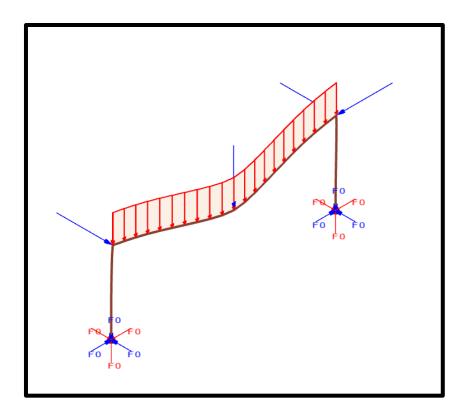
SAFIR[®] training session – level 1

Johns Hopkins University, Baltimore

Example: 3D structural model of a frame

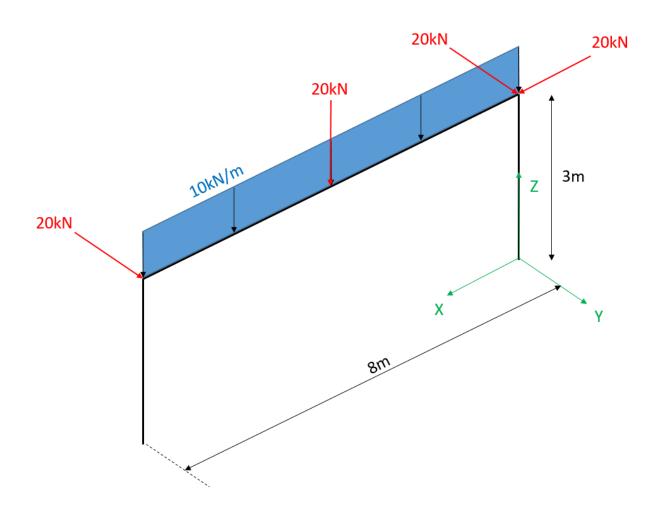
"3D frame with concrete columns and a steel beam"

T. Gernay & J.M. Franssen



1. General description

This example deals with a 3D frame of 3 m height and 8 m length. The columns are reinforced concrete section of 30 cm x 30 cm, with 4 φ 20 steel bars with an axis cover of 30mm, heated on 3 faces. The beam is a IPE 300 steel profile, heated on 3 faces. The columns are fixed and the frame is subjected to a combination of distributed loads and point loads as shown below.



Steel for IPE300:

- Yield strength 355 MPa

Steel for reinforcement bars:

- Yield strength 500 MPa

Concrete:

- Compressive strength 30 MPa
- Tensile strength 1 MPa

2. Section for IPE300 steel beam

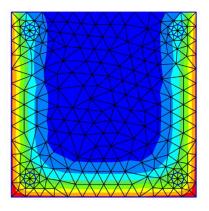
The model created in a previous exercise will be used here.

NNNN WERE SOUGANNAAN	
VIIIII A CONTRACTOR OF CONTRACTO	

The .tem file of this model ('*ipe300.tem*') will need to be copy-pasted in the folder with the structural model.

3. Section for concrete column

The model created in a previous exercise will be used here.



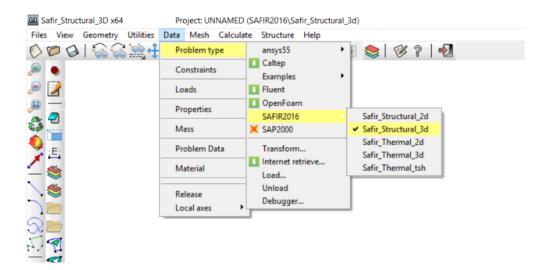
The .tem file of this model ('rc30x30.tem') will need to be copy-pasted in the folder with the structural model.

4. Create model for the 3D structure

4.1. 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 $or \bigtriangleup$ or [Ctrl + s]

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

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

GiD creates a directory with the name 3DFrame.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.

4.2. Copy-Paste the section files in the structural analysis directory

GiD has created the directory 3DFrame.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 'ipe300.tem' and 'rc30x30.tem' in the directory 3DFrame.gid

4.3. Create the system geometry (3D structure)

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

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 4 nodes that define the frame in the command box. 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: 0 0 3

Press [Enter]

Command: 803

Press [Enter]

Command: 800

Press [Enter]

Then press [*Esc*] to leave the line creation menu.

