

DIANA 10

Integrated Environment

Staalbouwdag Ab van den Bos



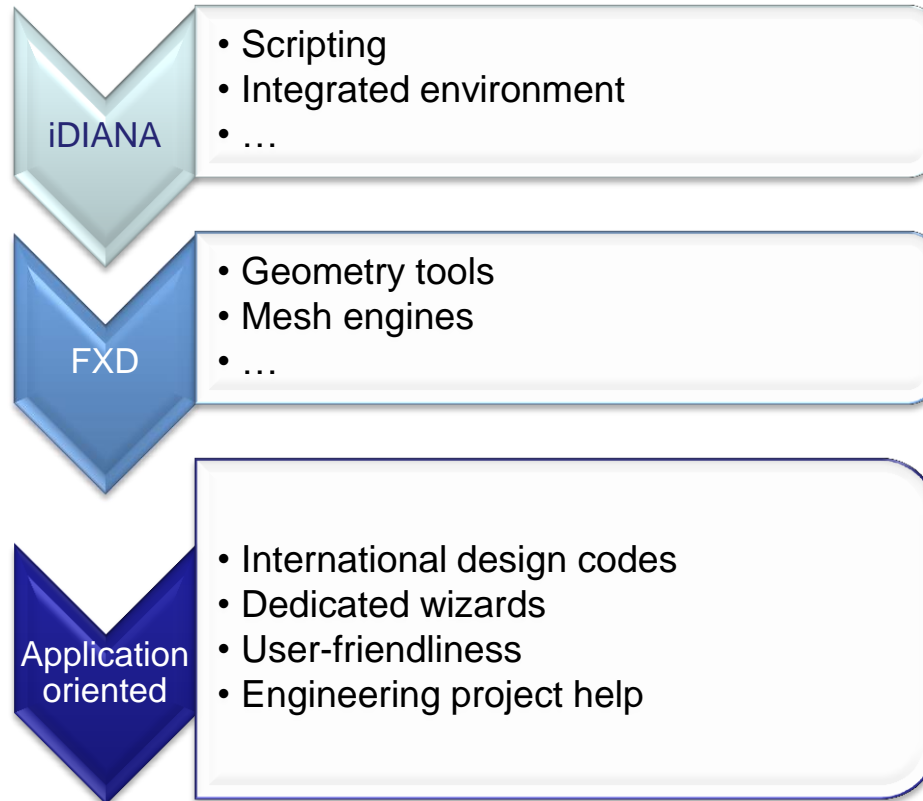
Civil Engineering
Geotechnical Engineering
Petroleum Engineering

Introduction of a new TNO DIANA service

- TNO DIANA *Engineering* began in April 2014
 - to help users with their own projects.
 - when time is critical and human resources are sensitive to planning schemes.
 - when ‘know how’ or workflow for a new topic has not yet been standardised.
 - different levels of aid include;
 - Setting up models
 - Carrying out analysis
 - Reporting



Background and development direction



DIANA 10

resulting in a new integrated environment

A friendly face to finite elements;

- Allows creative designs in a friendly environment
- Scripting for the definition of repetitive jobs
- Parameterisation reduces duplication and enhances productivity
- Powerful parallel processing reduces run times and your wait times
- Comprehensive material models allow high order analysis
- Soil structure interaction
- Fluid structure interaction
- Comprehensive seismic analysis options

Features from Civil Engineers for Civil Engineers

DIANA 10

Steel connections - earthquake loading

Ballio, Mazzolani - Strutture In Acciaio

- Material ductility
- Joint ductility (redistribution)
- Global structural ductility

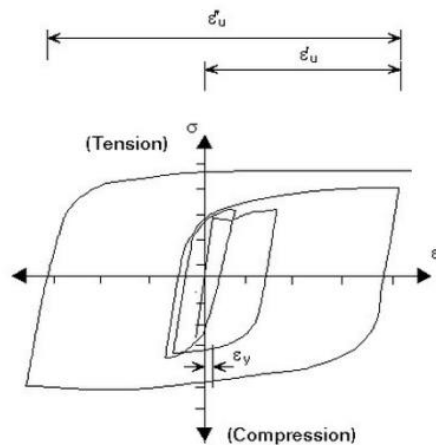


Figure 1 Stress-strain relationship of a structural steel under uniaxial hysteretic loops

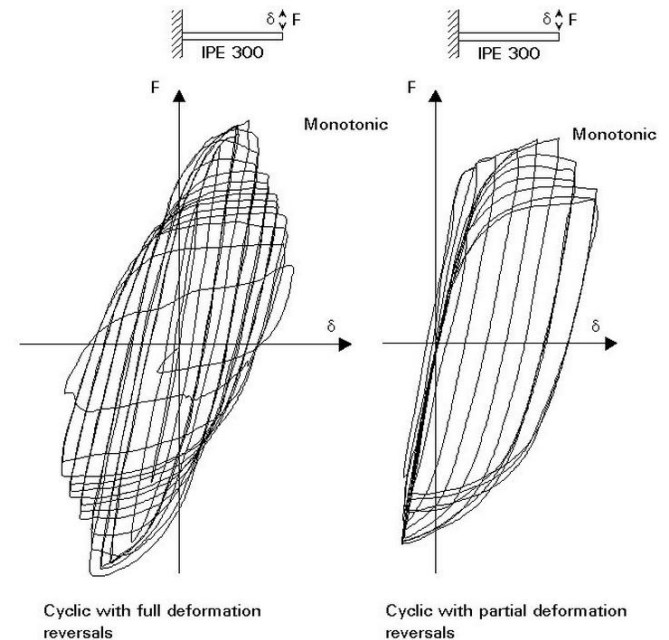
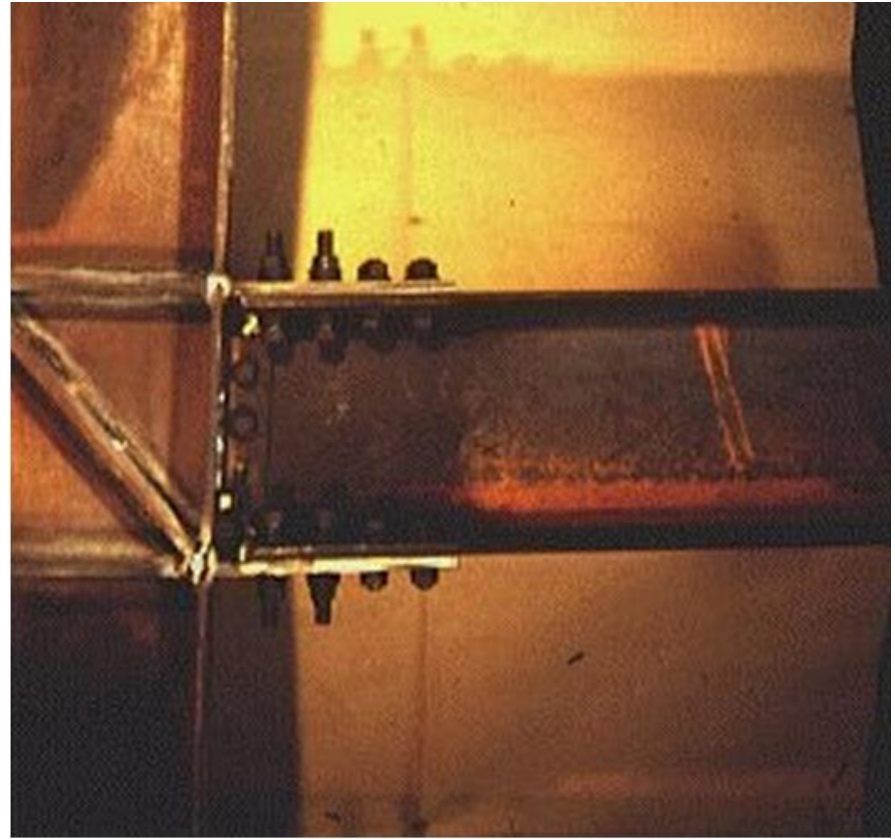
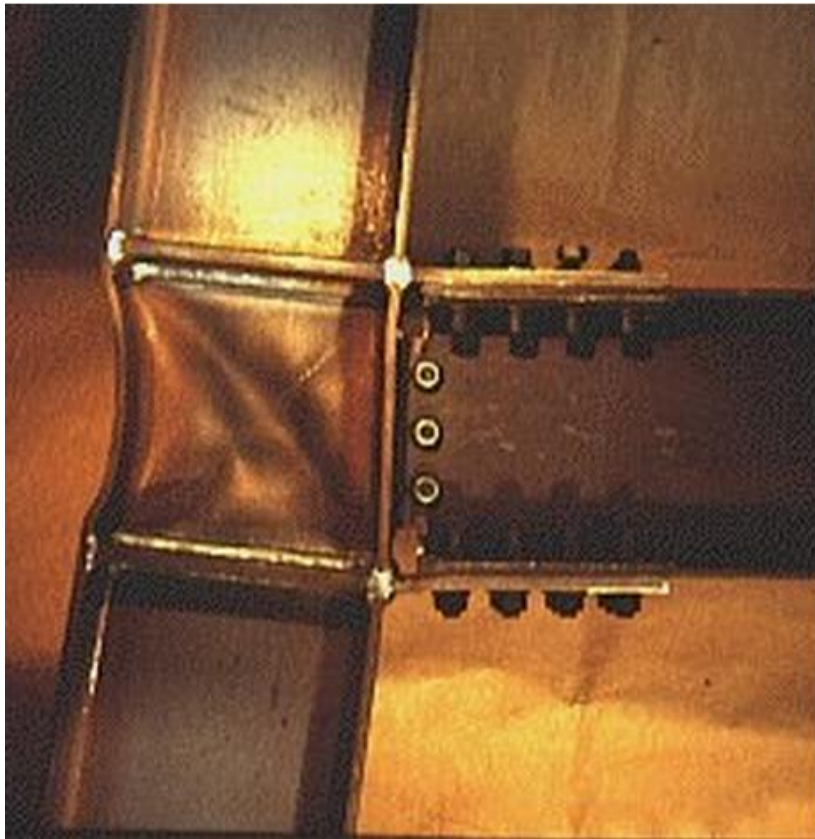


