

**TUD COST ACTION TU0904- INTEGRATED FIRE ENGINEERING AND RESPONSE
(IFER) SHORT TERM SCIENTIFIC MISSION**

VERIFICATION AND NUMERICAL VALIDATION OF BENCHMARK STUDIES

Final Report

STSM Applicant: Carlos Couto

Department of Civil Engineering – University of Aveiro (UA), Portugal



Host:

Dr. Bin Zhao

Centre Technique Industriel de la Construction Métallique (CTICM), France



COST STSM Reference Number: COST-STSM-TU0904-12093
Period: 14th of January to 15th of February of 2013

Contents

- 1 Purpose of the STSM..... 2
- 2 Description of the work carried out during the STSM..... 2
 - 2.1 Summary..... 2
 - 2.2 Development of the software RUBY 2
 - 2.2.1 Introduction 2
 - 2.2.2 Description..... 2
 - 2.2.3 Validation of the software RUBY 6
 - 2.3 Benchmark study of the cross-sectional resistance of an I-shape profile under pure bending at elevated temperature 17
 - 2.3.1 Comparison of the results obtained with SAFIR and ANSYS 20
 - 2.3.2 Improvement of the procedure used in ANSYS to describe the steel material law at elevated temperatures..... 21
- 3 Description of the main results obtained..... 23
- References..... 24

1 Purpose of the STSM

In the framework of WP4 - Benchmark studies, the candidate used numerical software to verify and validate a case study. Numerical software to calculate the critical load factor of tapered steel beam-columns with class 4 cross-section submitted to different types of loading at elevated temperatures has been developed.

2 Description of the work carried out during the STSM

2.1 Summary

The STSM has been held from 14th of January till 15th of February at the Centre Technique Industriel de la Construction Métallique (CTICM), in Saint Aubin, Paris. During this period the work that was carried out can be grouped into two main tasks: a) the development of the software RUBY which is a simple tool that allows the user to perform a linear buckling analysis of a finite element model of SAFIR (Franssen, 2005) and b) the benchmark study of an I-shape profile at elevated temperature, allowing for the verification and validation of a case study with the software SAFIR and ANSYS (ANSYS, 2011).

2.2 Development of the software RUBY

2.2.1 Introduction

In the first part of the STSM the software RUBY (Run a buckling analysis) has been developed. This is a simple tool that allows the user to perform a linear buckling analysis of a SAFIR model made of shell elements, overcoming this limitation of the SAFIR code, thus allowing the calculation of the critical load factors and the respective eigen-modes. RUBY has also the capability of modifying these models in order to apply geometrical imperfections based on the calculated eigen-modes as suggested in the EN1993-1-5 (CEN, 2012). RUBY uses the finite element software Cast3M (Cast3M, 2012) to perform the linear buckling analysis of the model but the advantage of using RUBY is that the user can do this without any knowledge of Cast3M and in a very simple manner.

2.2.2 Description

The software RUBY was developed using the programming language VB.NET and the finite element software Cast3M in order to perform a linear buckling analysis (LBA) of a SAFIR model. RUBY works in a very simple way, the user only needs to select a SAFIR data file, click the "Go" button (see Fig. 1) and wait for the results. RUBY automatically translates the SAFIR model into a Cast3M model and runs Cast3M to perform the LBA behind the scenes, in the end the results of Cast3M are translated again into the results format file of SAFIR meaning that the user can use the same tools for the post-processing of the results as he normally did before.

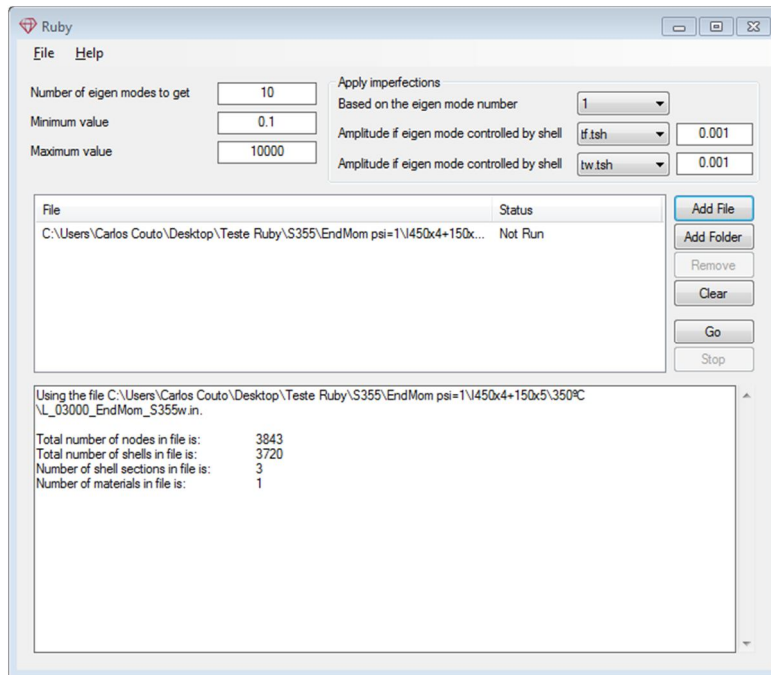


Fig. 1 – Main window of the software RUBY.

With this software is possible to choose the number of eigen modes to get and also to apply geometrical imperfections to the initial SAFIR model based on the shape of a specific eigenmode and with an amplitude defined by the user.

Some advanced options are also included within the software such as: forcing only local or global modes, applying a global imperfection of amplitude = L/x where x is a value defined by the user and L is the member length, forcing only one global and one local mode and applying an imperfection that has a shape based on a combination of both global and local modes.

RUBY can also run on a batch mode meaning that the user can select multiple input files, click “Go” and wait for the results of all the input files without the need of user interaction during the process.

To perform the linear buckling analysis in Cast3M the procedure «FLAMBAGE» is used. As it was mention before, RUBY translates the SAFIR model into a Cast3M model in order to calculate the results, an example of the DGIBI file created by the software is shown next:

```

OPTION ECHO 0 ;

*Carlos Couto (ccouto@ua.pt)
*DATE: 21-01-2013 @ CTICM

*TITRE 'BUCKLING ANALYSIS OF BEAM ' ;
OPTION DIME 3 ELEM QUA4 MODE TRID ;
MESS;
MESS 'SAFIR/Cast3M BUCKLING ANALYSIS v1.0';
MESS '21-01-2013';
MESS;
MESS 'STARTING THE PROBLEM.';
MESS 'BUILDING THE MESH...';

%ENTER_THE_MESH_HERE%

MESS 'MESH OK.';
MESS;

DEP1 = RESO RIGT FT ;
SIG1 = SIGM MODT MATT DEP1 ;

MESS;

```

```

MESS 'CALCULATION OF THE BOUNDARY CONDITIONS OK.';

KSI1 = KSIGMA MODT SIG1 CART 'FLAM' ;
MESS 'CALCULATION OF THE INITIAL STIFFNESS MATRIX OK.';

NMODES = %ENTER_NUMBER_OF_MODES_HERE% ;
ETAB = TABLE ;
ETAB.'CLIM' = BLT;
*ETAB.'RIGI' = RIGT ;
*ETAB.'KSIG' = KSI1 ;
ETAB.'SIG1'=SIG1;
ETAB.'MATE'=MATTO;
ETAB.'CARA'=CART;
ETAB.'LAM1' = %MIN_VALUE% ;
ETAB.'LAM2' = %MAX_VALUE% ;
ETAB.'NMOD' = nmodes ;
ETAB.'OBJM'=MODT;

MESS 'RUNNING THE BUCKLING ANALYSIS...';
MESS;
STAB = FLAMBAGE ETAB ;
MESS;
MESS 'END OF THE BUCKLING ANALYSIS. OK.';

MESS 'WRITING THE BUCKLING MODES TO A FILE...';
BMODE=0;
Defprint = VALE IMPRE;
HEADER = 0;

nmodes = (dime stab) ;
MESS 'FOUND ' nmodes ' mode(s)';
SI (nmodes > 0);

FICH='%ENTER_THE_OUTPUT_FILE_HERE%';
OPTI IMPR 10 IMPR FICH;

REPETER BDRaw (dime stab);
BMODE=BMODE + 1;

LA1 = STAB . BMODE . LAMB ;

MM1 = STAB . BMODE . DEPL ;
DE = DEFO MM1 SHELL 1. ROUGE ;

MM2 = CHAN POI1 SHELL;
i = 0;
SI (HEADER EGA 0);
MESS 'NODENUM X      Y      Z';
HEADER = 1;
REPE Bouc1 (NBNO MM2) ;
  i = i + 1 ;
  PO1 = MM2 POIN i ;
  X Y Z = COOR PO1 ;
  *MESS 'NODE ' i ;
  MESS i X Y Z;
  ValX = EXTR MM1 UX PO1 ;
  ValY = EXTR MM1 UY PO1 ;
  ValZ = EXTR MM1 UZ PO1 ;
  RotX = EXTR MM1 RX PO1 ;
  RotY = EXTR MM1 RX PO1 ;
  RotZ = EXTR MM1 RX PO1 ;
  * MESS valx '      ' valy '      ' valz '      ' rotx '      ' roty '      ' rotz ;
FIN Bouc1 ;
FINSi;
i = 0;

MESS '#MODE ' BMODE ' = ' LA1;

REPE Bouc1 (NBNO MM2) ;
  i = i + 1 ;
  PO1 = MM2 POIN i ;
  X Y Z = COOR PO1 ;
  * MESS 'NODE ' i ;
  * MESS X Y Z;
  ValX = EXTR MM1 UX PO1 ;
  ValY = EXTR MM1 UY PO1 ;
  ValZ = EXTR MM1 UZ PO1 ;
  RotX = EXTR MM1 RX PO1 ;

```

```

RotY = EXTR MM1 RY PO1 ;
RotZ = EXTR MM1 RZ PO1 ;
MESS i'' valx ' ' valy ' ' valz ' ' rotx ' ' roty ' ' rotz ;
* MESS valx ' ' valy ' ' valz
FIN Bouc1 ;

```

```

FIN BDRaw;
FINSI;
OPTI IMPR Defprint;
MESS ;
MESS 'DONE.' ;
FIN;

```

The following instructions are used by RUBY to translate the SAFIR model and other user defined options to the Cast3M model:

- %ENTER_THE_MESH_HERE%
- %ENTER_NUMBER_OF_MODES_HERE%
- %MAX_VALUE% / %MIN_VALUE%
- %ENTER_THE_OUTPUT_FILE_HERE%

The idea of RUBY was to make it as simple as possible for a SAFIR user to perform a linear buckling analysis of a model and to be able to post-process the results in a way that he was familiar with. On the basis of this assumption, it was clear that the results produced by RUBY should be in the same format as the results produced by SAFIR, this was also accomplished.

