



Tutorial
API 2A RP/ISO 19902/Norsok N004

ANSYS[®]

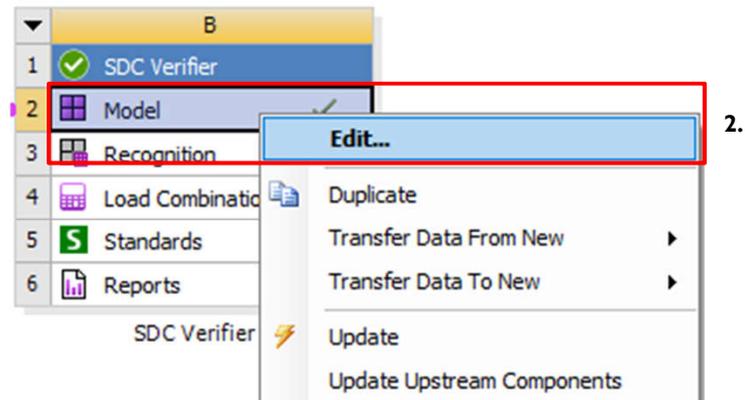
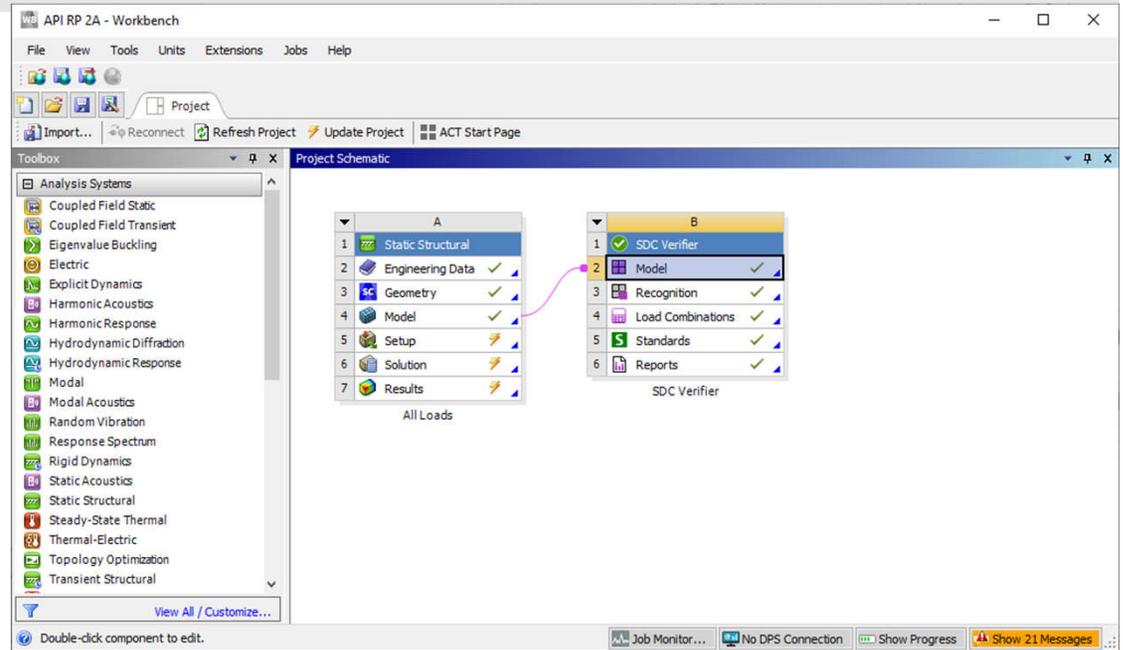
8 Dec 2020
version 2020.0.2

- ▶ In this tutorial, API 2A RP/ISO 19902/Norsok N004 Beam Design Checks are reviewed.
- ▶ A beam model of a steel frame has been used as a start FEM model.
- ▶ Beam member finder was used to recognize beam member length in 3 directions.
- ▶ The report was generated with the help of report designer.