Figure 3 Influence of loading history on the cyclic behaviour of beams

Steel connections - earthquake loading



$R_d \geq 1,20 R_{fy}$ in other words : connection \geq adjacent element

Steel connections - earthquake loading

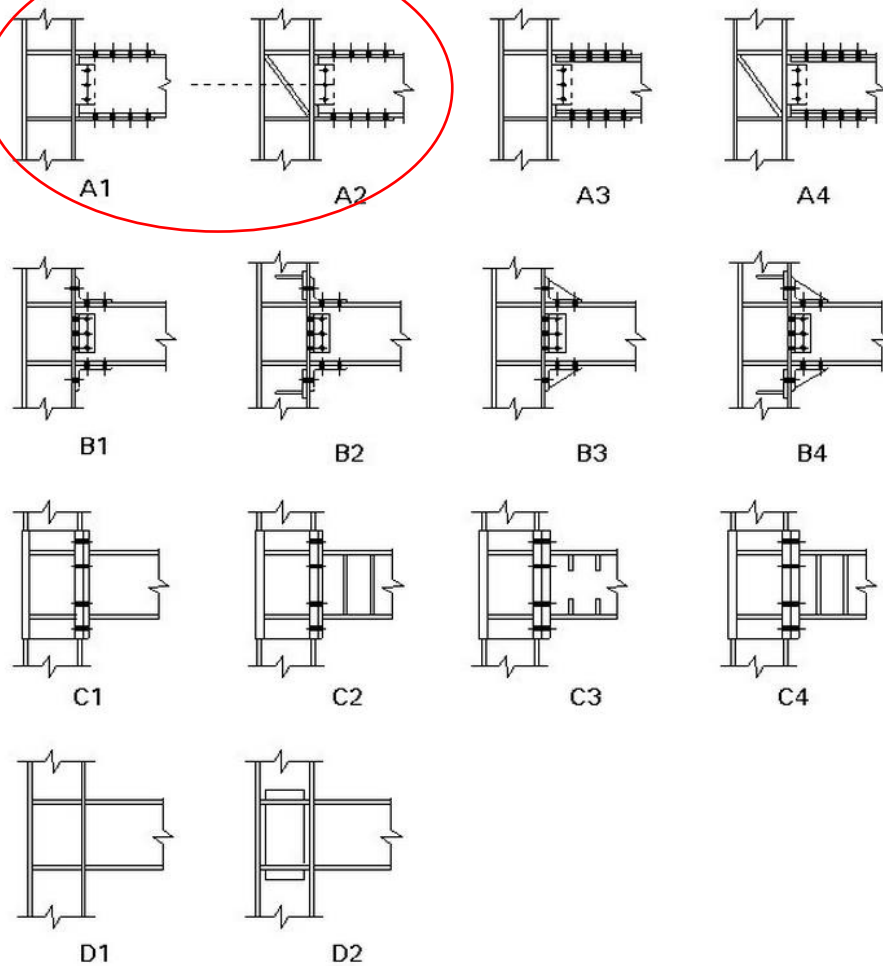


Figure 11 Types of beam-to-column connections tested by Ballio, Mazzolani et al.

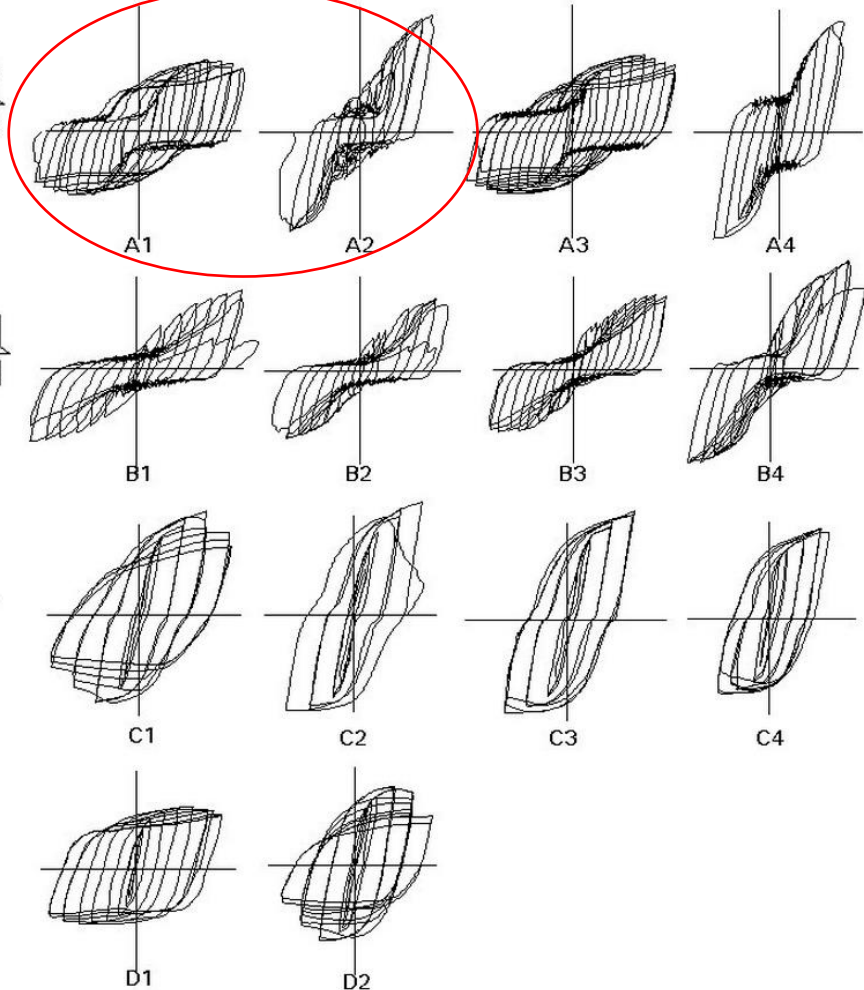
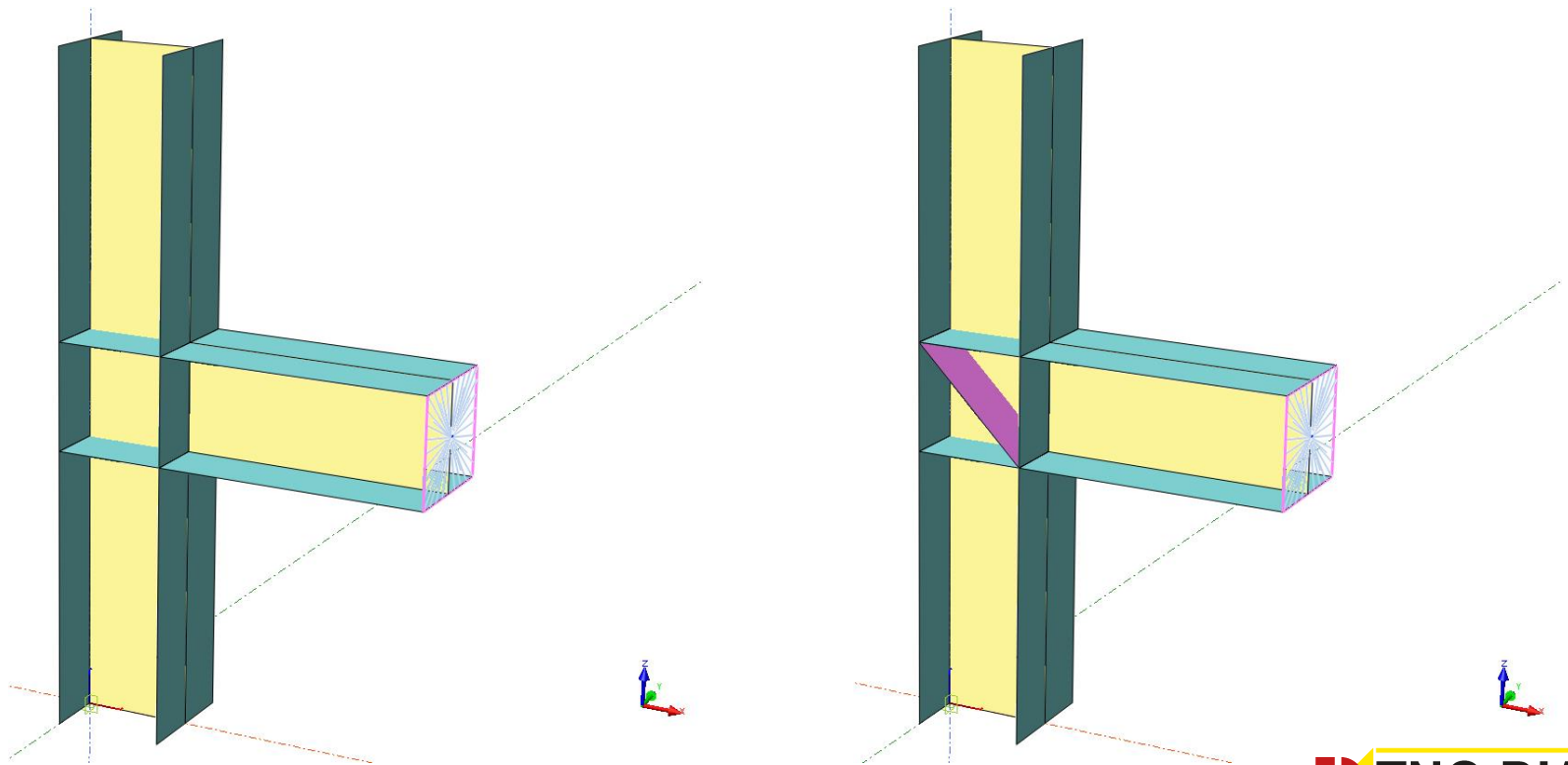


Figure 12 Hysteresis loops of beam-to-column connections tested by Ballio, Mazzolani et al.