You should see this in GiD:

Safir_Structural_3D x64 Project: 3DFrame	(SAFIR2016\Safir_Structural_3d)			
Files View Geometry Utilities Data Mesh Calco				
		a -		
	🚯 Layer0 🔽 📚 🕅 💞	8 1		
ا کې 😣 🔎				
۵ 🗾				
a —				
) () () () () () () () () () () () () ()				
63			1	
- 9 🔚				
<u>`- \$</u>				
 See 1 				
		/		
6 J 🏹				
Y a				
AI 7				
<u> </u>				
2. C		ſ		
A *				
a B				
8				
10 A				
No		1		
🧐 👖				
· ● () ● () () ● () () () () () (
Leaving line creation. 3 new lines				
Enter points to define line (ESC to leave)				
Commut 1				
Command:	Nodes O. Flowends: O	Desider Manual	Lauran 1	(2 2022 2 20752 0)

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

4.4. Define constraints for the supports

From the pull down menu select

Data->Constraints

Select Point Constraints

Tick all the boxes for a fully fixed condition. Assign these constraints to *POINT 1* and *POINT 4* and press *[Esc]*.

Constraints	2
•	
Point-Constraints v 🥥	
☑ X-Constraint:	2
✓ Y-Constraint:	1
Z-Constraint:	
ROTX.Constraint:	
ROTY.Constraint:	
ROTZ.Constraint:	3
Assign Entities Draw Unassign	3
Close	A

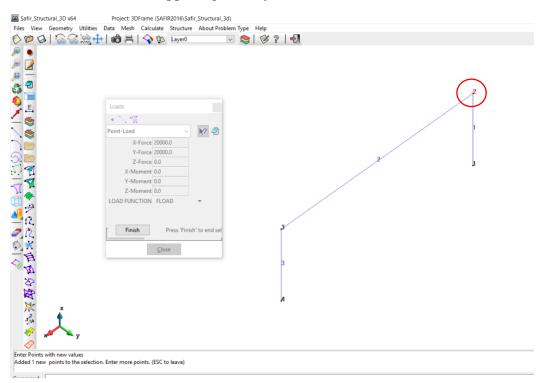
In the dial box, with *Draw->Colors* you can display the constraints. Press *Finish* or *[Esc]* to leave this view mode.

4.5. Assign the loads

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

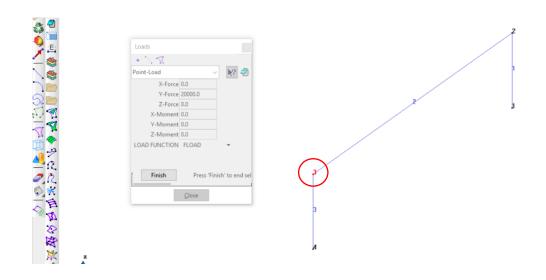
4.5.1. Punctual load at the top of the columns

Assign a punctual load of 20 000 N in X and Y directions on node 2. Use the function *FLOAD* so the load is applied gradually over the first 20 seconds of the simulation.



Press *Finish* or *[Esc]* to validate.

Assign a punctual load of 20 000 N in Y direction on node 3. Press Finish or [Esc].



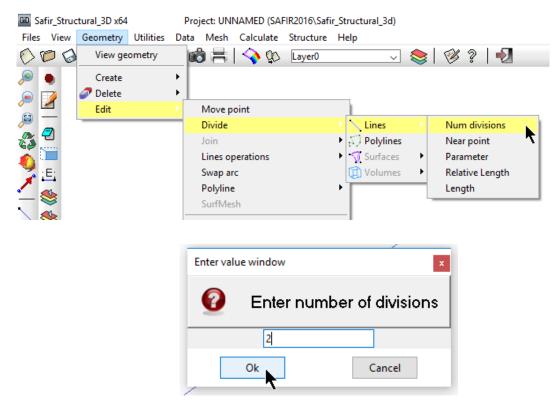
4.5.2. Distributed load on the beam

Assign a distributed load of -10 000 N/m in Z direction on the beam. Press *Finish* or [Esc] to validate.