So, the post-processing of the results of RUBY can be done in the same manner as if the user is dealing with a SAFIR analysis. In Fig. 2, the results of a linear buckling analysis with RUBY are shown, each time step available will correspond to an eigen-mode and its value to the “critical load factor multiplier” for that mode.

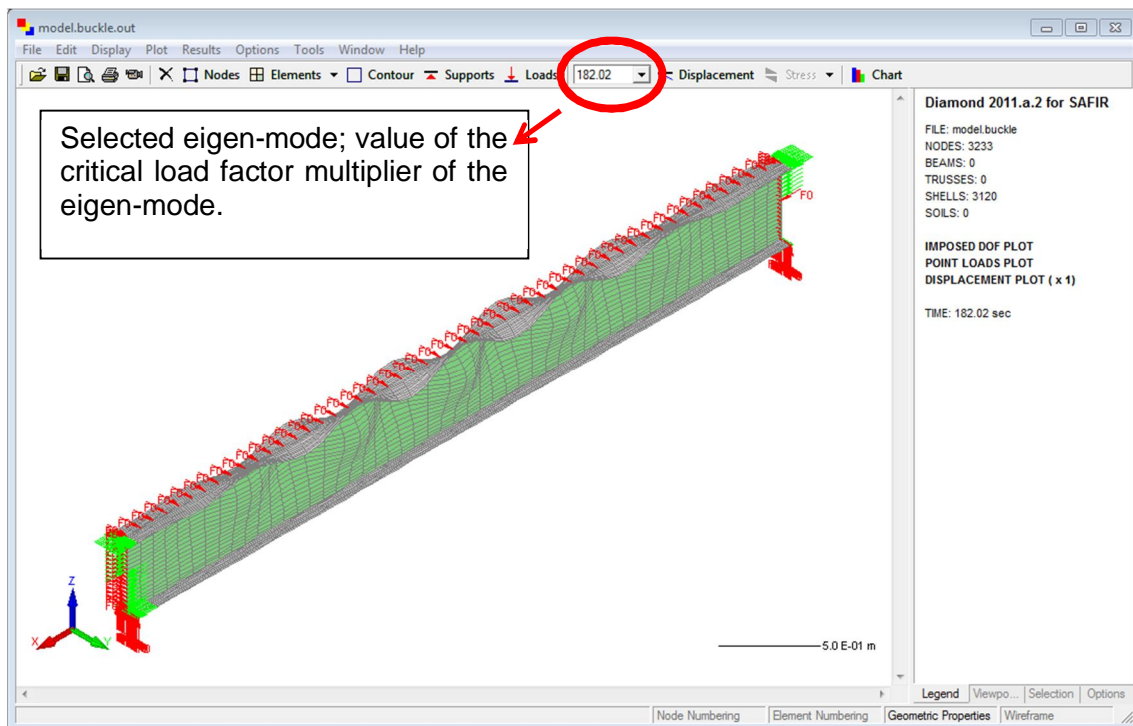


Fig. 2 – Post-processing of the results produced by RUBY, here showing the eigen-shape (deformed shape) for an eigen-mode with a critical load factor multiplier of $\alpha_{cr} = 182.02$.

The actual version of the software is capable of translating a SAFIR model to Cast3M if the following conditions are met:

- The model is in 3D.
- Model is made of Shell elements.

- The loading is composed of “Point loads” (forces/moments). No imposed displacements.
- Load functions will be ignored (in the Cast3M model all the point loads will be considered to have a load function of type F1 – see SAFIR documentation).
- Shell section properties files (*.TSH) are read in the translating process to determine the thickness. All the remaining information in these files will be ignored.
- Cast3M version 12.0 “education” must be installed; this version is free for research purposes. RUBY may not run properly with other versions of Cast3M.

The development of the software RUBY has been supervised by Prof. Paulo Vila Real and co-supervised by Prof. Nuno Lopes from University of Aveiro and Dr. Bin Zhao from Centre Technique Industriel de la Construction Métallique (CTICM). Besides the valuable co-supervision of Dr Bin Zhao, the cooperation with CTICM in this software was also in terms of the explanation of CAST3M language by Ms. Gisèle Bihina, the comparison of simple cases between RUBY software and ANSYS that allowed for the validation of RUBY with the help of Mr. Arnaud Sanzel and also the contribution of Mr. André Beyer regarding the use and some technical aspects of LTBeam that made possible to compare RUBY and LTBeam. In Fig. 3 the about screen of the software RUBY is depicted.

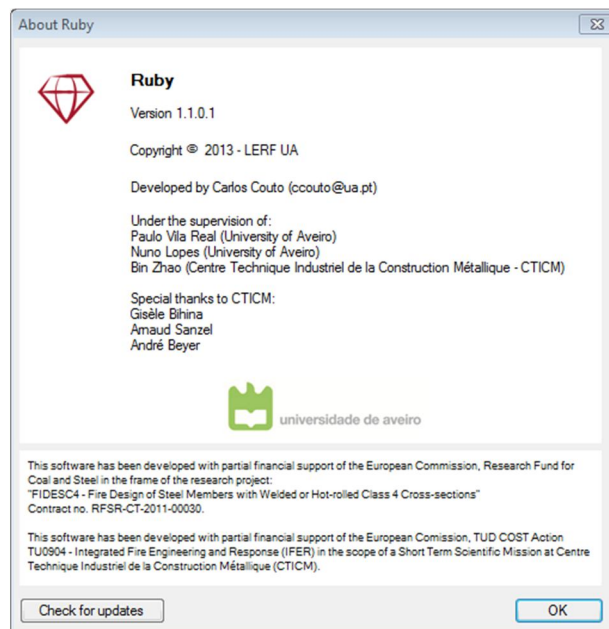
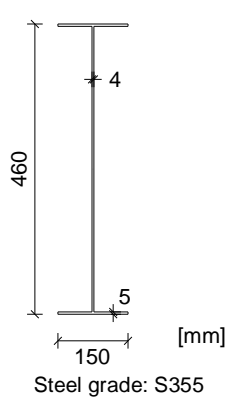


Fig. 3 – About software RUBY.

2.2.3 Validation of the software RUBY

2.2.3.1 Axial compressed member

A column with a total height of 6.0 m has been analysed under axial load only. The column is made of an I-shape welded profile indicated in Fig. 4. This cross-section is classified as Class 4 according to EN 1993-1-1 (CEN, 2005a), meaning that local buckling may occur before the cross-section attains full cross-section resistance. The column is fully fixed in the base and free on the top and the applied axial force is $N_{ed} = 0.43$ kN. The mesh used was 100 divisions along the length, 10 divisions of the flange and 22 divisions of the web.



Normal temperatures classification			
	Compression	Bending about y-y	Bending about z-z
flange	4	4	4
web	4	4	n.a.
class	4	4	4

Elevated temperatures classification			
	Compression	Bending about y-y	Bending about z-z
flange	4	4	4
web	4	4	n.a.
class	4	4	4

Fig. 4 – Studied class 4 cross-section details and classification.

In the next figure, the local mode obtained with RUBY is depicted.

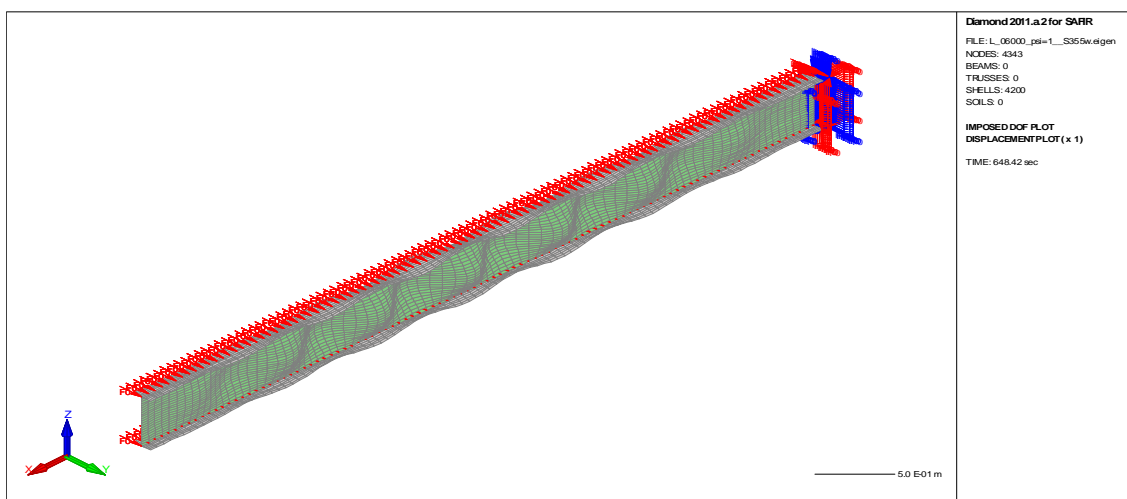


Fig. 5 – Local failure mode of the column calculated with RUBY ($\alpha_{cr} = 648.42$).

In Fig. 6 the local mode obtained with ANSYS is shown.

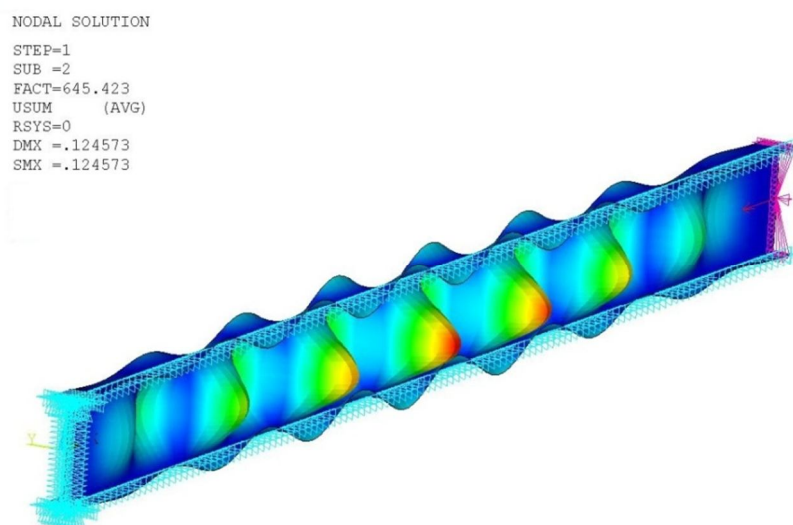


Fig. 6 – Local failure mode of the column calculated with ANSYS ($\alpha_{cr} = 645.42$).

In the next figure the global mode obtained with RUBY is depicted for the strong axis.