Steel connections - earthquake loading

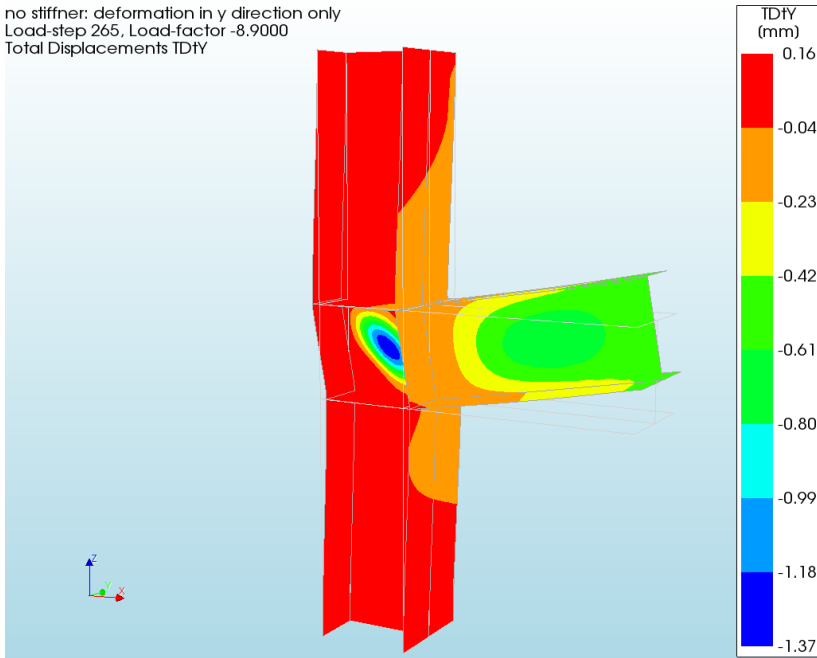
Standard profile: HEA180

Additional stiffener (purple): $t=6$ mm

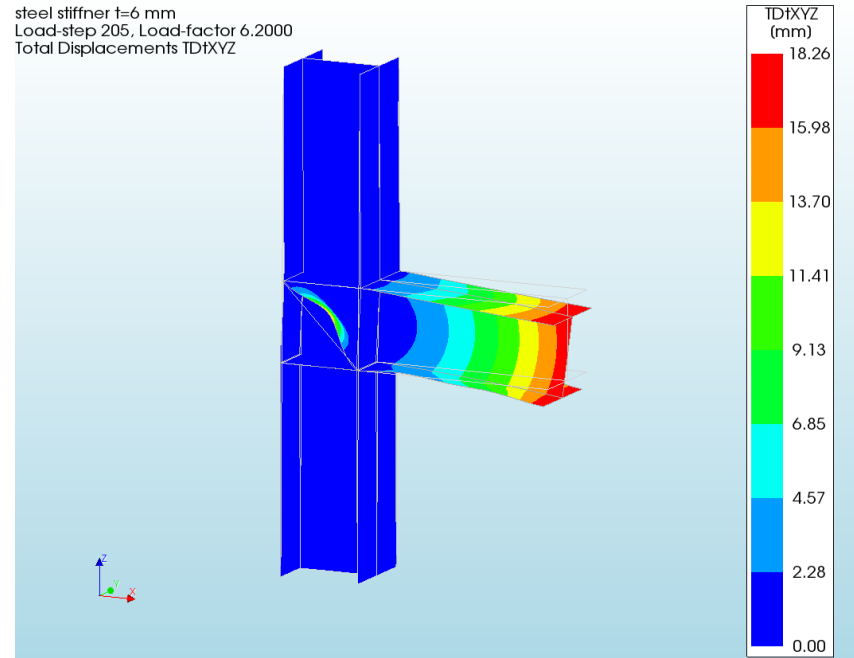


Steel connections - earthquake loading

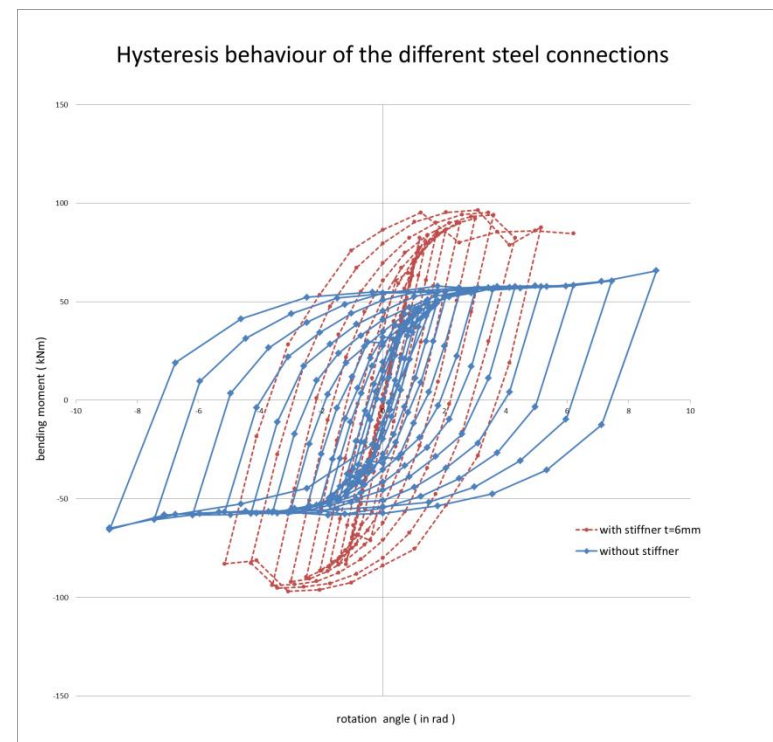
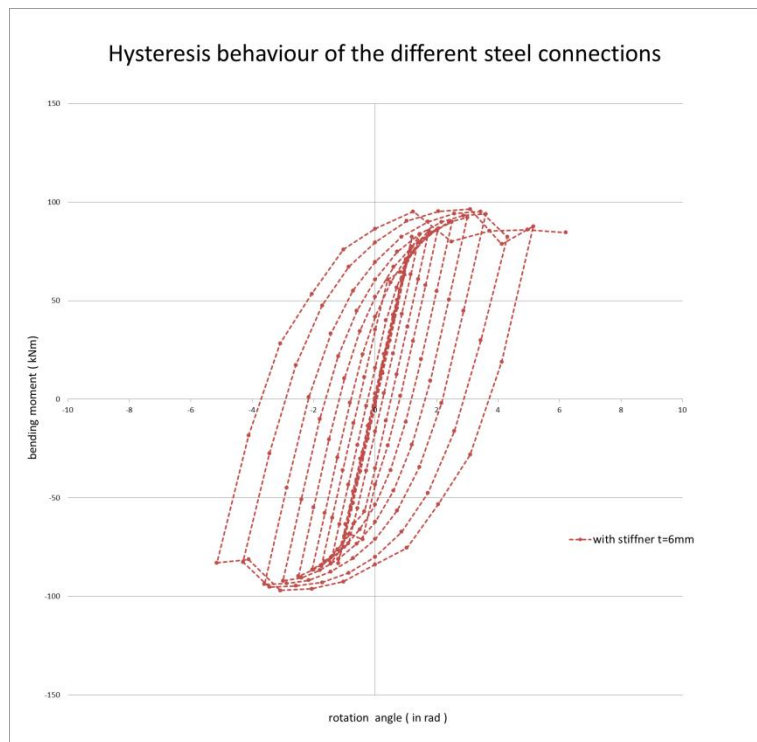
no stiffner: deformation in y direction only
Load-step 265, Load-factor -8.9000
Total Displacements TDtY



steel stiffner t=6 mm
Load-step 205, Load-factor 6.2000
Total Displacements TDtXYZ