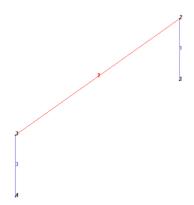
Loads ×	2
X Pressure 0.0 Y Pressure 0.0 Z Pressure -10000.0 LOAD FUNCTION FLOAD	2 1
	3
Assign <u>E</u> ntities <u>D</u> raw <u>U</u> nassign <u>C</u> lose	3 A

4.5.3. Punctual load at mid-span of the beam

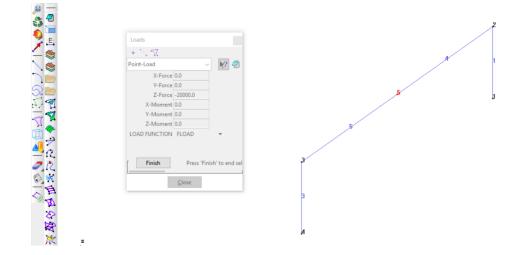
First, create a node at mid-span of the beam. Use *Edit -> Divide -> Lines -> Num divisions: 2*



After clicking on OK, select the line 2 to divide.

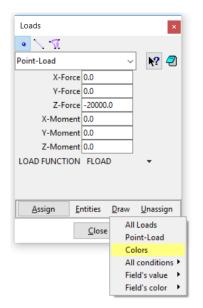


Once the line selected, press [*Esc*] to validate. GiD creates a new node at the center of the line. This node is numbered 5.



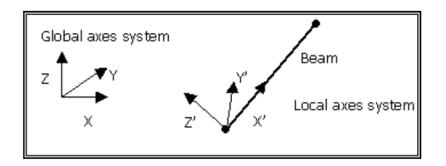
Now back in *Loads*, assign a punctual load of -20 000 N in Z direction on node 5.

Finally, you can check the applied loads by drawing them:



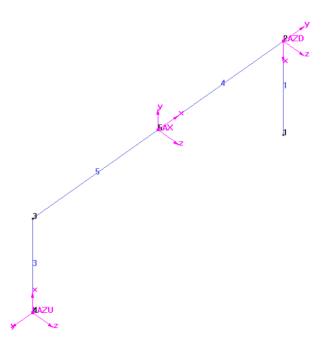
4.6. Create Local Axes

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



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.

The objective is to create 3 local axes (LAX, LAZU, LAZD) as shown in the figure below.



From the pull down menu select: *Data->Local Axes->Define*

Enter the name of the new local axe *LAZD* Select *3 points XZ*

Dialog window		×
🕜 Choo	se definition mo	de or delete
<u>3</u> Points XZ	Xand Angle	<u>D</u> elete

Select **Point 2** as the local axis center. Note: Press "CLTR + A" to allow the selection of an existing point with the mouse.

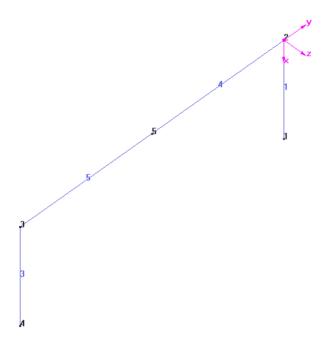
Select *Point 1* as the point in positive x axis.

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.

->013 Entered new	/Local axes definition. Leaving
Command:	0 1 3
	Zoom: 1x

Press Enter

You should see this in GiD. This local axis *LAZU* will be used for the column 1.



Repeat the operations to create the local axe *LAZU*

Select *Point 4* as the local axis center.

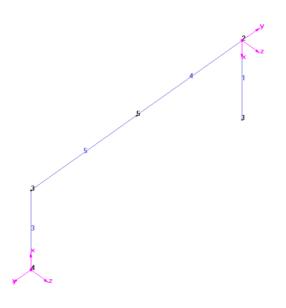
Select *Point 3* as the point in positive x axis.

For the third node (point in positive z axis), use the following coordinates:

End joining. Enter new Points Enter point in positive Z axis (the point will be e	exactly inside the axes)
Command: 810	
Zoom: 1x	Nodes: 0, Eler

Press Enter

You should see this in GiD. This local axis *LAZU* will be used for the column 3.



Finally, repeat the operations to create the local axe LAX

Select *Point 5* as the local axis center.