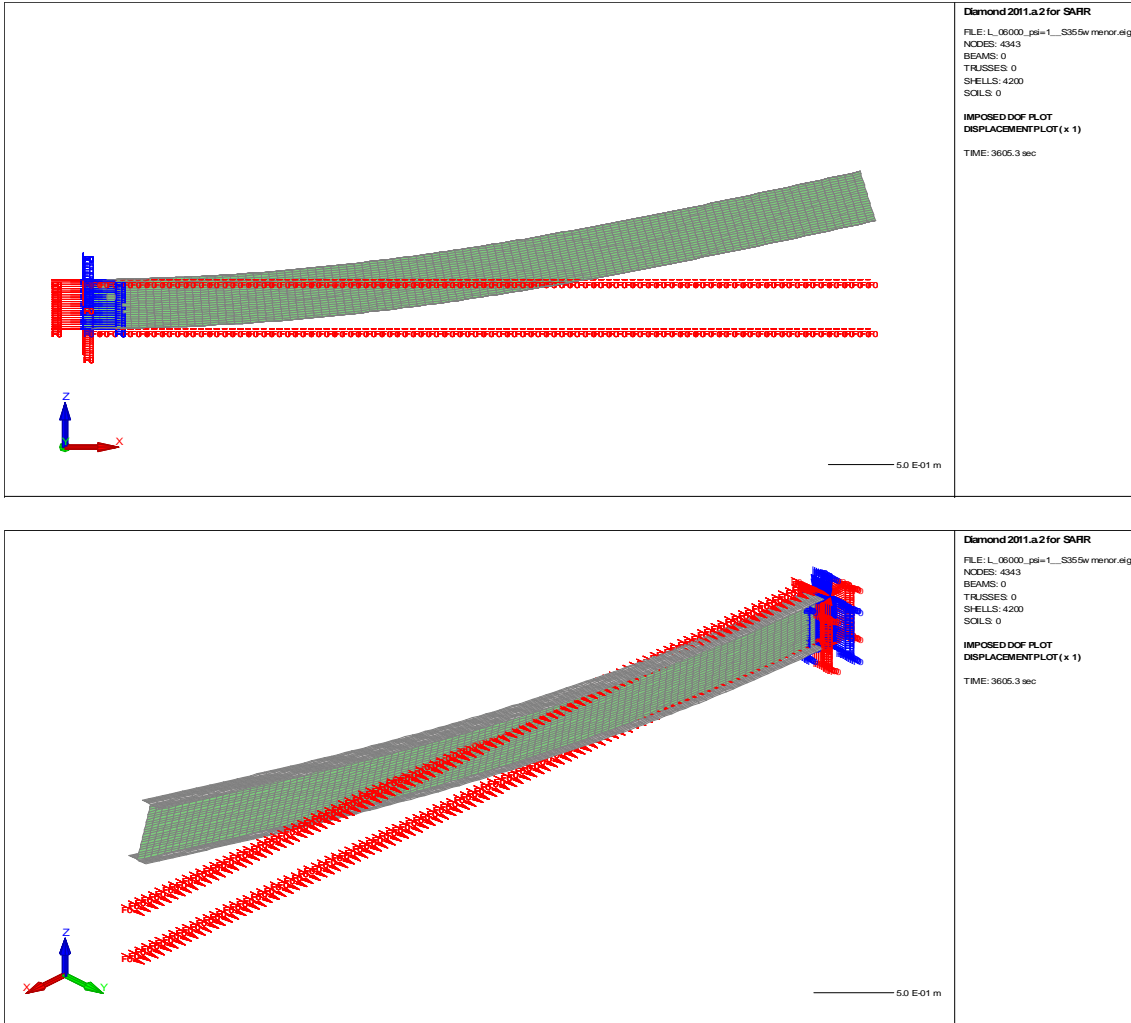
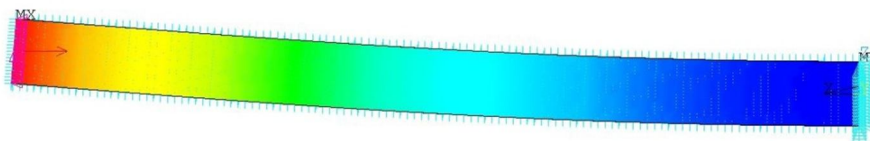


Fig. 7 – Global failure mode of the column about the strong-axis calculated with RUBY ($\alpha_{cr} = 3605.3$).

In Fig. 8 the same case is shown but in ANSYS.

```

NODAL SOLUTION
STEP=1
SUB =1
FACT=3605.67
USUM (AVG)
RSYS=0
DMX =1.00173
SMX =1.00173
  
```



```

NODAL SOLUTION
STEP=1
SUB =1
FACT=3605.67
USUM (AVG)
RSYS=0
DMX =1.00173
SMX =1.00173

```

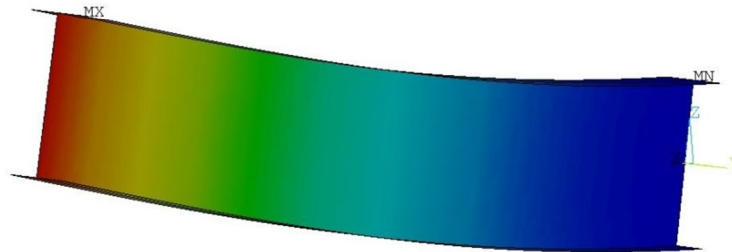


Fig. 8 – Global failure mode of the column about the strong-axis calculated with ANSYS ($\alpha_{cr} = 3605.67$).

In the next figure the global mode obtained with RUBY is depicted for the weak axis.

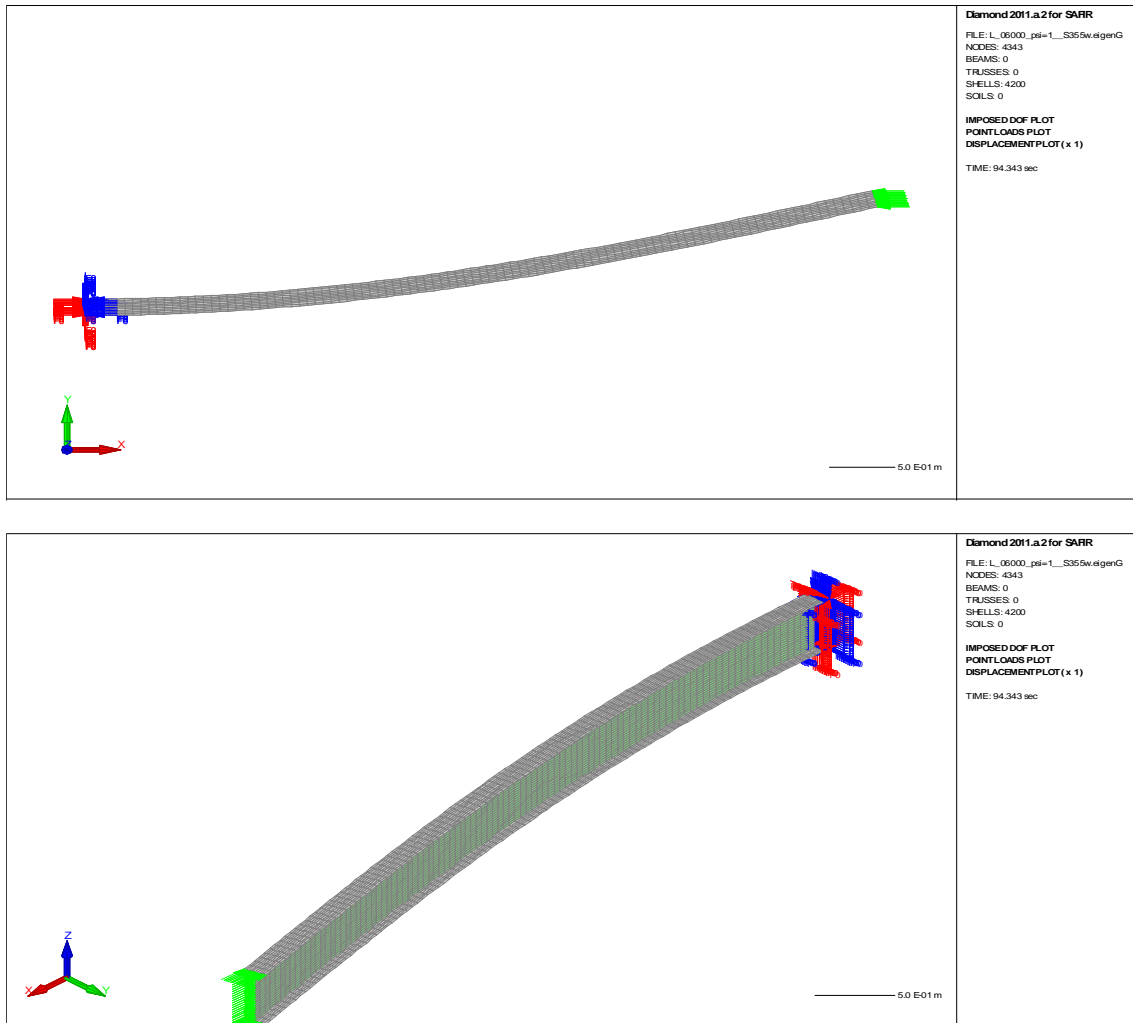


Fig. 9 – Global failure mode of the column about the weak-axis calculated with RUBY ($\alpha_{cr} = 94.343$).