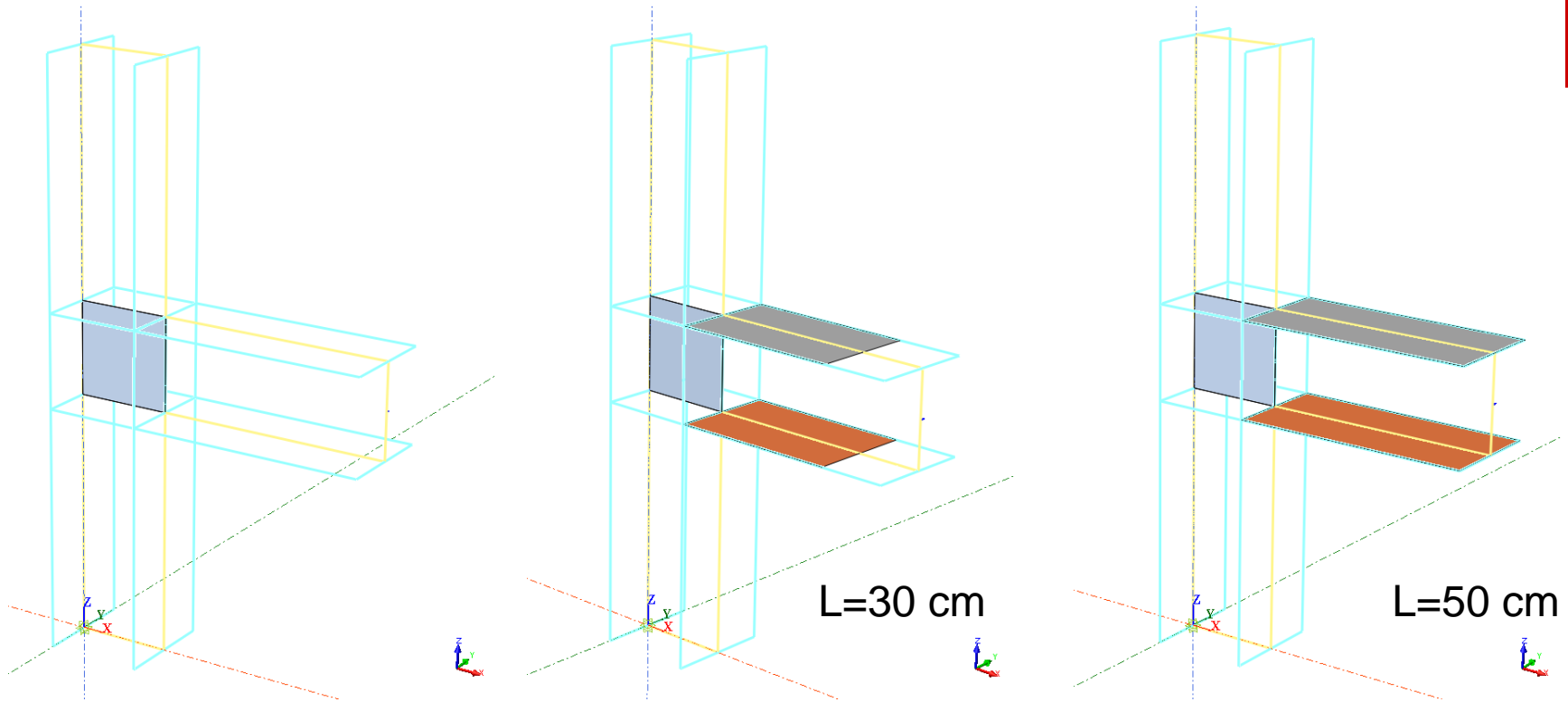


Steel connections - earthquake loading



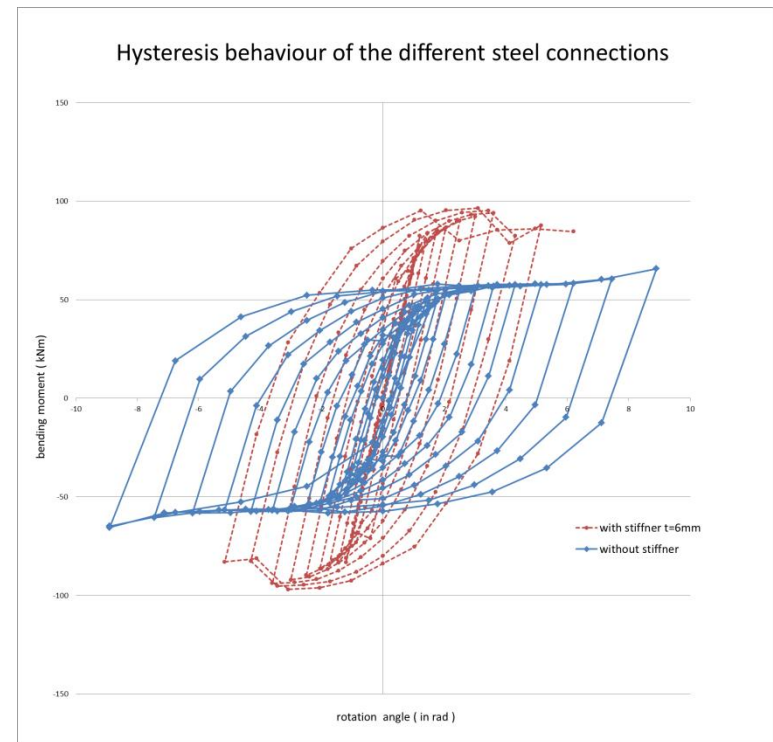
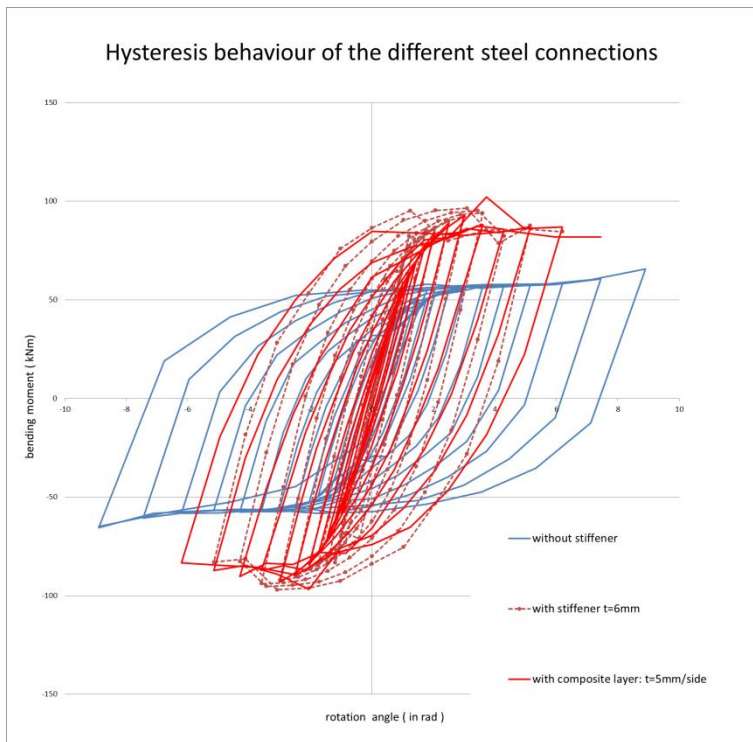
Steel connections earthquake loading

Composite layer: $t=5\text{mm}$, variant solutions



Steel connections earthquake loading

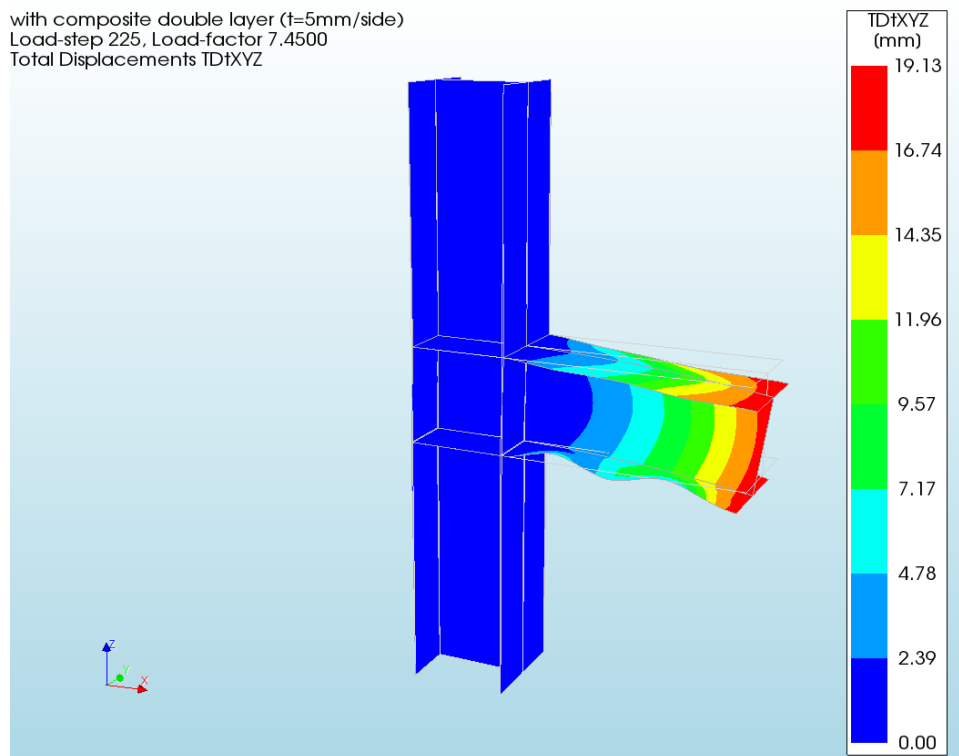
Only 5 mm in the section joint



Steel connections earthquake loading

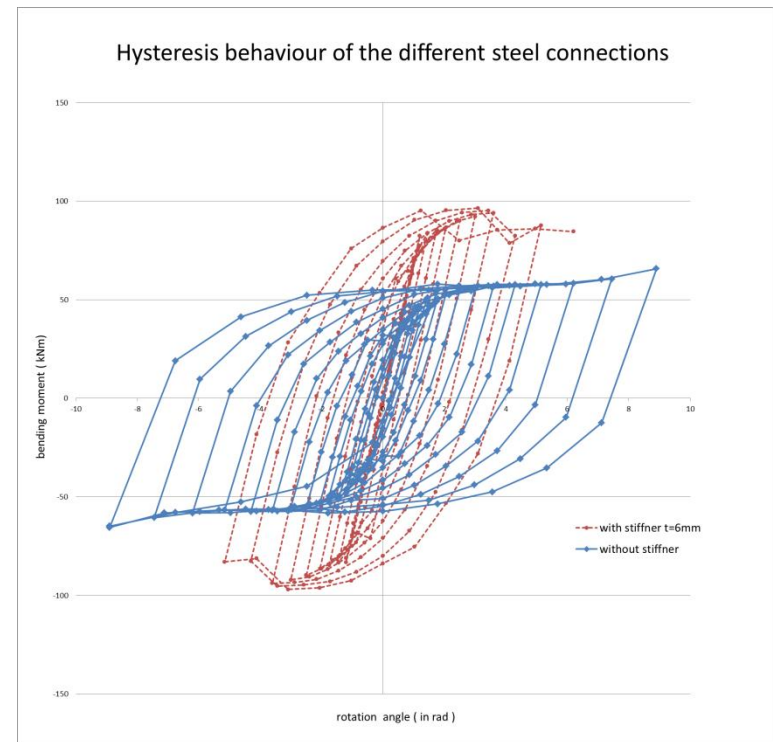
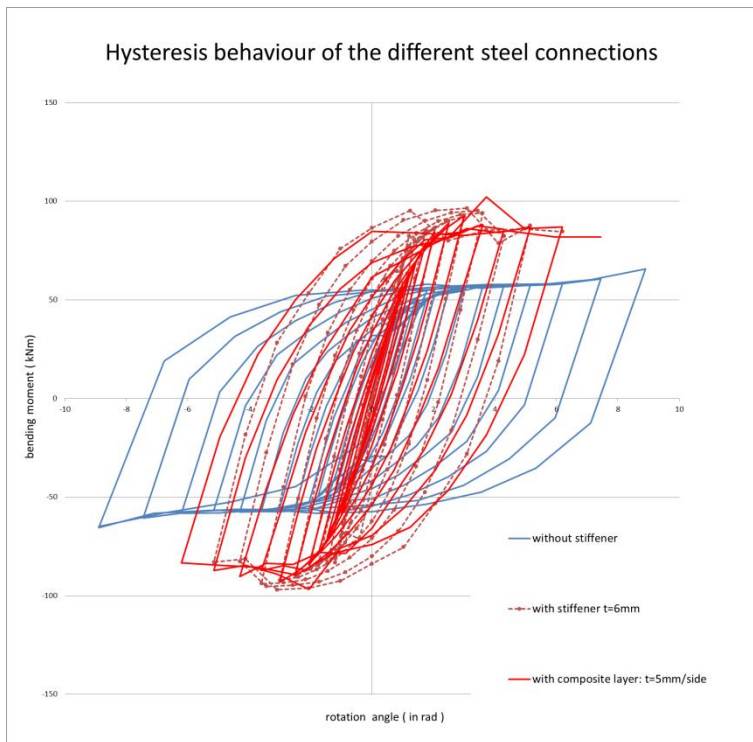
Only 5 mm in the section joint

with composite double layer (t=5mm/side)
Load-step 225, Load-factor 7.4500
Total Displacements TDtXYZ