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

For the third node (point in positive z axis), use the following coordinates:

	nter new Points positive Z axis (the point will be exactly inside the axes)
Command:	4 1 3
	Zoom: 1x

Press Enter

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

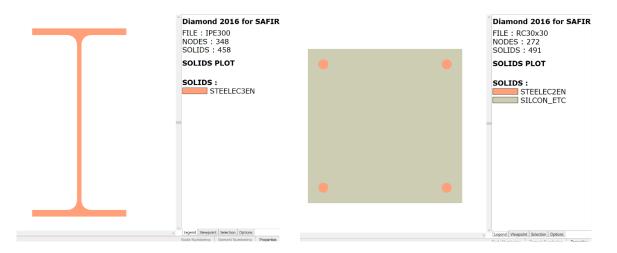
4.7. Define the global materials

From the pull down menu select Data->Material

There are 3 materials in the model:

- The steel for the profile IPE300: STEELEC3EN
- The steel for the reinforcement of the concrete columns: STEELEC2EN
- The concrete for the columns: SILCON_ETC

In the section IPE300: MAT 1 = STEELEC3EN In the section RC30x30: MAT 1 = STEELEC2EN, MAT 2 = SILCON_ETC



In the structural model, we decide that the order is:

MAT 1 = STEELEC3EN MAT 2 = STEELEC2EN MAT 3 = SILCON_ETC

Material	×
Temperatures General Material1 Material2 Material3	2
remperatures ocnerar materiari materiaiz materiais	
Number of materials 3 🔻	
Accept <u>C</u> lose	

Material	×
Temperatures General Material1 Material2 Material3	2
Material1 STEELEC3EN -	
Mat1 E-Modulus 2.1e11 Mat1 Poisson ratio 0.3	
Mat1 Yield strength 3.55e8	
Mat1 Max Temperature 1200.	
Mat1 Rate Decrease Yield Strength 0.	
- <u> </u>	
<u>A</u> ccept <u>C</u> lose	
Material	×
	2
Temperatures General Material1 Material2 Material3	
Material2 STEELEC2EN 👻	
Material STELECZEN	- 1
Mat2 Poisson ratio 0.3	- 1
Mat2 Yield strength 5.0e8	- 1
Mat2 Max Temperature 1200.	- 1
Mat2 Rate Decrease Yield Strength 0.	- 1
Mat2 Process Fabrication HOTROLLED -	- 1
Mat2 Class Ductility CLASS B 🔫	- 1
	- 1
Accept <u>C</u> lose	
2009 2000	_
Material	
Watchar	
	2
Temperatures General Material1 Material2 Material3	
Material3 SILCON ETC 👻	- 1
Mat3 Poisson ratio 0.2	- 1
Mat3 Compressive strength 3.0e7	- 1
Mat3 Tension strength 1.0e6	- 1
	- 1
Accept Close	

Note: a description of the material *SILCON_ETC* is given in the following paper:

Gernay, T., & Franssen, J. M. (2012). A formulation of the Eurocode 2 concrete model at elevated temperature that includes an explicit term for transient creep. *Fire Safety Journal*, *51*, 1-9.

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 ipe300 and rc30x30 to the model lines.

	ି ି ଲିଲ୍ଲି 🕂 🔞 🛱 💊 🖗 Liyer) 🔍 🧕 😻		G
	Properties		
	N-72.00		
	BEAM Cross Section V 🕺		
	File-Name rc30x30.tem		
	Local-Axes LAZD -		
	Number of materials 2 👻		
	Mat1 Global Nr 2		
	Mat2 Global Nr 3		
	Mat3 Global Nr 3		
	Mat4 Global Nr 4		
	Finish Press 'Finish' to end sel		
	Linear Construction of the second		
	Close		
		3	
		L.	
	z		ipe300.tem LAX 1 1 2 3
1			rc30x30.tem LAZD 2 2 3
			rc30x30.tem LAZU 2 2

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 *ipe300.tem*.

Change Local-Axes from -Automatic- to LAX

Keep the number of materials to 1. The Mat1 Global Nr is 1 (it is the STEELEC3EN). Assign the *ipe300.tem* section to the beams 4 and 5 (the horizontal beams of the frame).

Properties		×	
BEAM Cross Section	~	k? 🕗	
File-Nam	e ipe300.tem		×
Local-Axe	es LAX	-	
Number of materia	ls 1 🔻		5
Mat1 Global N	lr 1		
Mat2 Global N	lr 2		5
Mat3 Global N	lr 3		
Mat4 Global N	lr 4		a
Assign <u>E</u> nt	ities <u>D</u> raw	<u>U</u> nassign	3
	<u>C</u> lose		А

Then, change the file name to *rc30x30.tem*. Change *Local-Axes* to *LAZU*. Specify 2 materials. The Mat1 Global Nr is 2 (it is STEELEC2EN). The Mat2 Global Nr is 3 (SILCON_ETC). Assign to column 3.

