	-0.3000E-02	-0.2000E-02 -0.1000E-02 0.0000E+00 0.1000E-02 0.2000E-02 0.3000E-02					
13 14 15	TIME= 0.00	00 SECONDS OR 16 MIN. 40 SEC.					
16	-0.0030	20.00					
17	-0.0020	20.00					
18 19	-0.0010	20.00 20.00					
20	0.0010	20.00					
21	0.0020	20.00					
22	0.0030	20.00					
24 25	TIME= 100.	0000 SECONDS OR 16 MIN. 40 SEC.					
26	-0.0030	600.00					
27	-0.0020	600.00					
28 29	-0.0010 0.0000	600.00 600.00					
30	0.0010	600.00					
31	0.0020	600.00					
32 33	0.0030	600.00					
34 35	TIME= 1000	0.0000 SECONDS OR 16 MIN. 40 SEC.					
36	-0.0030	600.00					
37	-0.0020	600.00					
38	-0.0010	600.00					
39 40	0.0000	600.00 600.00					
41	0.0020	600.00					
42	0.0030	600.00					
43							

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



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

Files->Save or \bigcirc or [Ctrl + s]

Enter a file name, e.g.: shell

GiD creates a directory with the name *shell.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 shell.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 'plate.tsh' in the directory shell.gid

3.3. Create the system geometry

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

or 🌖

Create the system lines:

Geometry->Create->Straight Line

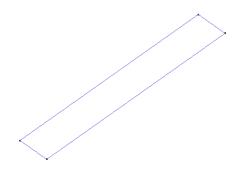
Enter in the command line (at the bottom of the widows) successively the coordinates of the nodes that define the lines. After typing the coordinates of a node, click *[Enter]* to validate.

Leaving line creation. 0 new lines				
Enter points to define line (ESC to leave)				
Command:	000			
	7 0.001			
Press [Ente.	r]			
Enter points ->000	to define line (ESC to leave)			
Command:	0 0.15 0			
	Zoom: 0.601x			
Press [Ente	r]			
Command:	1 0.15 0			
	Zoom: 0.745x			
Press [Ente	r]			
Command:	100			
	Zoom: 0.745x			
Press [Ente	r]			
1				
Command:	000			
	Zoom: 0.745x			
D (E.	7 1 1 4 7 3 41 4			

Press [Enter] and select Join, then press [Esc].

Create point procedure x				
Click the image to get the existing point or create a new one				
No join	Join			
步 . 1				
New point is created	Existing point is selected			
NOTE: Ctrl-a toggles between Join and No join modes.				

You should see this in GiD:



Create the first surface:

Geometry->Create->Surfaceor Select the lines that define the contour of the surface.Then, press the [Esc] key to validate. You should see this in GiD:

Proceed similarly to create the second plate. First create the system lines:

or

Geometry->Create->Straight Line Enter in the command line:

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

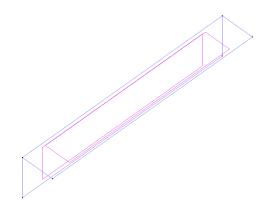
Press [Enter] and select Join

Command:	@0 0 -0.2
	Zoom: 3.29x
Press [Ente	r]
Command:	@100
	Zoom: 1x
Press [Ente	r]
Command:	@0 0 0.2
	Zoom: 3.29x

Press [*Enter*] and select *Join*, then press [*Esc*]. You should see this in GiD:

Then create the second surface:

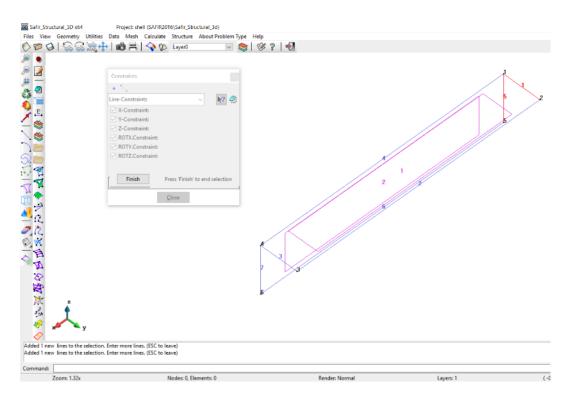
Geometry->Create->Surfaceor $\begin{tabular}{ll}$ Select the lines that define the contour of the second surface.Then, press the [Esc] key to validate. You should see this in GiD:



To see nodes, lines and surfaces numbers select: *View->Label->All*

3.4. Define constraints for the supports

The angle beam is fully fixed at one end. From the pull down menu select Data->Constraints Select Line Constraints Tick all the boxes. Assign these constraints to Line 1 and Line 5 and press [Esc] or Finish to validate.