Steel connections earthquake loading

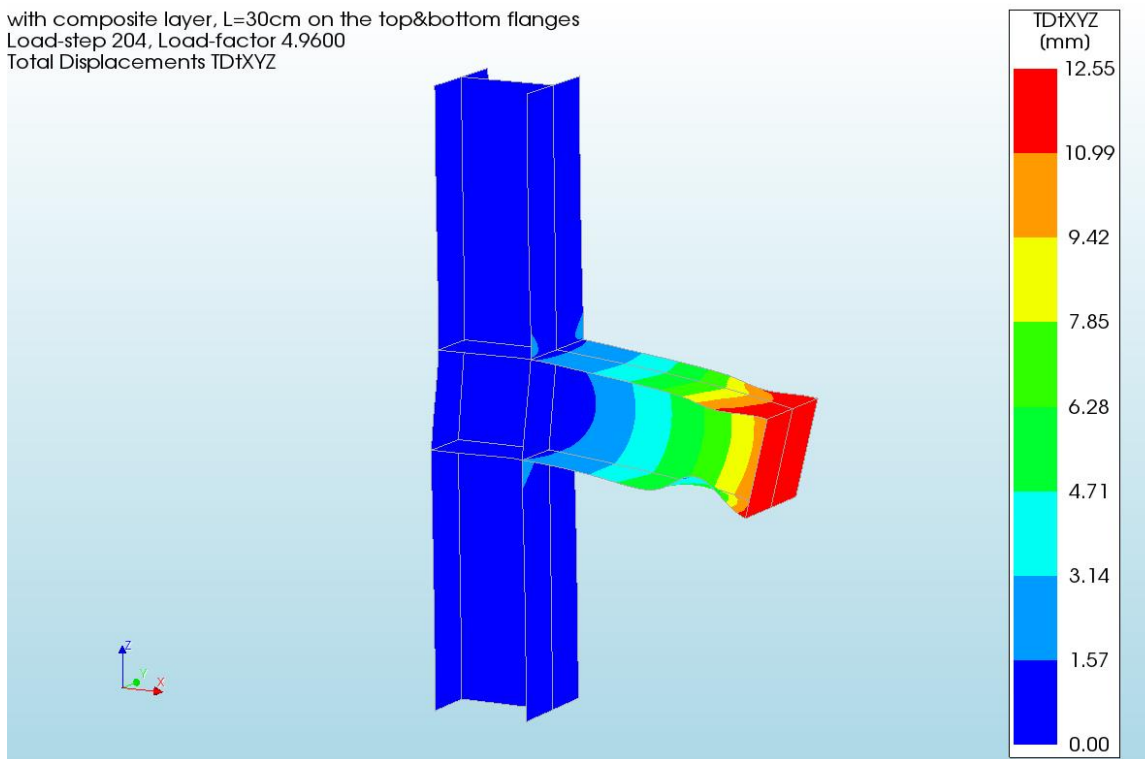
Only 5 mm in the section joint



Steel connections earthquake loading

With extra layers on the top&bottom flanges, L=30 cm

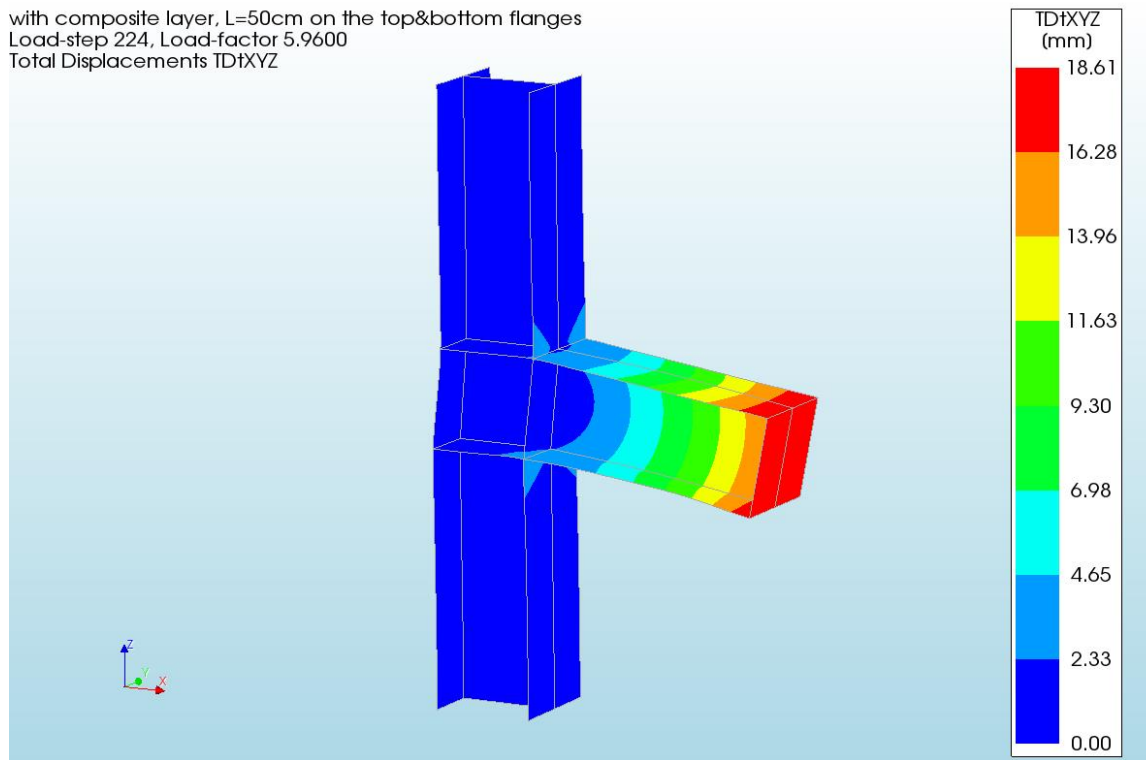
with composite layer, L=30cm on the top&bottom flanges
Load-step 204, Load-factor 4.9600
Total Displacements TDtXYZ



Steel connections earthquake loading

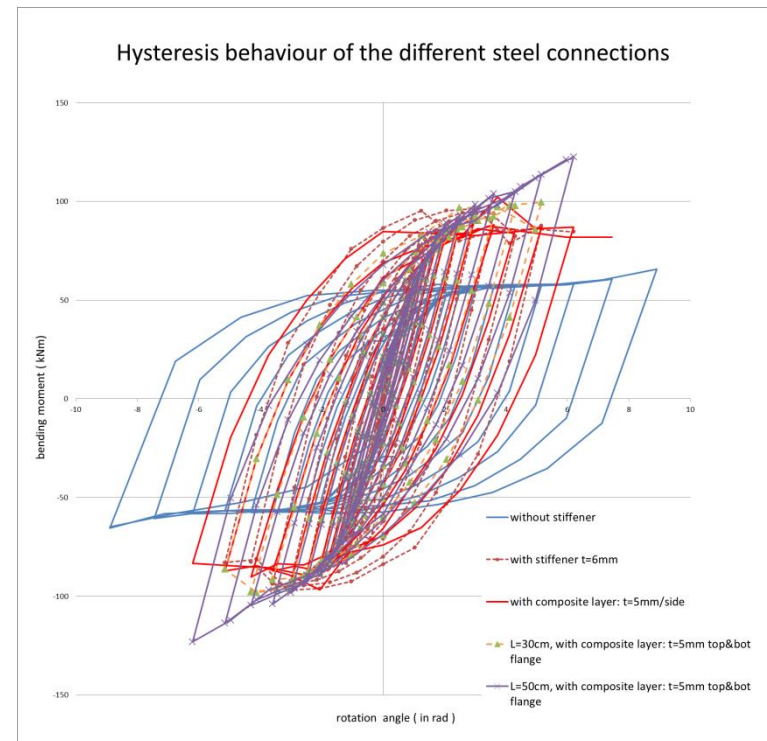
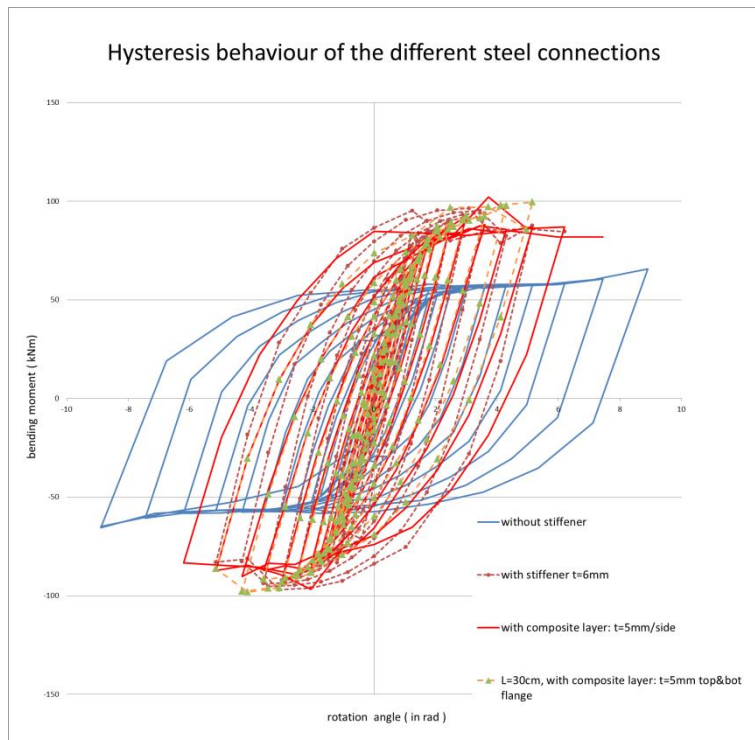
With extra layers on the top&bottom flanges, L=50 cm

with composite layer, L=50cm on the top&bottom flanges
Load-step 224, Load-factor 5.9600
Total Displacements TDtXYZ



Steel connections earthquake loading

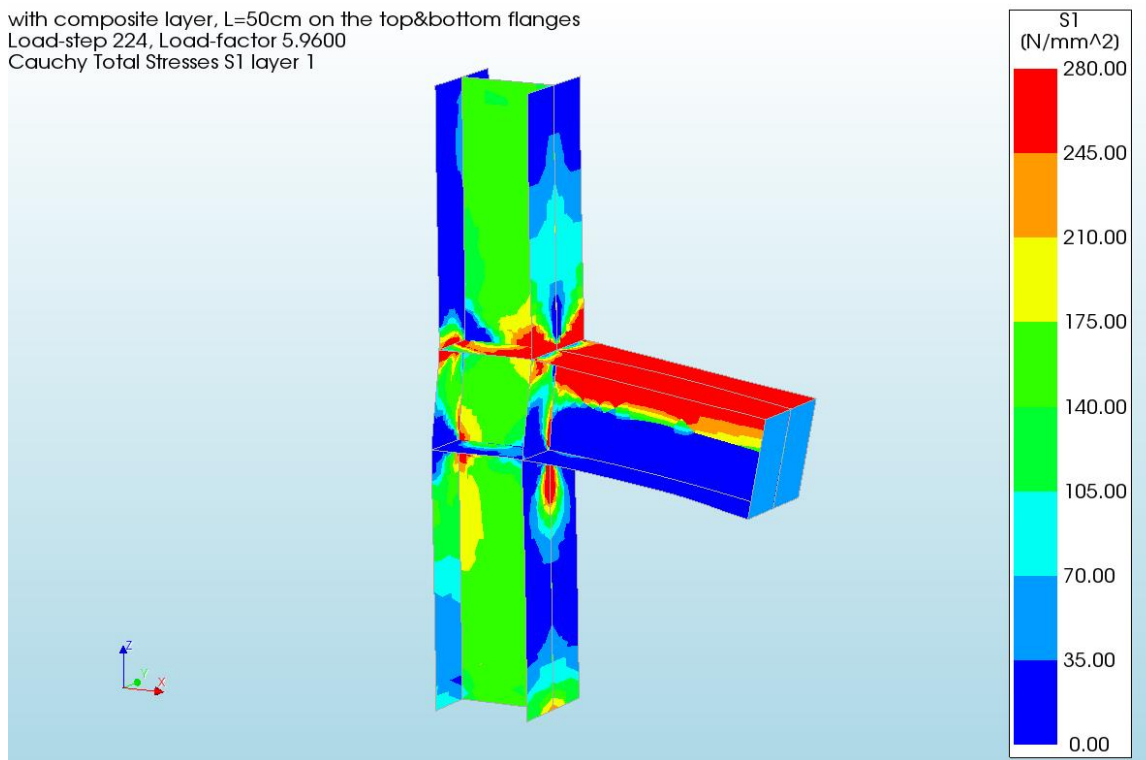
Only 5 mm in the section joint and strengthened flanges