Properties	2
<u> </u>	
BEAM Cross Section 🗸 🤘 🔁	
File-Name rc30x30.tem	
Local-Axes LAZU -	F
Number of materials 2 🔻	i i
Mat1 Global Nr 2	
Mat2 Global Nr 3	5
Mat3 Global Nr 3	
Mat4 Global Nr 4	3
Assign Entities Draw Unassign	3
Close	А

Finally, keep the file name to *rc30x30.tem* but change *Local-Axes* to *LAZD*. Keep 2 materials with Mat1 Global Nr is 2 and Mat2 Global Nr is 3. Assign to column 1.

Properties	×	
20		7
BEAM Cross Section V	k? 🥘	
File-Name rc30x30.tem		A 1
Local-Axes LAZD	-	
Number of materials 2 💌		5 1
Mat1 Global Nr 2		
Mat2 Global Nr 3		5
Mat3 Global Nr 3		
Mat4 Global Nr 4		2
<u>Assign</u> <u>Entities</u> <u>D</u> raw	<u>U</u> nassign	3
Close		A

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

Image: Constraint of the set Image: Constraint of the set	Real Cross Section Boint-Load Beau Cross Section Boint-Constraints
	File-Name irc30x30.tem Local-Aces LAZD Number of materials Z Matt Global Nr 2 Matt Global Nr 3 Matt Global Nr 4 Finish Press 'Finish' to end set

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 all the beam elements.

Mass for Dynamic	Calculation			×
Mass on Beam			~	k? 🥏
Distributed-Beam-Mass 100.0				
Rotational-Inertia 2.0				
Assign	<u>E</u> ntities	<u>D</u> raw	<u>U</u> r	nassign
	<u>C</u> lo	se		

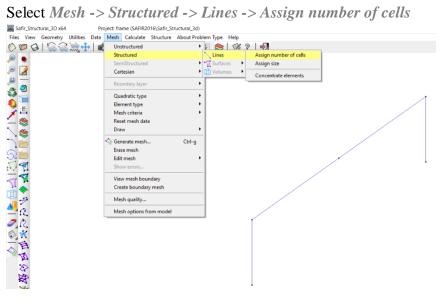
4.10. Define the general problem data

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

General	×	General	
	K? 🕘		∖? 4
Calculation parameters Outp	out optional results	Calculation parameters	Output optional results
Alculation parameters Outp Title 1 Safir_Static_3 Title 2 Mesh_from_ SOLVER PARDISO NCORES 1 Loads DYNAMIC F Convergence COMEBACH TIMESTEPMIN 0.002 Consider max displacemen PRECISION 1.0e-3 NGEOBEAM 2 NG 2 NFIBERBEAM 2000 NGEOTRUSS 0 NGEOSHELL 0		PRINTTMPRT Print PRINTVELAC Print v PRINTREACT Print v PRINTMN Print inte PRINEIBEAM Print st PRINTSHELL Print s PRNNXSHELL Print PRNEASHELL Print PRNEASHELL Print PRNEISHELL Print PRNEISHELL Print PRNEISHELL Print	temperatures in the fibers velocity and acceleration reactions ernal forces of beams
NGSHELLTHICK 3 NREBARS 0			
TIMESTEP 12 UPTIME 3600	-		
TIMESTEPMAX 36.			
TIMEPRINT 12			
		•	
Accept	<u>C</u> lose	Accept	Close

4.11. Define the mesh

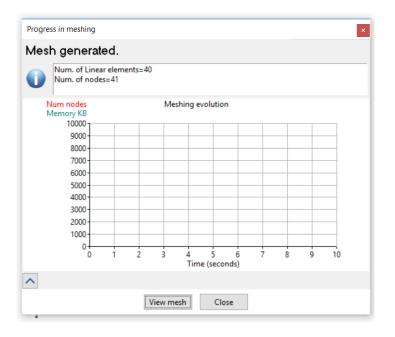
Select Mesh -> Structured -> Lines -> Assign number of cells



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

Enter valu	ie window	×
0	Enter number of lines	cells to assign to
	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 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.