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 Select Global Shell Load

Specify a distributed load of -100 N/m² in Z direction Use the function User Defined As Filename, write load.fct Assign this load on the Surface 1 Press Finish or [Esc] to validate

Loads	
Global Shell Load V X Pressure 0.0 Y Pressure 0.0	5 2
Z Pressure -100.0 LOAD FUNCTION User Defined Filename3.fct load.fct	2
Assign Entities Draw Unassign	8

3.6. Define the global materials

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

There is only 1 material in the model: STEELEC32D (note that it is a plane stress model)

Material	×		
Temperatures General Material1	2		
Number of materials 1 🔻			
<u>A</u> ccept <u>C</u> lose			
Material	×		
Temperatures General Material1			
Material1 STEELEC32D 👻			
Mat1 E-Modulus 2.1e11			
Mat1 Poisson ratio 0.3			
Mat1 Yield strength 2.75e8			
Mat1 Max Temperature 1200.			
Mat1 Rate Decrease Yield Strength 0.			
<u>A</u> ccept <u>C</u> lose			

3.7. Define the properties (i.e. assign temperature files)

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

The objective is to assign the .tsh file named *plate.tsh* to the model surfaces. In the dial box of *Data->Properties*, select the *SHELL Section Property* Change the File-Name to *plate.tsh*

Keep the number of materials to 1

Assign the *plate.tsh* section to the surfaces.

Properties	5 2
Shell Temp File-Name plate.tsh Number of shell materials 1 ×	
Shell Mat1 Global Nr 1	
Shell Mat2 Global Nr 2	
Shell Mat3 Global Nr 3	
Assign Entities Draw Unassign	7 3
⊆lose	

You can draw the properties to check the model. Select *Draw -> Colors*

3.8. Assign the mass

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

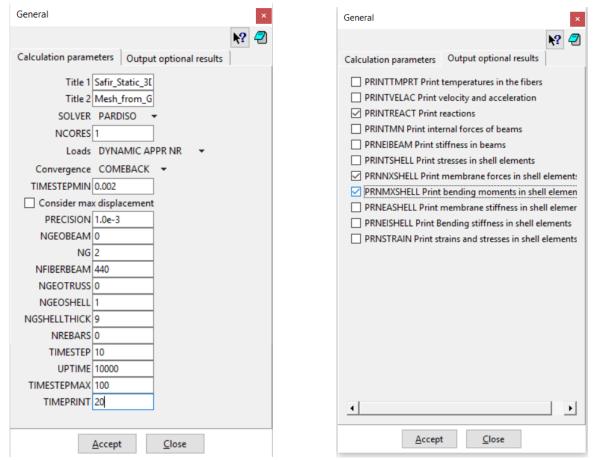
```
Data-> Mass
```

Select *Mass on Shell* and put 100 kg/m² as Distributed-Shell-Mass. Assign the mass to the two surfaces and validate.

Mass for Dynamic Calculation ×				
• 🔨 🔨				
Mass on Shell			~	k? 🕗
Distributed-Shell-Mass 100.0				
Assign	<u>Entities</u>	<u>D</u> raw	<u>U</u> n	assign
Close				

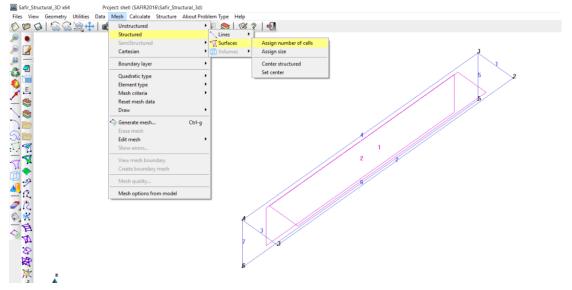
3.9. Define the general problem data

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



3.10. Define the mesh

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



Select the two surfaces, for which a structured mesh will be applied. Press [Esc] to validate.