Steel connections earthquake loading

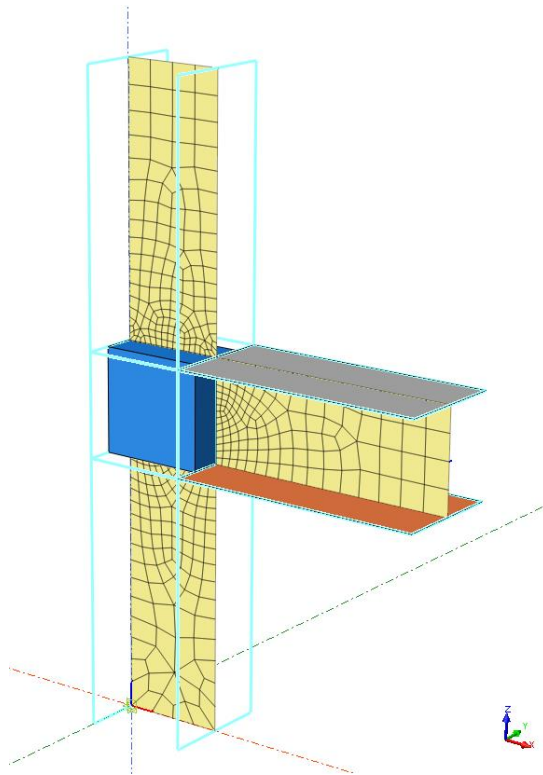
Stress in the steel sections

with composite layer, L=50cm on the top&bottom flanges
Load-step 224, Load-factor 5.9600
Cauchy Total Stresses S1 layer 1

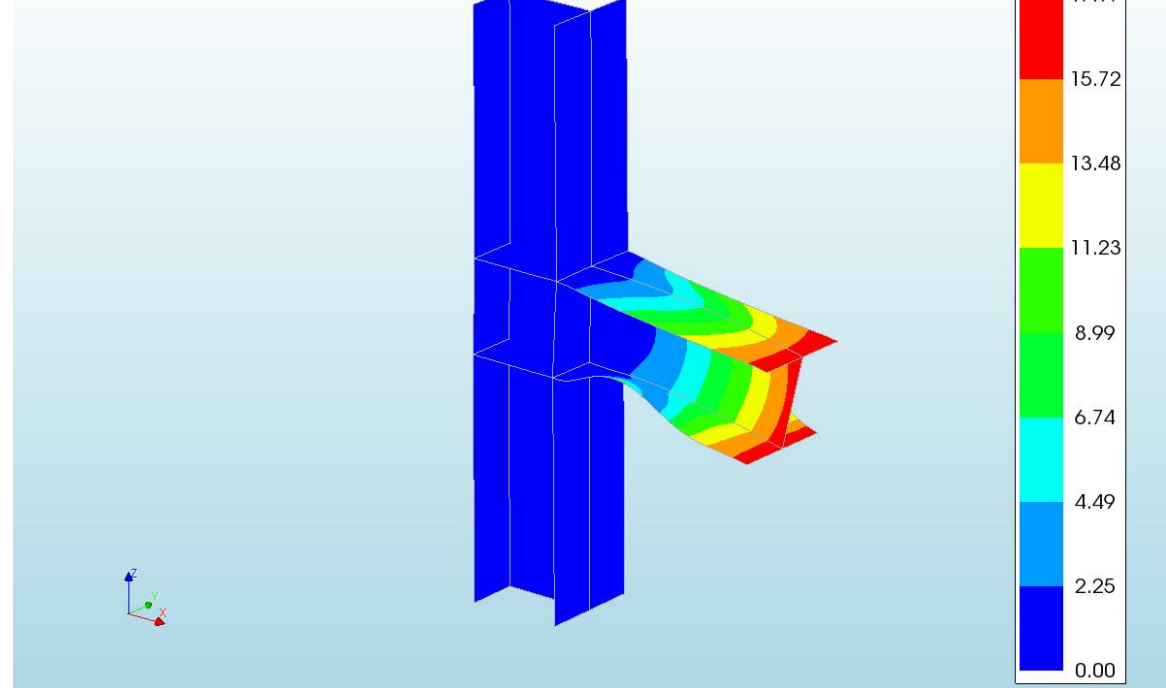


Steel connections earthquake loading

Composite block: $t=50\text{mm}/\text{side}$ at the crossing and $t=5\text{mm}$ for the top&bottom flanges.



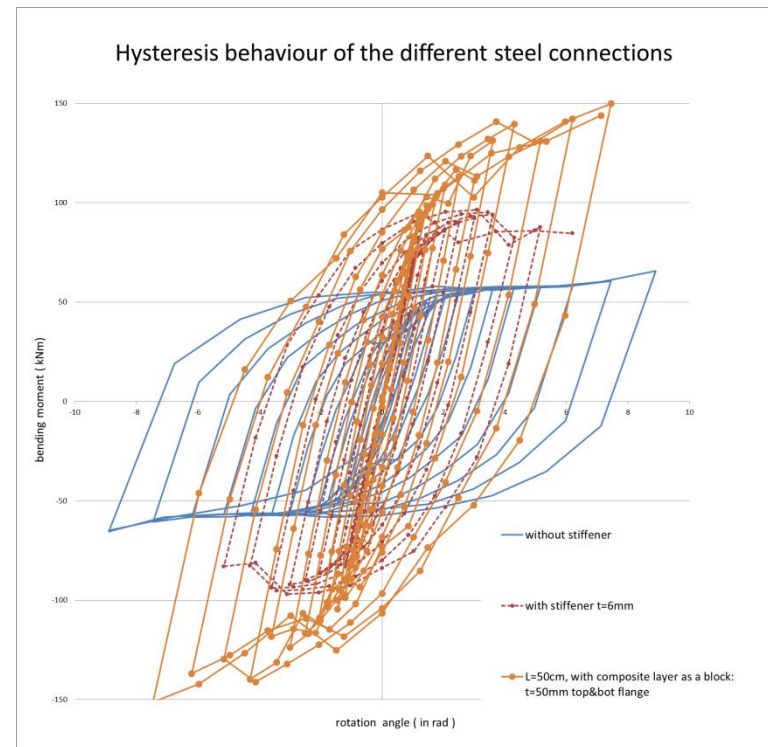
with composite block ($t=50\text{mm}/\text{each side}$), $L=50\text{cm}$ on the top&bottom flanges
Load-step 244, Load-factor 7.1200
Total Displacements TDtXYZ



Steel connections earthquake loading

with 50 mm in the section joint

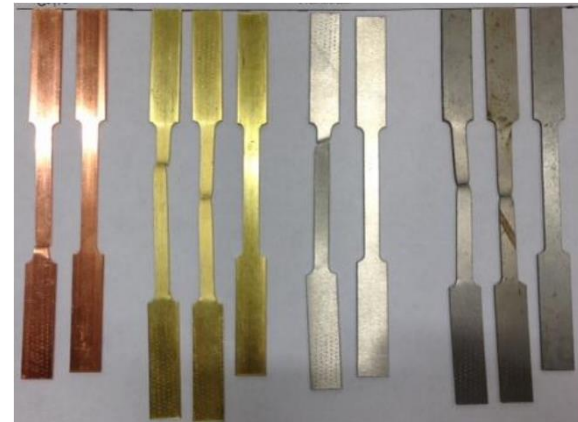
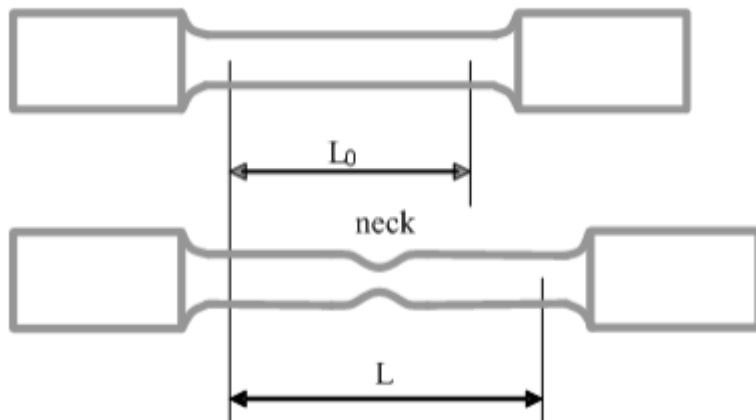
Stronger and only little
bit less ductile in
bending.



Steel localisation

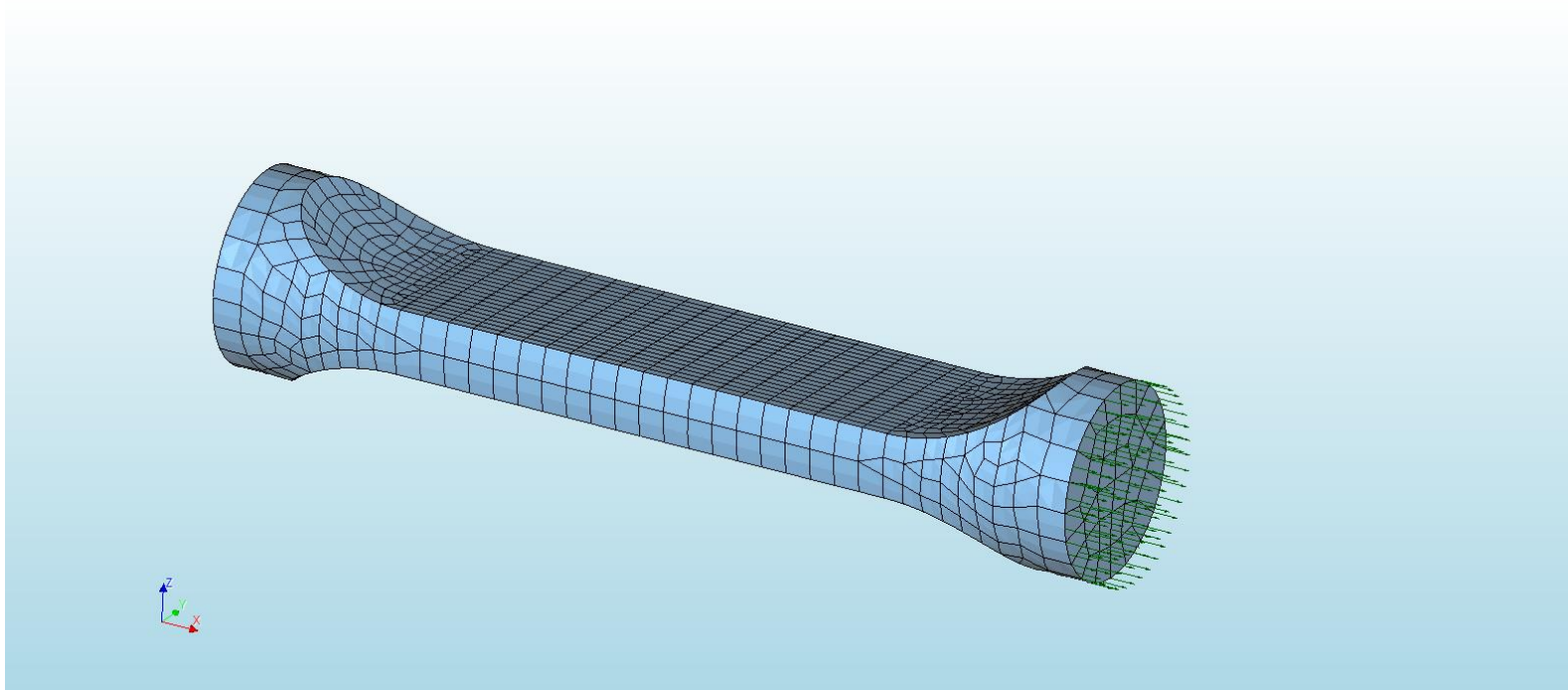
Tensile specimen as example

Rupture somewhere in the reduced area.