The corresponding results obtained with ANSYS are shown next:

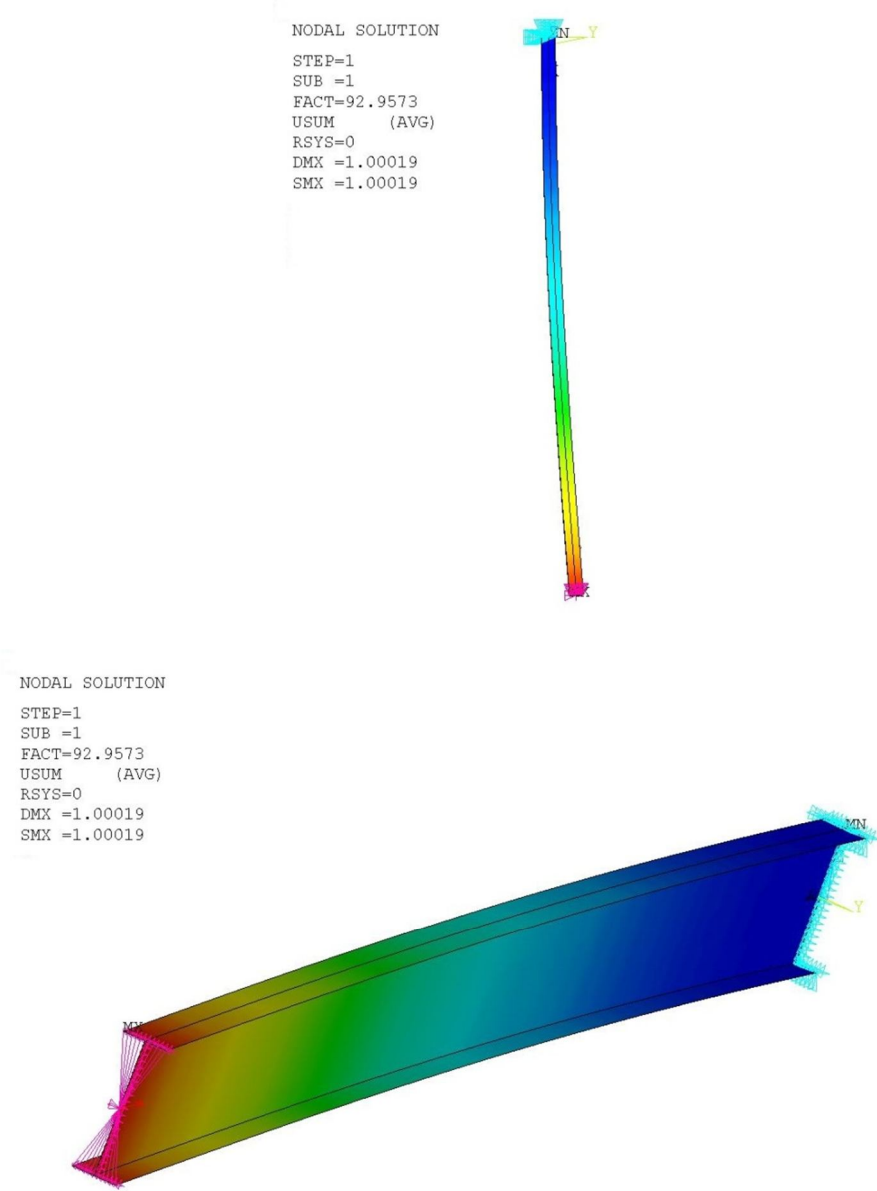


Fig. 10 – Global failure mode of the column about the weak-axis calculated with ANSYS ($\alpha_{cr} = 92.9573$).

In the Table 1 the results obtained are summarized and compared with the analytical solution provided by the Euler critical load:

$$N_{cr} = \frac{\pi^2 EI}{(kL)^2} \tag{1}$$

where k is buckling length factor. For a column fixed in the base and free in the other extremity this takes the value of $k = 2.0$. The value of RUBY (and ANSYS) is calculated as:

$$N_{cr} = \alpha_{cr} N_{Ed} \tag{2}$$

Table 1 – Results of the comparison of an axial compressed member in terms of critical load with the Euler formulation.

Mode	N_{cr} RUBY - FEM [kN]	N_{cr} Euler [kN]	Relative Diff (%)
Local	278.82	n.a.	n.a.
Global (strong-axis)	1550.28	1554.64	-0.280
Global (weak-axis)	40.57	40.51	0.148

Table 2 – Results of the comparison of an axial compressed member in terms of critical load between RUBY and ANSYS software.

Mode	N_{cr} RUBY - FEM [kN]	N_{cr} ANSYS - FEM [kN]	Relative Diff (%)
Local	278.82	277.53	0.465
Global (strong-axis)	1550.28	1550.44	-0.010
Global (weak-axis)	40.57	39.96	1.501

The results shown in the previous table show a very good agreement between the results of RUBY and Euler theory and also with ANSYS. From Fig. 5 to Fig. 10 it can also be seen a very good agreement between RUBY and ANSYS in terms of the eigen-shape.

2.2.3.2 Uniform member submitted to uniform bending moment

A beam with a 6.0 m span submitted to uniform bending moment has been analysed. The cross-section of the beam is the same of the previous section (see §2.2.3.1). The beam is considered to fork-support conditions and it is submitted to a uniform bending moment of $M_{ed} = 0.101$ kN.m. The mesh used was 100 divisions along the length, 10 divisions of the flange and 22 divisions of the web.

In the next figure, the buckling mode obtained with RUBY can be seen. This mode is clearly a lateral-torsional buckling mode.

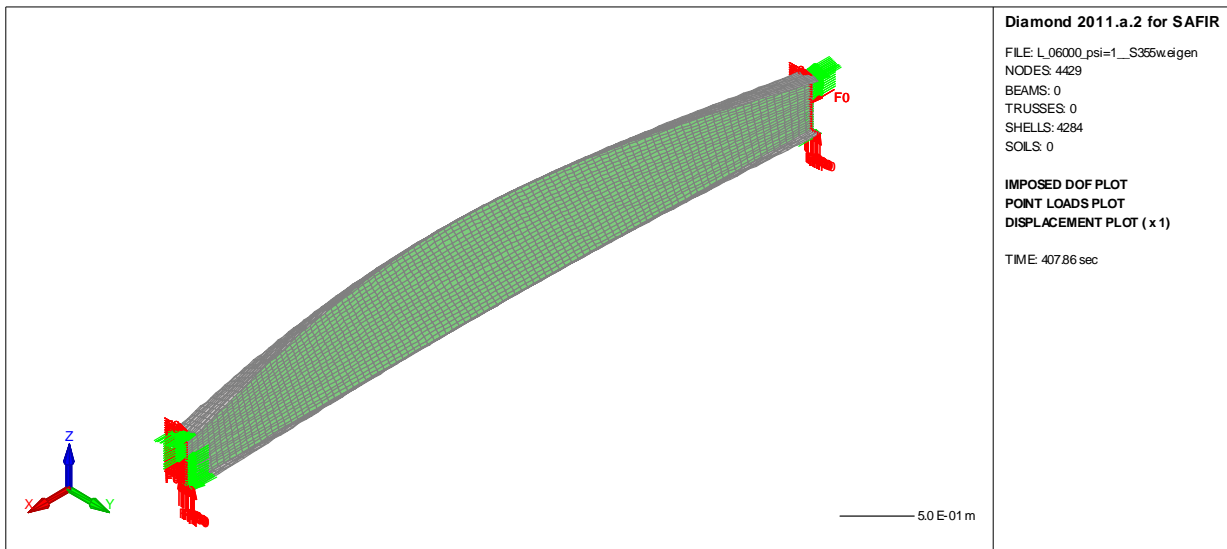


Fig. 11 – Global failure mode of the beam calculated with RUBY ($\alpha_{cr} = 407.86$).

In Fig. 12 the same eigen-shape is obtained in ANSYS. The critical load factor multiplier is different because the applied bending moment intensity is also different. In the ANSYS model the applied bending moment is $M_{ed} = 0.0544$ kN.m.

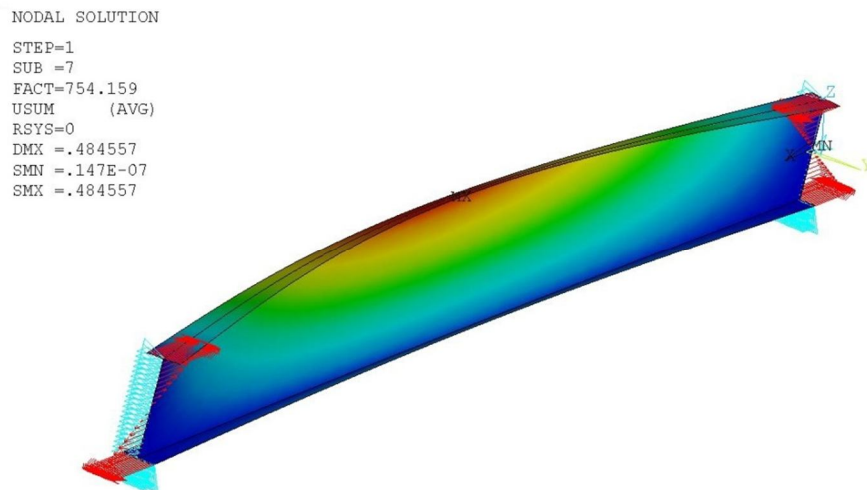


Fig. 12 – Global failure mode of the beam calculated with ANSYS ($\alpha_{cr} = 754.159$).

In the Table 3 and Table 4 the results obtained are summarized and compared with the software LTBeam (CTICM, 2012) and with ANSYS. The value of RUBY (and also ANSYS) is calculated as:

$$M_{cr} = \alpha_{cr} M_{Ed} \quad (3)$$

Table 3 – Results of the comparison of a beam submitted to uniform bending moment in terms of critical load with LTBeam.

Mode	M_{cr} RUBY - FEM [kN.m]	M_{cr} LTBeam [kN.m]	Relative Diff (%)
Lateral-torsional	41.19	40.58	1.503

Table 4 – Results of the comparison of a beam submitted to uniform bending moment in terms of critical load with ANSYS.

Mode	M_{cr} RUBY - FEM [kN.m]	M_{cr} ANSYS - FEM [kN.m]	Relative Diff (%)
Lateral-torsional	41.19	41.03	0.390

The results from the previous table show very good agreement between RUBY, ANSYS and LTBeam.

In the next figure the local buckling mode of the beam is also depicted for RUBY.

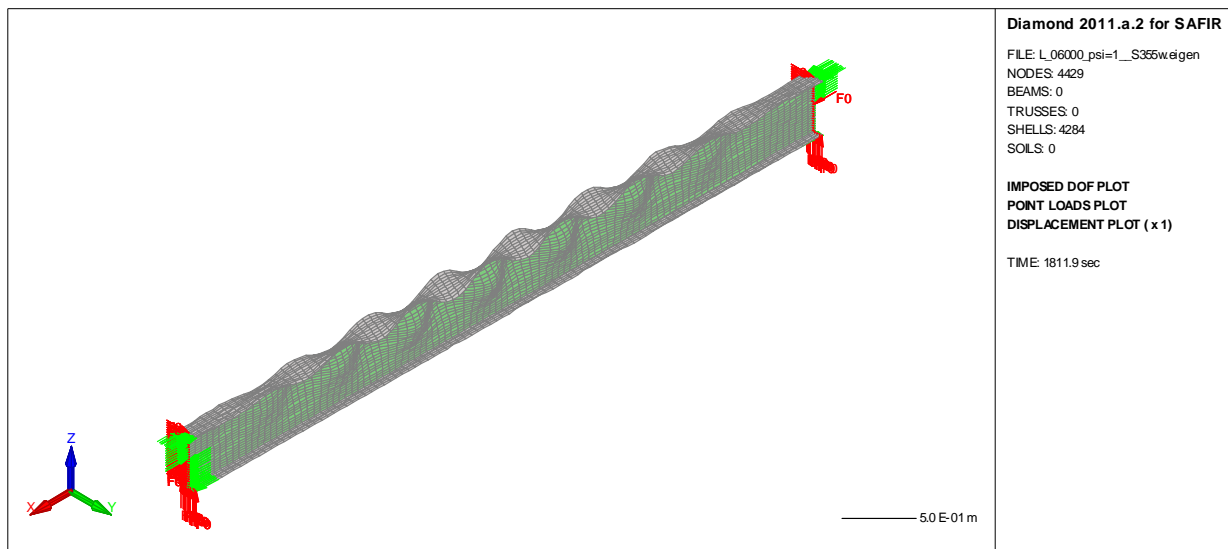


Fig. 13 – Local failure mode of the beam calculated with RUBY ($\alpha_{cr} = 1811.9$).