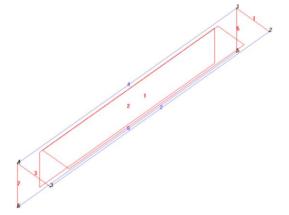
Enter 10 as the number of cells to assign to lines. Select the lines 2, 4 and 6 (1 m long lines). Press *[Esc]* to validate.

Enter value window			
0	Enter number of cells to ass lines	ign to	
	10		
[Assign Close		

Then enter 4 as another number of cells to assign to lines.

Enter value window				
0	Enter another number of cells to assign to lines			
	4			
	Assign	Close		

Select the lines 1, 3, 5 and 7 (short lines). Press [Esc] to validate.



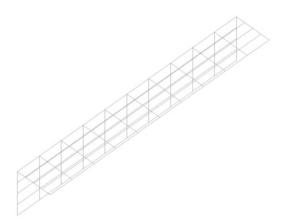
Click on Close.

Next, it must be specified that the lines in the model need not be meshed as beam finite elements.

Select *Mesh -> Mesh criteria -> No mesh -> Lines* Select all the lines in the model. Press [*Esc*] to validate.

Safir_Structural_3D x64 Pro	ject: UNNAMED (SAFIR2016\Safir_Structural_	3d)
<u>Files View G</u> eometry <u>U</u> tilities <u>D</u> ata	Mesh Calculate Structure Help	
(> (=) 🖓 🎧 🖓 🕂 🛍	Unstructured •	🕗 📚 🛛 🥙 🤋 🚽
	Structured •	•
	SemiStructured	
2	Cartesian •	_
	Boundary layer	
3 🕘	Quadratic type	
🔥 💷	Element type	
🖌 <u>正</u>	Mesh criteria	Default 🕨
*_	Reset mesh data	Mesh
N 💑	Draw	No mesh Points
	♦ Generate mesh Ctrl-g	Default mesh
	Erase mesh	Skip Surfaces
	Edit mesh	No skip
A) 🔏	Show errors	Automatic skip
\overline{a}	View mesh boundary	Skip by
	Create boundary mesh	Force points to
	Mesh quality	· · · · · · · · · · · · · · · · · · ·
<u> </u>		Duplicate
	Mesh options from model	No Duplicate
🗢 M		

Select Generate Mesh, OK and then View Mesh



3.11. Create the loading file

In this model a user-defined load function was used, named *load.fct* (see 3.5). This file needs to be created and located in the directory *shell.gid*.

The text file *load.fct* can be edited in a text editor. It is structured as shown below, where the first column is the time (in sec) and the second column is the load (in N). You can define as many time-load pairs as needed. SAFIR will interpolate linearly between the given values.

Here, it is decided to keep the load to zero while the structure is heated (from 0 to 100 sec) then to increase the load linearly as time/10.

Note that here this function will be multiplied by -100 N/m^2 as defined in Section 3.5. As a result, we increase the load by 1 kN/m² every 100 sec in the vertical direction pointing downwards, starting after 100 sec.

/ load.f	fct - Bloc-notes			_	×
Fichier E	dition Format	Affichage	?		
0	0.				,
100.	0.				
1100.	100.				
10100	1000.				

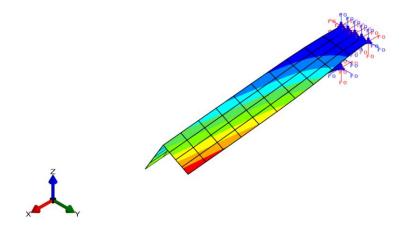
3.12. Start the calculation

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*

3.13. Check the results

Open the .XML file in Diamond to check the model.



Here below is the result with a finer mesh (not suitable for the demonstration version).