Steel localisation

Tensile specimen as example
Input model.



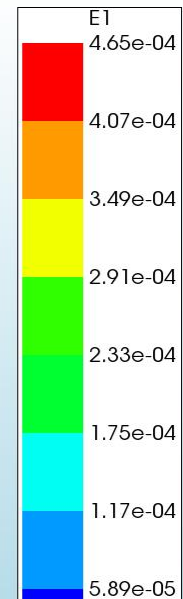
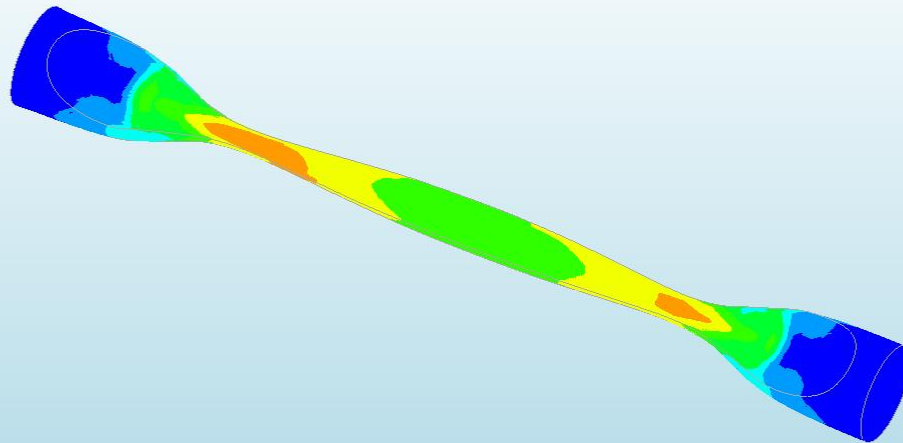
Steel localisation

Tensile specimen as example

Output stress is constant over the length.

Where to plasticize, left, middle or both ??

Analysis3
Load-step 30, Load-factor 0.27789
Total Strains E1



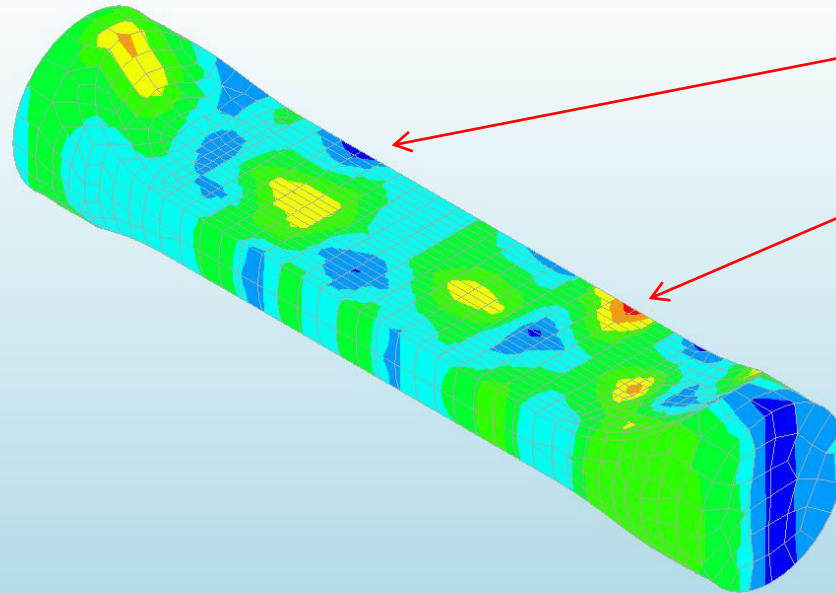
Steel localisation

Tensile specimen as example

Output stress is constant over the length.

Stochastic material over the model.

Analysis3
new load case
Total strain parameters TENSTR



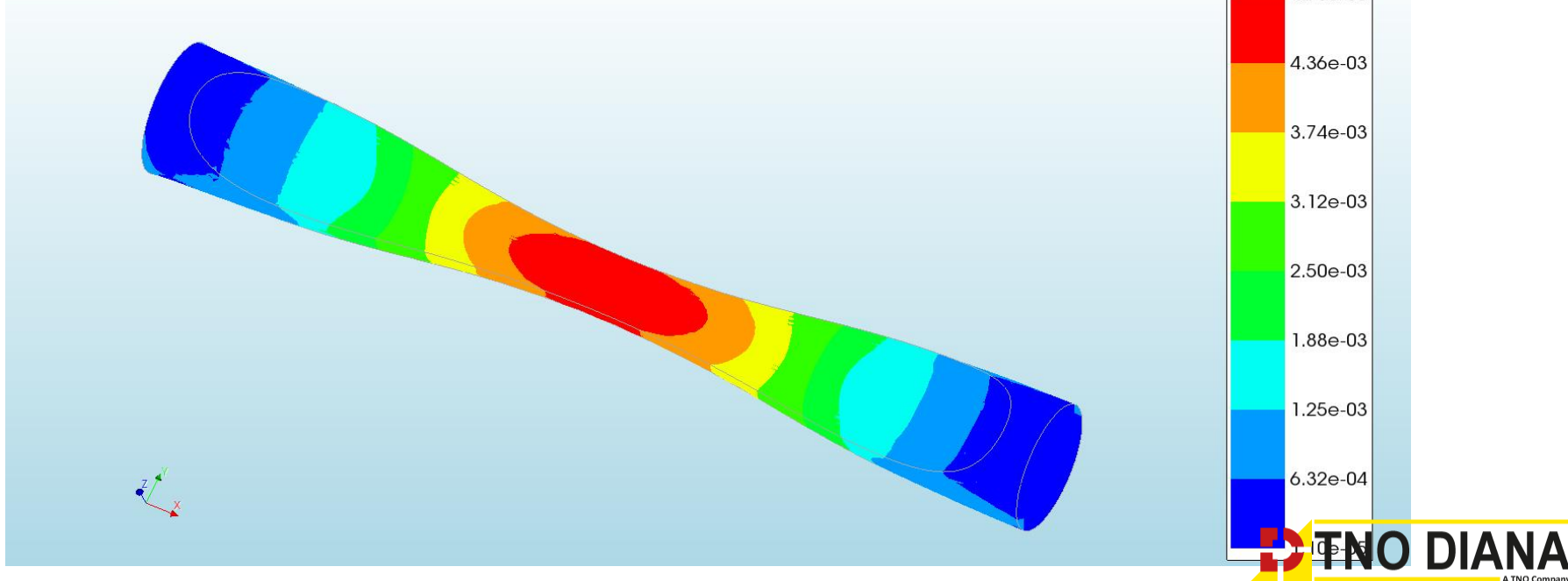
Weaker
Stronger

Steel localisation

Tensile specimen as example

Output plastic strain is introduced in weak spots and finds its way through the section.

Analysis3
Load-step 32, Load-factor 2.9969
Total Strains E1

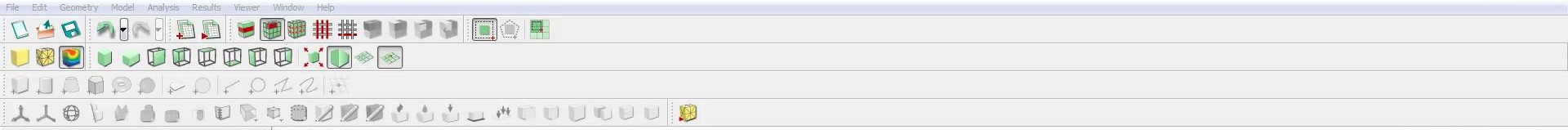


Is modelling difficult?

```
1 newProject( "tensile_test", 1 )
2 setModelAnalysisAspects( [ "STRUCT" ] )
3 setModelDimension( "3D" )
4 setDefaultMeshOrder( "QUADRATIC" )
5 setDefaultMesherType( "HEX_QUAD" )
6 setUnit( "LENGTH", "MM" )
7 setUnit( "FORCE", "N" )
8 #
9 # Setup Geometry
10 createCylinder( "Cylinder 1", [ 0, 0, 0 ], [ 1, 0, 0 ], 15, 140 )
11 createCylinder( "Cylinder 2", [ 30, -50, 35 ], [ 0, 1, 0 ], 30, 100 )
12 #
13 duplicateShape( "Cylinder 2" )
14 translate( "duplicate 3", [ 80, 0, 0 ] )
15 duplicateShape( "Cylinder 2" )
16 duplicateShape( "duplicate 3" )
17 createBlock( "Block 6", [ 30, -50, 5 ], [ 80, 100, 60 ] )
18 duplicateShape( "Block 6" )
19 #
20 translate( "duplicate 4", [ 0, 0, -70 ] )
21 translate( "duplicate 5", [ 0, 0, -70 ] )
22 translate( "duplicate 7", [ 0, 0, -70 ] )
23 #
24 subtract( "Cylinder 1", [ "Cylinder 2", "Block 6", "duplicate 3", "duplicate 4", "duplicate 7", "duplicate 5" ],
25 perspectiveProjection( True )
```

Is modelling difficult?

The screenshot displays the Diana Interactive Environment software interface. The main window shows a 3D wireframe model of a curved pipe. The interface includes a menu bar (File, Edit, Geometry, Model, Analysis, Results, Viewer, Window, Help), a toolbar with various icons, and a Results panel on the left. The Results panel shows the current analysis is 'Analysis1', Case is empty, and Layer is '1 of 1'. Below the Results panel is a 'result view setting' section and a 'Properties' panel with tabs for Model, Mesh, Analysis, and Results. The Properties panel currently shows 'No properties'. At the bottom, there is a Command console and a Messages window. The Command console displays a traceback error: 'NameError: name 'analysis1' is not defined'. The Messages window shows the following text: 'SOLVING SYSTEM OF EQUATIONS', 'PARALLEL DIRECT SPARSE SOLVER', 'Cancelled (runSolver)', 'Could not run solver for analysis 'Analysis1': execution of C:\Program Files\Diana Dev\binseg\ap\nl41.exe failed.', 'Result case 'Analysis1' is not valid', and 'Error executing 'C:/Users/bsa/Documents/tensile_test/tensile_test_nonlin.py', see console window for details'.