And the corresponding results obtained with ANSYS:

```

NODAL SOLUTION
STEP=1
SUB =7
FACT=3330.47
USUM      (AVG)
RSYS=0
DMX  =.08738
SMN  =.603E-09
SMX  =.08738

```

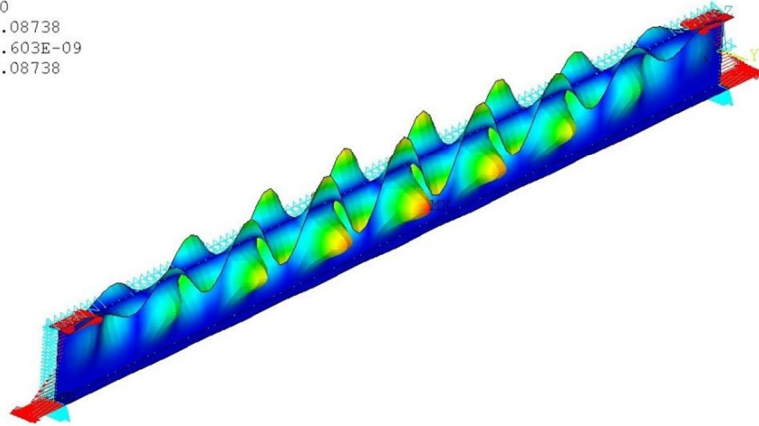


Fig. 14 – Local failure mode of the beam calculated with ANSYS ($\alpha_{cr} = 3330.47$).

In the Table 5 the results obtained are summarized and compared with ANSYS. The values were calculated using Eq. (3).

Table 5 – Results of the comparison of a beam submitted to uniform bending moment in terms of critical load with ANSYS.

Mode	M_{cr} RUBY - FEM [kN.m]	M_{cr} ANSYS - FEM [kN.m]	Relative Diff (%)
Local buckling	183.00	181.18	1.004

Again, it can be seen from the previous figures and the table that there is very good agreement between the results of RUBY and ANSYS.

2.2.3.3 Tapered member submitted to uniform bending moment

A tapered beam with a 6.0 m span submitted to uniform bending moment has been analysed. The cross-section of the beam is the same of the previous section (see §2.2.3.1) with a tapered ratio of twice the height of the cross-section as shown in Fig. 15. The beam is considered to have fork-support conditions and it is submitted to a uniform bending moment of $M_{ed} = 0.645273$ kN.m. The mesh used was 100 divisions along the length, 10 divisions of the flange and 22 divisions of the web.

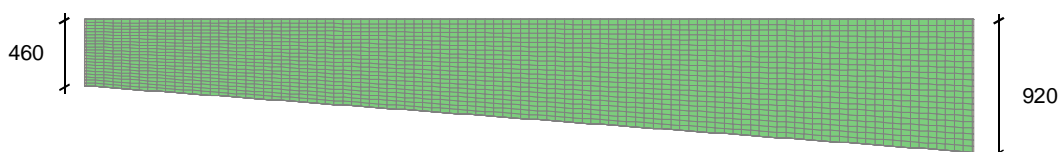


Fig. 15 – The beam has a tapered ratio of twice the height of the cross-section.

In Fig. 16 the global failure mode of the beam can be observed. It is clearly a lateral-torsional buckling mode.

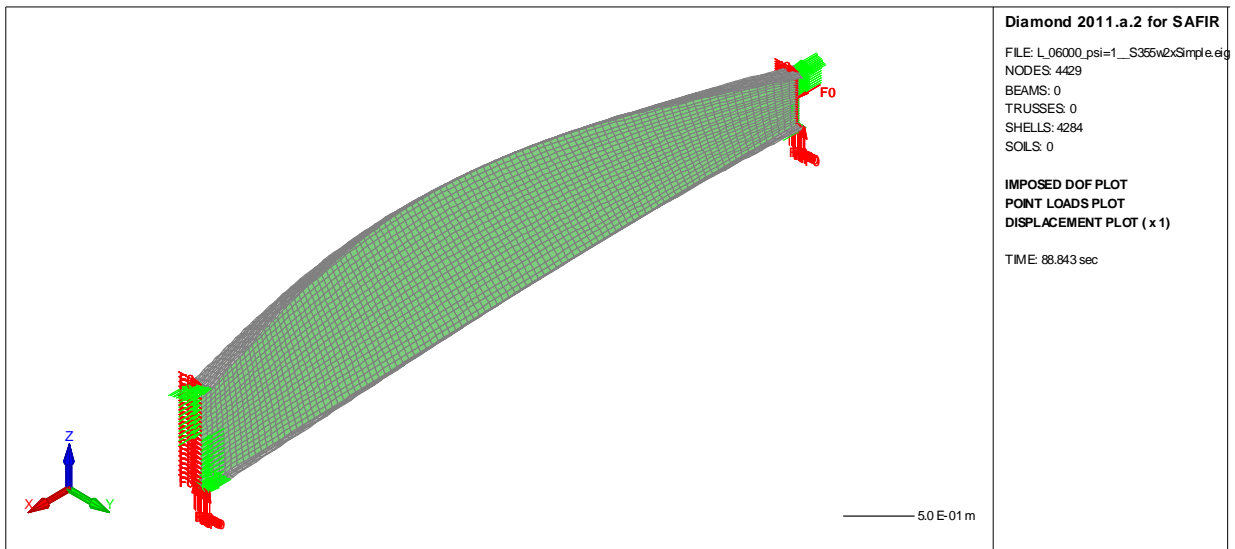


Fig. 16 – Global failure mode of the tapered beam calculated with RUBY ($\alpha_{cr} = 88.843$).

In the next figure the ANSYS results are shown. The applied bending moment is $M_{ed} = 0.2177$ kN.m.

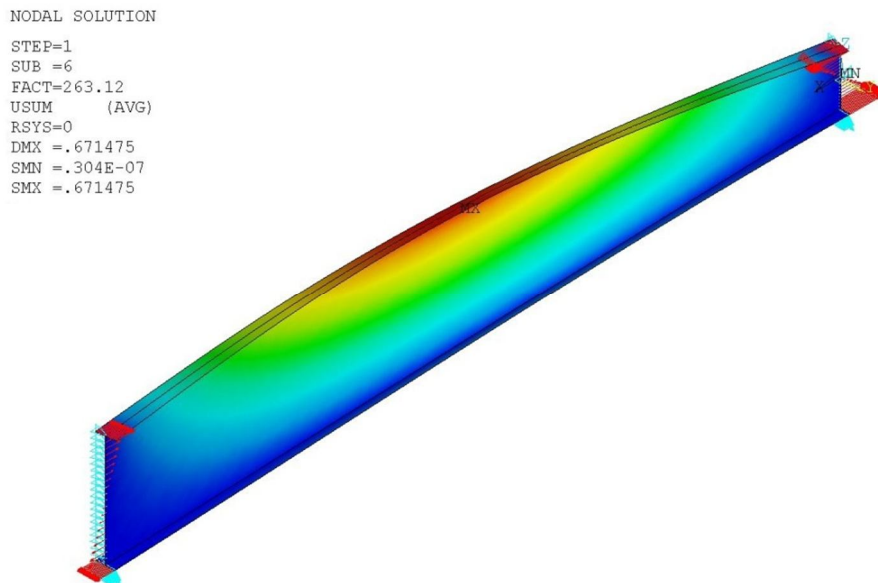


Fig. 17 – Global failure mode of the tapered beam calculated with ANSYS ($\alpha_{cr} = 263.12$).

In Table 6 and Table 7 the results obtained are summarized and compared with the software LTBeam and ANSYS. RUBY and ANSYS results were calculated using Eq. (3).

Table 6 – Results of the comparison of a tapered beam submitted to uniform bending moment in terms of critical load with LTBeam.

Mode	M_{cr} RUBY - FEM [kN.m]	M_{cr} LTBeam [kN.m]	Relative Diff (%)
Lateral-torsional	57.33	57.43	-0.181

Table 7 – Results of the comparison of a tapered beam submitted to uniform bending moment in terms of critical load with ANSYS.

Mode	M_{cr} RUBY - FEM [kN.m]	M_{cr} ANSYS - FEM [kN.m]	Relative Diff (%)
Lateral-torsional	57.33	57.28	0.087

The values in the previous table show a good agreement between the results obtained with RUBY software and calculated with the software LTBeam and also with ANSYS.

The local failure mode is also shown in Fig. 18.

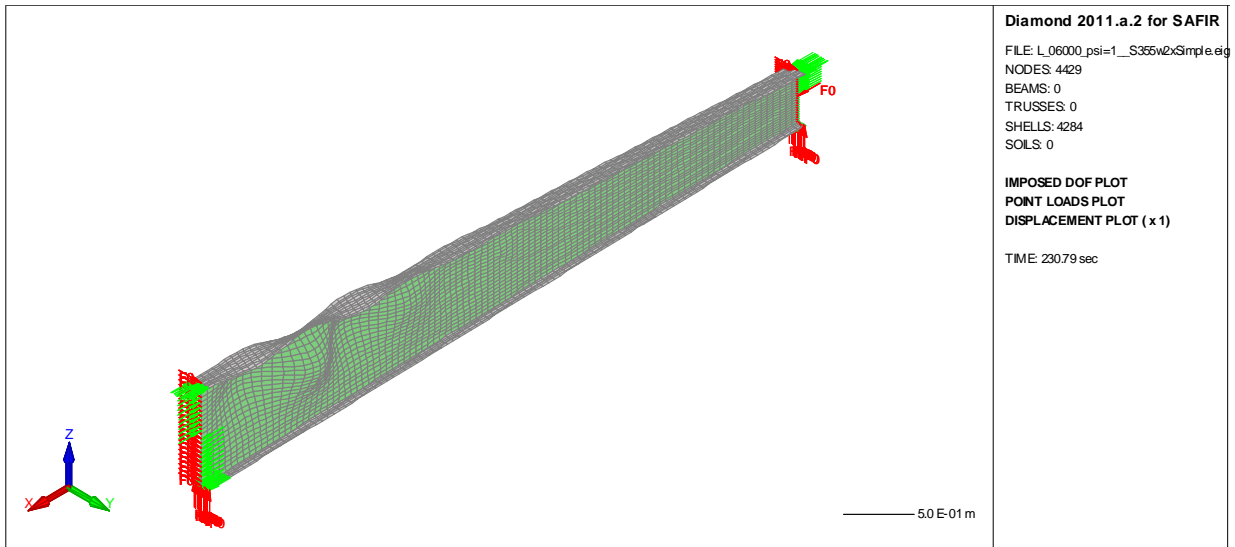


Fig. 18 – Local failure mode of the tapered beam calculated with RUBY ($\alpha_{cr} = 230.79$).

And the corresponding results of ANSYS are shown in the next figure.

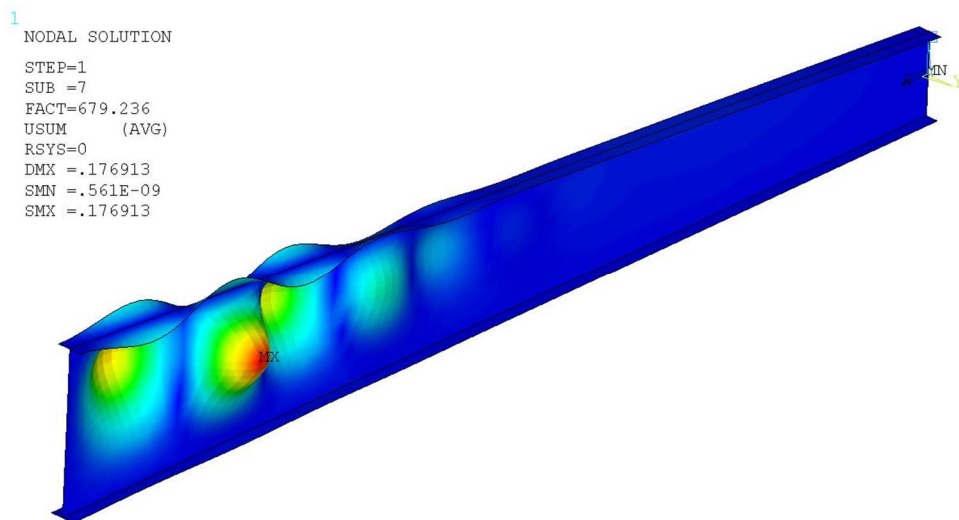


Fig. 19 – Local failure mode of the tapered beam calculated with ANSYS ($\alpha_{cr} = 679.236$).

In Table 8 the results for this local mode are compared between RUBY and ANSYS. Results are calculated using Eq. (3).

Table 8 – Results of the comparison of a tapered beam submitted to uniform bending moment in terms of critical load with ANSYS.

Mode	M_{cr} RUBY - FEM [kN.m]	M_{cr} ANSYS - FEM [kN.m]	Relative Diff (%)
Local buckling	148.92	147.87	0.710

For the local mode, it can be seen again a very good agreement between the results between RUBY and ANSYS.

2.3 Benchmark study of the cross-sectional resistance of an I-shape profile under pure bending at elevated temperature

The cross-sectional resistance of an I-shape profile under pure bending at elevated temperatures has been analysed. The rules of EN1993-1-2 (CEN, 2005b) have been followed regarding the material law of steel at elevated temperatures. Recommendations of EN1993-1-5 regarding the numerical modelling by Finite Element Method have been taken into account. SAFIR and ANSYS computer codes have been used and the results obtained with both software programs are compared.

The numerical model used in SAFIR is depicted in Fig. 20 and described next.

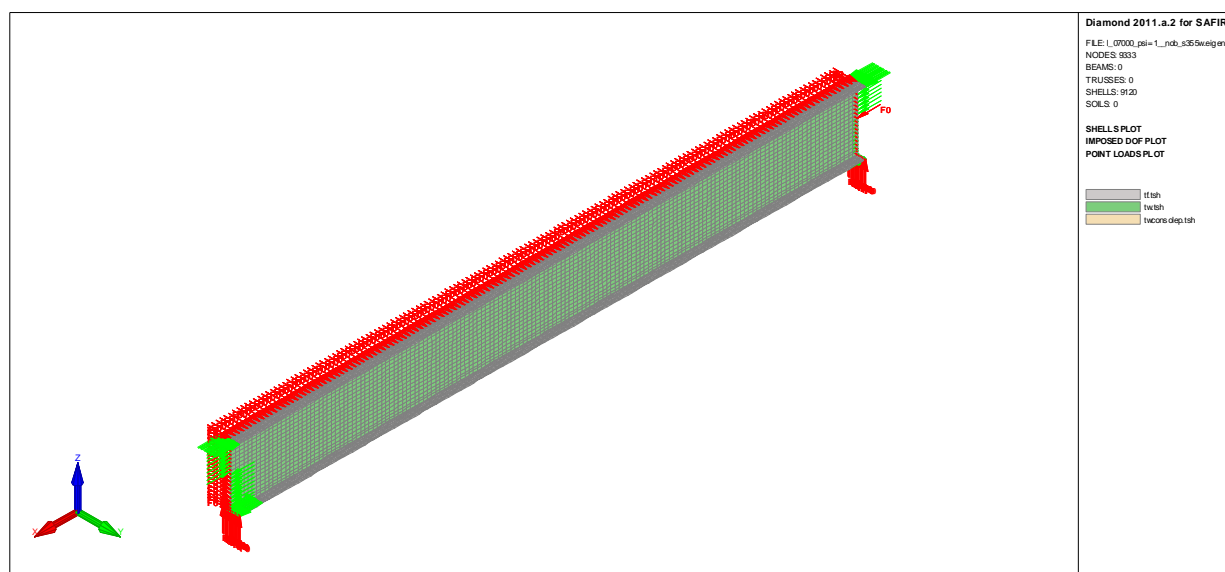


Fig. 20 - Example of the numerical model adopted in this study.

The beam has been discretized into several quadrangular shell elements with four nodes and six degrees of freedom (3 translations and 3 rotations). In SAFIR these shell elements adopt the Kirchhoff's theory formulation with a total co-rotational description. The material law is a two-dimensional constitutive relation with the von Mises yield surface. The integration on the shell element follows a Gauss scheme with 2×2 points on the surface and 3 points through the thickness. Since shell elements are used the root fillet of the cross-sections has been disregarded.

A mesh density study has been performed and the solution seemed to converge by using 20 shell elements for the flange, 20 shell elements for the web and 100 shell elements along the length.

The uniform bending moment is applied as a sum of nodal forces (see Fig. 21).

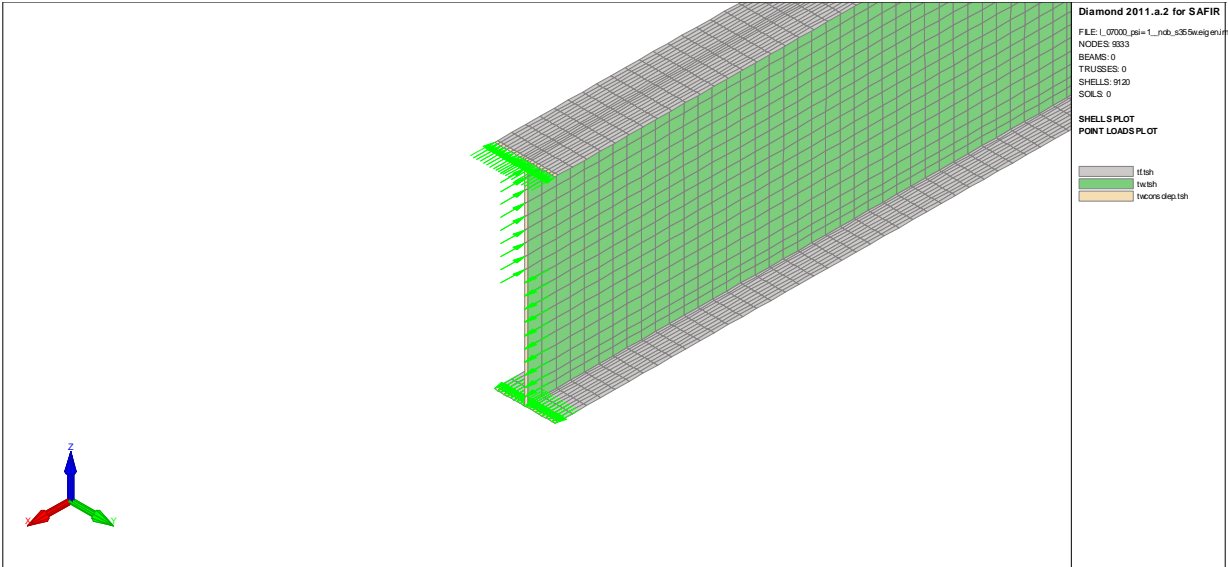


Fig. 21 – Load application in the model.

Some numerical problems have been found when applying these nodal forces with slender cross-sections due to reduced rigidity of its plates. In order to solve this problem, an additional layer of shell elements with increased rigidity has been introduced in the model as it can be seen in Fig. 21.

The beam is considered to be simply supported with the possibility to rotate about the minor axis. From these considerations a fork-support type condition has been modelled through imposing that, in the extremities, vertical displacements of the bottom flange and out-of-the plane horizontal displacements of the web are blocked, to add lateral restraint to the beam, an additional support has been added in the intersection between the upper flange and the web (see Fig. 22).

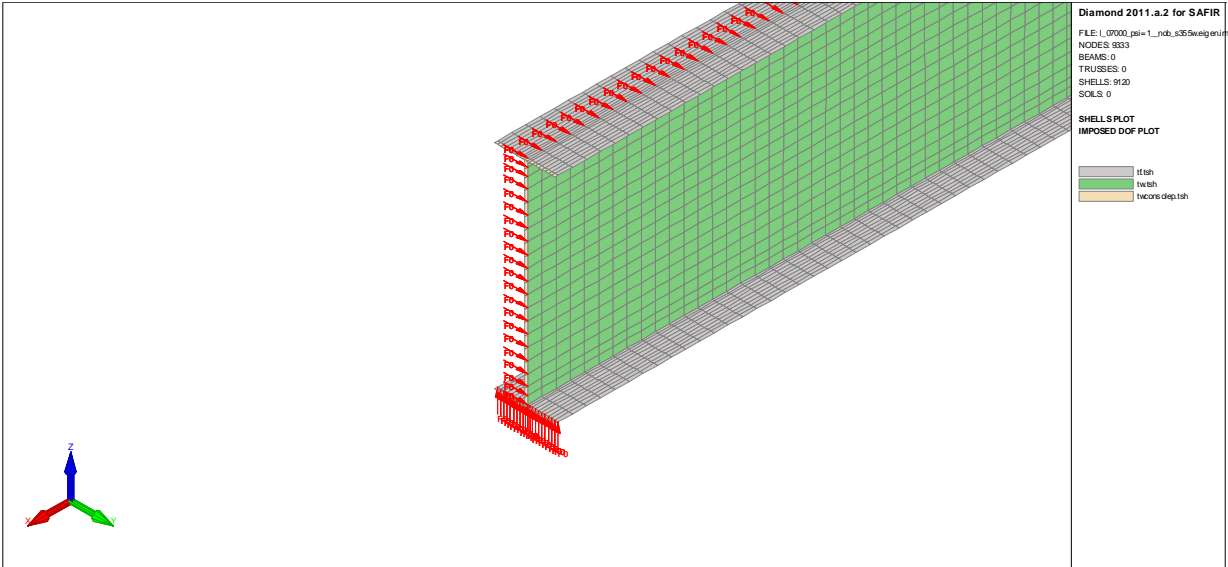


Fig. 22 – End support conditions and lateral restraint.