Results

Analysis1
Load-step 1, Load-factor 0.10000
Total Displacements TDtX

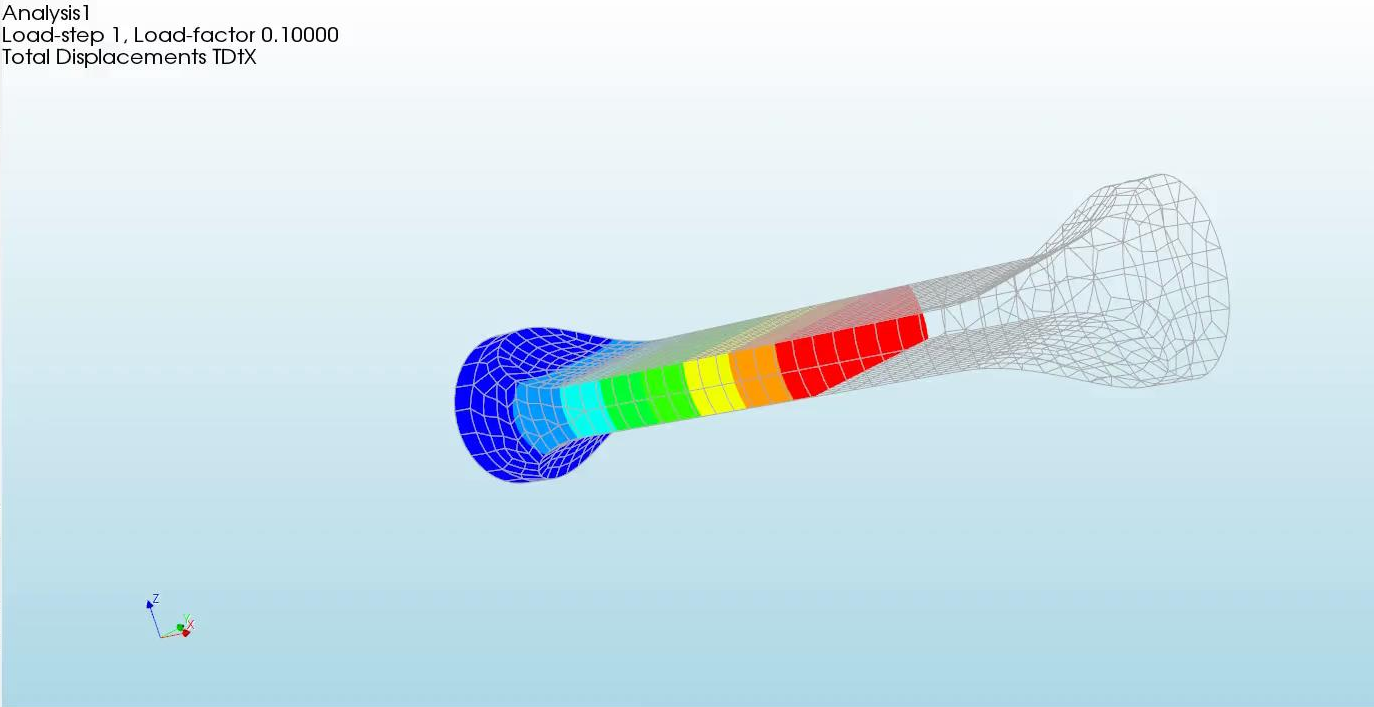
Analysis: Analysis1
Case: Load-step 1, Load-factor 0.10000
Layer: 1 of 1

Output

- Nodal results
 - Total Displacements
 - TDtX
 - TDtY
 - TDtZ
 - TDtXYZ
- Element results

result view setting

Model Mesh Analysis Results



Command console

```
> Traceback (most recent call last):  
File "C:/Users/bsa/Documents/tensile_test/tensile_test_nonlin.py", line 109, in <module>  
[ analysis],  
NameError: name 'analysis1' is not defined
```

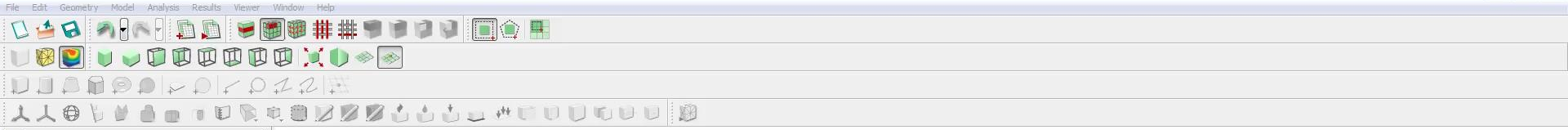
No properties

Messages

```
STEP 30 : DISPLACEMENT NORM = 7.490E-01 TOLERANCE = 1.000E-02  
STEP 30 : FORCE NORM = 1.143E+03 TOLERANCE = 1.000E-02  
RELATIVE OUT OF BALANCE FORCE = 3.300E-13 CHECK = TRUE  
STEP 30 TERMINATED, CONVERGENCE AFTER 0 ITERATIONS  
TOTAL LOAD FACTOR: LOADING( 1) * 2.779E-01
```

/DIANA/DC/END 12:07:36 601.37-CPU 57.89-IO STOP

Error executing 'C:/Users/bsa/Documents/tensile_test/tensile_test_nonlin.py', see console window for details



Results

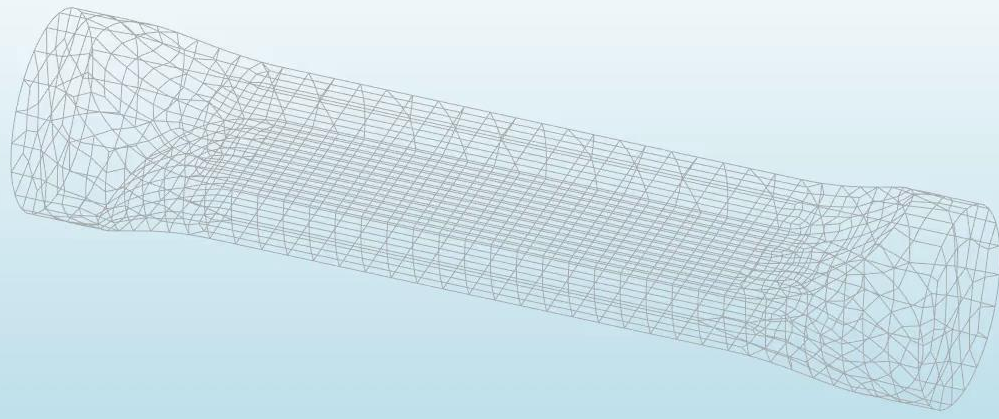
Analysis: Analysis2
Case: new load case
Layer: 1 of 1

Output linear static analysis
Element results

result view setting

Model Mesh Analysis Results

Analysis2
new load case



Properties

No properties

Command console

>

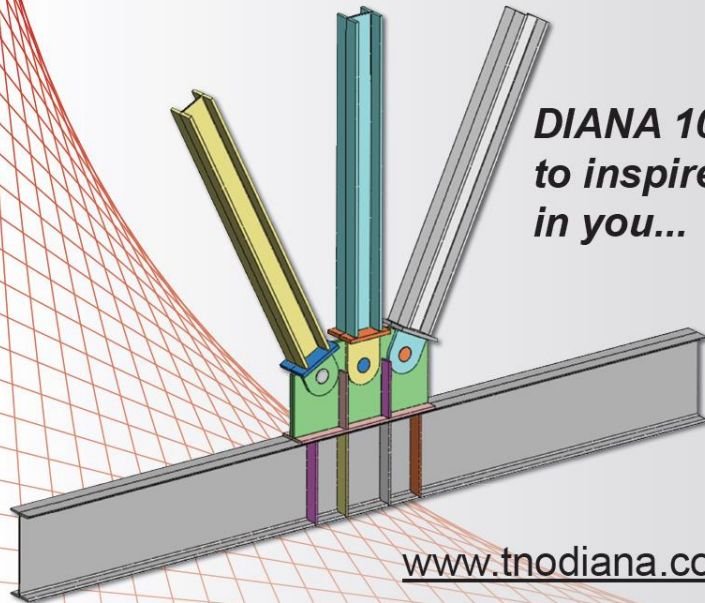
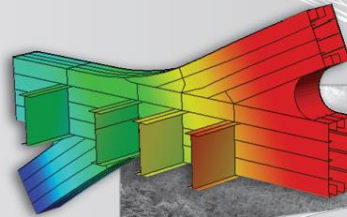
Messages

```
SOLVING SYSTEM OF EQUATIONS  
PARALLEL DIRECT SPARSE SOLVER  
  
RELATIVE DISPLACEMENT VARIATION = 6.716E-02 CHECK = FALSE  
RELATIVE OUT OF BALANCE FORCE = 9.167E-01 CHECK = FALSE  
STEP 1 TERMINATED, NO CONVERGENCE AFTER 10 ITERATIONS  
TOTAL LOAD FACTOR: LOADING( 1) * 9.727E-02  
  
/DIANA/AP/NL41 12:50:20 190.90-CPU 14.84-IO STOP
```

See you at our stand ?

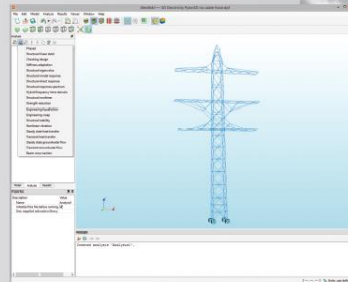


Create your model. Set your parameters in your own script... simply more productive!



DIANA 10 - Finite Elements to inspire the Civil Engineer in you...

www.tnodiana.com





WWW.TNODIANA.COM

TNO DIANA BV

Delftechpark 19a
2628 XJ Delft
The Netherlands
T +31 (0) 88 34262 00
F +31 (0) 88 34262 99

TNO DIANA BV
(Engineering Division)
Engineering (TD/E)
Vlamoven 34
6826 TN Arnhem
The Netherlands
T +31 (0) 88 34262 00

TNO DIANA UK Ltd

Ground Floor
Building 1000
Lakeside North Harbour
Western Road
Portsmouth PO6 3EZ
United Kingdom
T +44 (0) 23 9270 4175
F +44 (0) 23 9270 4001

**TNO DIANA North
America Inc.**

3871 Seven Mile Road
Suite 260
MI 48152
Livonia
T +1 (0) 877 70 34262