At elevated temperatures, a constant temperature of 350°C has been used and it has been introduced in the model. The temperature is constant along the cross-section.

Geometrical imperfections have been applied in the model with a shape affine to the first eigen-mode calculated with RUBY, and the amplitude of the imperfections considered are those suggested in Annex C of Part 1.5 of the Eurocode 3: 80% of $b/100$ or 80% of $h_w/100$ (see EN1090-2 table D.2.3 (CEN, 2008)), where b and h_w are the flange width and web height respectively. No residual stresses have been considered in this case.

The amplitude of the imperfection entered in the model depends on where the greatest amplitude of the eigen-mode occurs. In this case, the eigen-mode is controlled by the web as it can be seen in Fig. 23, so amplitude of 7mm has been chosen.

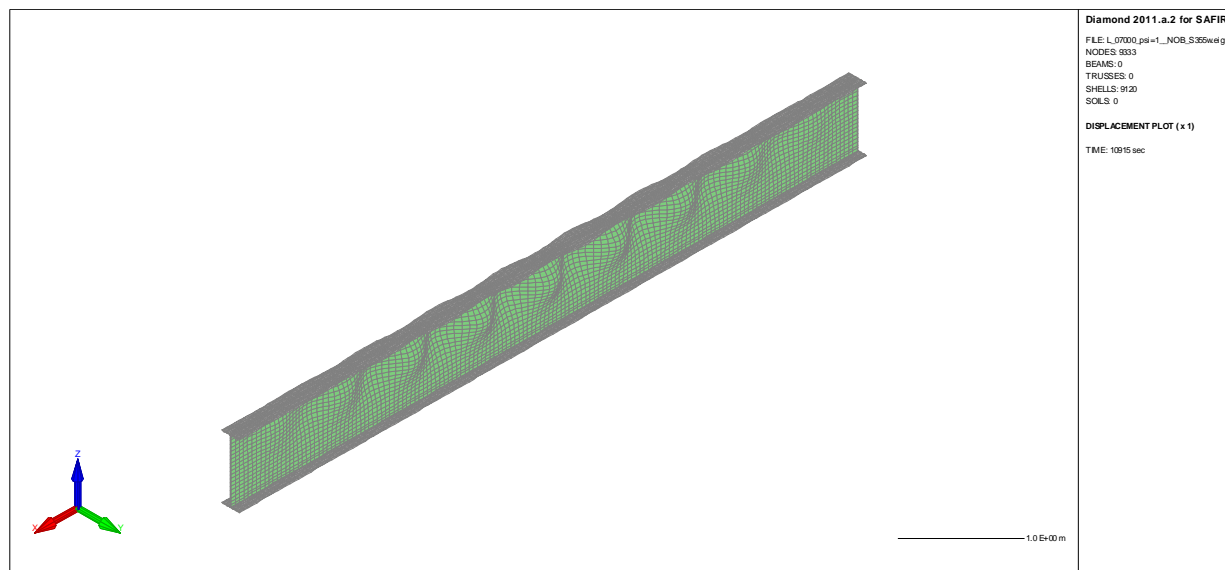


Fig. 23 – Eigen mode ($\alpha_{cr} = 1130.64$).

The numerical model used in ANSYS is identical to the one used in SAFIR. However it is possible to use a rigid-body element to transfer the boundary conditions and loads applied into the shell elements as indicated in Fig. 24. A comparison between entering the loads with this procedure and the procedure described previously (see Fig. 21) is also made showing that the procedures are equivalent and leading to the same results.

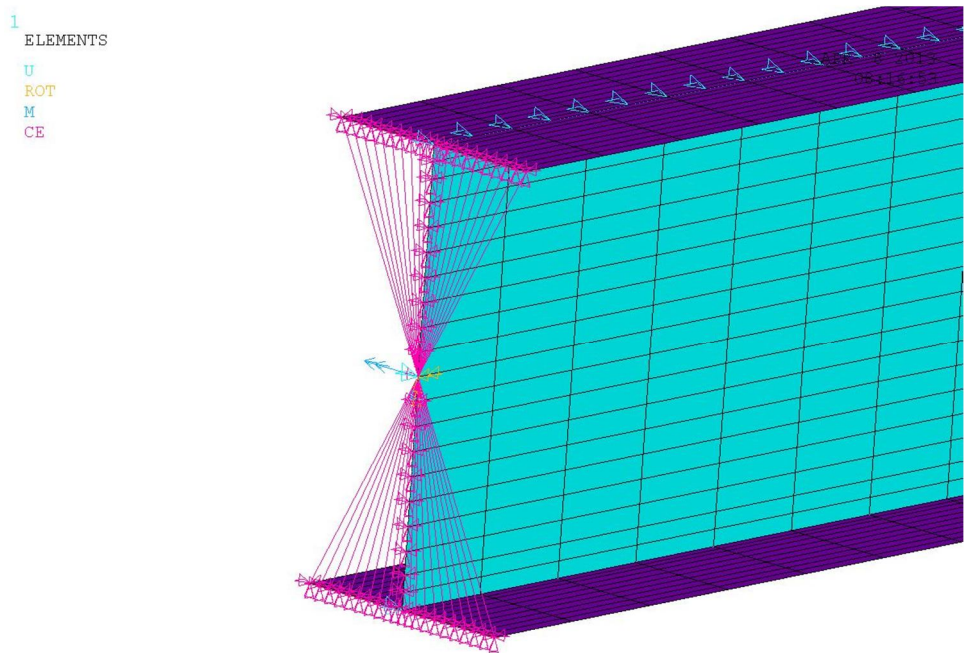


Fig. 24 – Boundary conditions used and the rigid-body element introduced in the ANSYS model.

2.3.1 Comparison of the results obtained with SAFIR and ANSYS

The results obtained with both software programs are shown in Fig. 25 in terms of the ultimate load bearing capacity as a function of the rotation at the support.

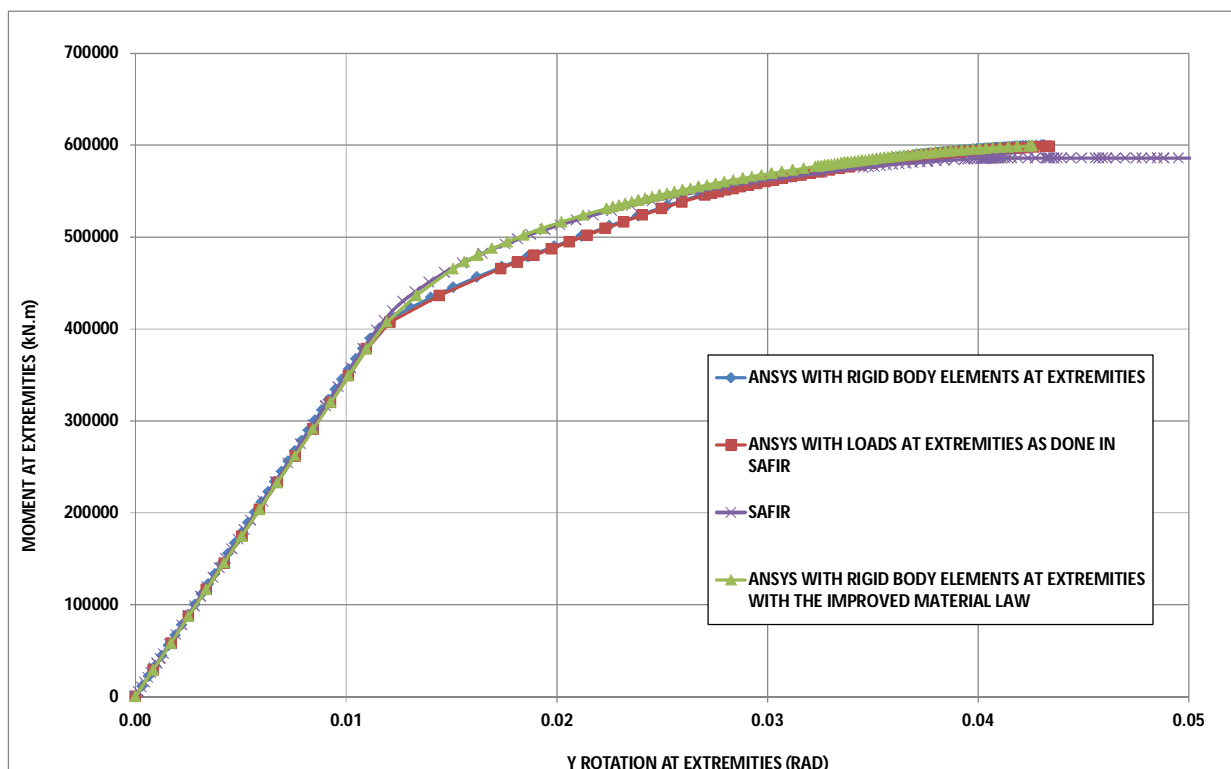


Fig. 25 – Ultimate load bearing capacity of the member function of the rotation of the support.

From this figure, it can be seen that there is a good agreement between the results. It is also evident that using the rigid-body element in ANSYS or the procedure used in SAFIR of adding an additional layer of thicker elements at the extremities leads to the same results, thus

showing that this procedure of SAFIR is equivalent. These results also show better correlation between SAFIR and ANSYS with an improved material law, this question is addressed in the next section.

In the next figures the deformed shape of the beam at failure obtained with SAFIR and ANSYS is shown.

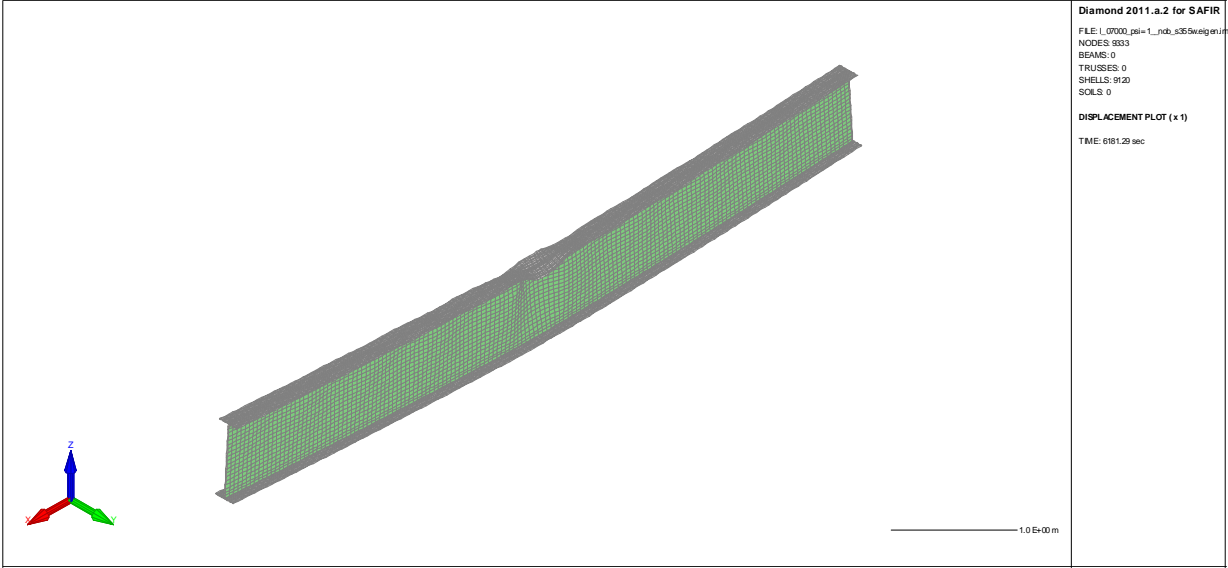


Fig. 26 – Deformed shape of the beam at failure with software SAFIR.

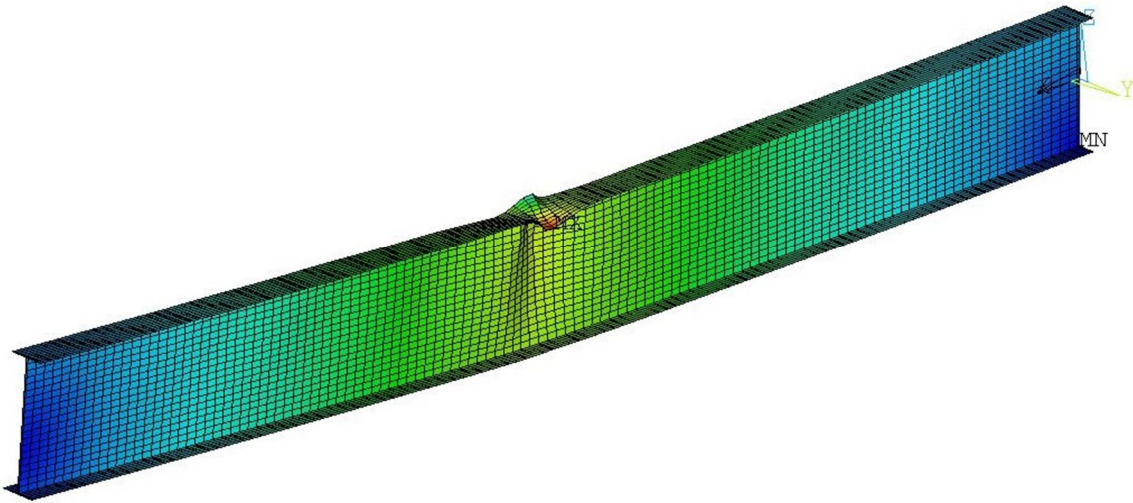


Fig. 27 – Deformed shape of the beam at failure with software ANSYS.

From the previous figures it can be seen again that both computer codes show the same deformed shape at failure.

2.3.2 Improvement of the procedure used in ANSYS to describe the steel material law at elevated temperatures

According to EN1993-1-2 the steel constitutive law at elevated temperatures is described with the formulae of Table 9.

Table 9 – Steel constitutive law at elevated temperatures

Strain range	Stress σ	Tangent modulus
$\varepsilon \leq \varepsilon_{p,\theta}$	$\varepsilon E_{a,\theta}$	$E_{a,\theta}$
$\varepsilon_{p,\theta} < \varepsilon < \varepsilon_{y,\theta}$	$f_{p,\theta} - c + (b/a)[a^2 - (\varepsilon_{y,\theta} - \varepsilon)^2]^{0.5}$	$\frac{b(\varepsilon_{y,\theta} - \varepsilon)}{a[a^2 - (\varepsilon_{y,\theta} - \varepsilon)^2]^{0.5}}$
$\varepsilon_{y,\theta} \leq \varepsilon \leq \varepsilon_{t,\theta}$	$f_{y,\theta}$	0
$\varepsilon_{t,\theta} < \varepsilon < \varepsilon_{u,\theta}$	$f_{y,\theta}(1 - (\varepsilon - \varepsilon_{t,\theta})/(\varepsilon_{u,\theta} - \varepsilon_{t,\theta}))$	-
$\varepsilon = \varepsilon_{u,\theta}$	0.00	-
Parameters	$\varepsilon_{p,\theta} = f_{p,\theta}/E_{a,\theta}$ $\varepsilon_{y,\theta} = 0.02$	$\varepsilon_{t,\theta} = 0.15$ $\varepsilon_{u,\theta} = 0.20$
Functions	$a^2 = (\varepsilon_{y,\theta} - \varepsilon_{p,\theta})(\varepsilon_{y,\theta} - \varepsilon_{p,\theta} + c/E_{a,\theta})$ $b^2 = c(\varepsilon_{y,\theta} - \varepsilon_{p,\theta})E_{a,\theta} + c^2$ $c = \frac{(f_{y,\theta} - f_{p,\theta})^2}{(\varepsilon_{y,\theta} - \varepsilon_{p,\theta})E_{a,\theta} - 2(f_{y,\theta} - f_{p,\theta})}$	

In Table 9 and Fig. 28 the meaning of the symbols is:

- $f_{y,\theta}$ effective yield strength;
- $f_{p,\theta}$ proportional limit;
- $E_{a,\theta}$ slope of the linear elastic range;
- $\varepsilon_{p,\theta}$ strain at the proportional limit;
- $\varepsilon_{y,\theta}$ yield strain;
- $\varepsilon_{t,\theta}$ limiting strain for yield strength;
- $\varepsilon_{u,\theta}$ ultimate strain.

In Fig. 28 the stress-strain relationship for carbon steel at elevated temperatures is shown.

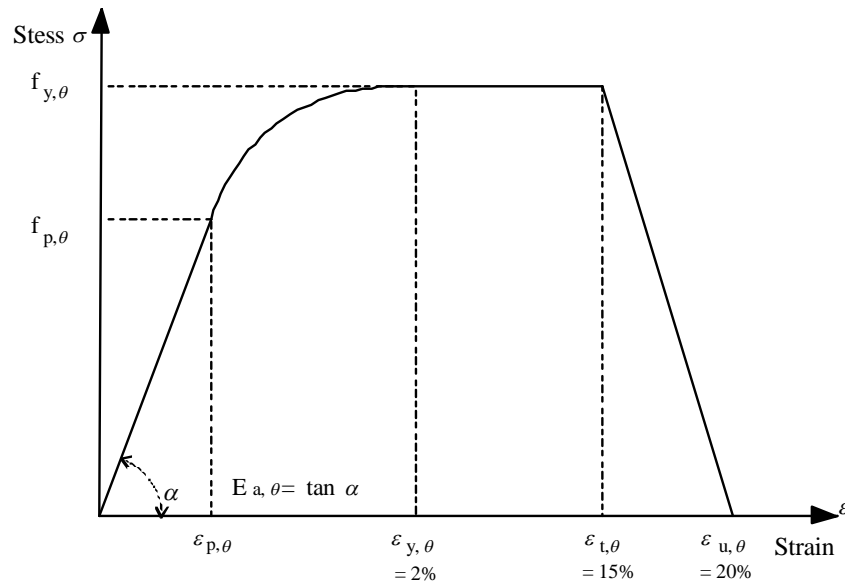


Fig. 28 - Stress-strain relationship for carbon steel at elevated temperatures

Since SAFIR is a Finite Element Method software program that was developed especially for the thermo-mechanical analysis of structures under fire conditions these formulae of the material law is implemented directly in the code. On the other hand, ANSYS is a generic Finite Element Method software program and the material law model is defined by the user through a set of stress-strain points. During the STSM comparisons have been made between SAFIR and ANSYS showing that the number of points used to define the material law (Fig. 20) in ANSYS influences the results. Fig. 25 compares the results of using 20 points for defining the material law with the results using 100 points and it can be seen in this figure that the material law model in ANSYS defined by 100 points leads to a better correlation with SAFIR.

3 Description of the main results obtained

With this STSM the main results obtained were:

- The development of the software RUBY that permits the SAFIR users to perform a linear buckling analysis overcoming this current limitation on the SAFIR software.
- The possibility of using RUBY to apply geometric imperfections to SAFIR models based on the shape of eigen-modes obtained with a linear buckling analysis.
- A comparison between SAFIR and ANSYS showing very good agreement between this computer codes.
- Finding that the procedure used in SAFIR for applying the loads by using additional thicker shell elements at the extremities of the members is equivalent to the use of a rigid-body element (the rigid-body element is currently not available in SAFIR).
- Finding that the number of points that describe the material law model in ANSYS influences the results and by using 100 points to describe the material law model there is a better correlation between ANSYS and SAFIR.

References

- ANSYS, "ANSYS User's Manual for Revision 14.0 – Volume IV – Theory", Swanson Analysis SYSTEM, INC., Houston USA, 2011.
- CAST3M. CAST 3M is a research FEM environment; its development is sponsored by the French Atomic Energy Commission; 2012. <<http://www-cast3m.cea.fr/>>.
- CEN European Committee for Standardisation, EN 1993–1–1, Eurocode 3: Design of Steel Structures – Part 1–1: General rules and rules for buildings, 2005a.
- CEN European Committee for Standardisation, EN 1993–1–2, Eurocode 3: Design of Steel Structures – Part 1–2: General rules - Structural fire design, 2005b.
- CEN European Committee for Standardisation. EN 1090–2, Execution of steel structures and aluminium structures - Part 2: Technical requirements for steel structures. Brussels, Belgium, European Committee for Standardisation, 2008.
- CEN European Committee for Standardisation, EN 1993–1–5, Eurocode 3, Design of Steel Structures – Part 1–5: Plated structural elements, 2012.
- CTICM Centre Technique Industriel de la Construction Métallique, LTBeam version 1.0.11 – A software to analyse the "Lateral Torsional Buckling of Beams" under bending action about their major axis, Saint-Aubin, Paris, France, 2012.
- Franssen, J.-M. (2005). SAFIR, A Thermal/Structural Program Modelling Structures under Fire, Engineering Journal, A.I.S.C. 42(3): 143-158, 2005.