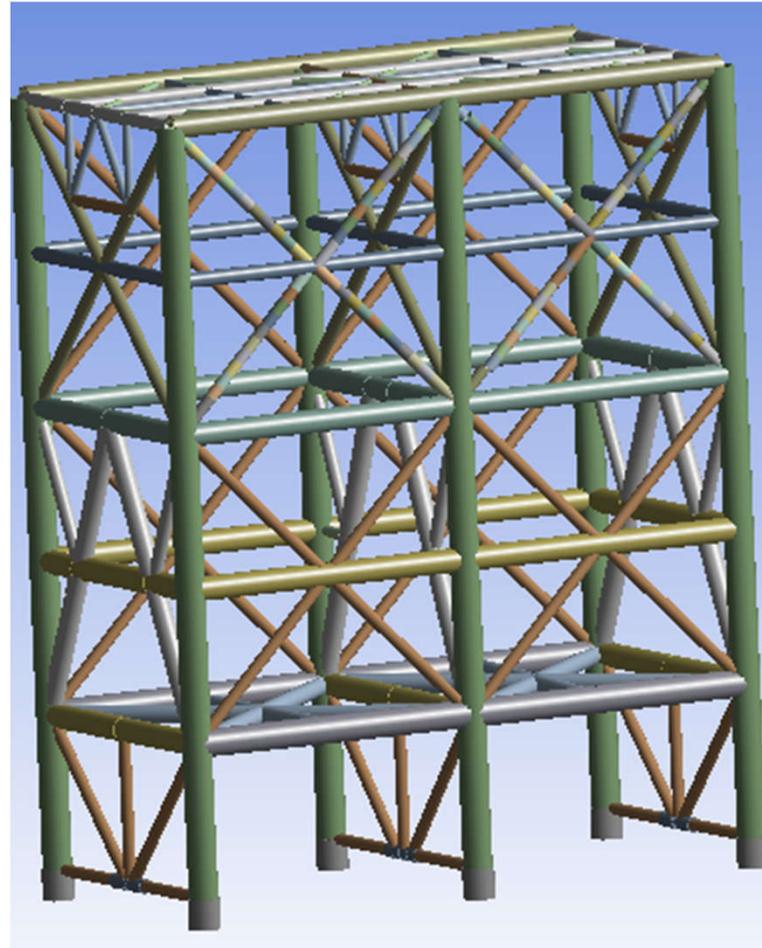
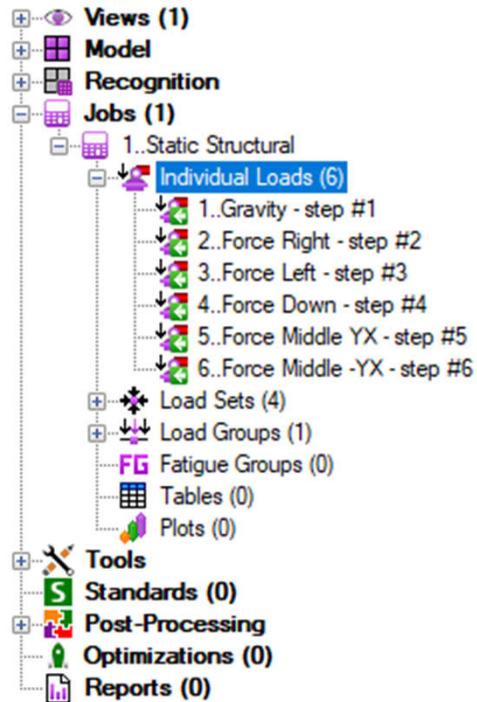
Launch SDC Verifier



- 1 Open in Ansys Workbench **WB**
API RP 2A.wbpj
- 2 Double Click on **Model** or in context menu click **Edit**



Predefined project

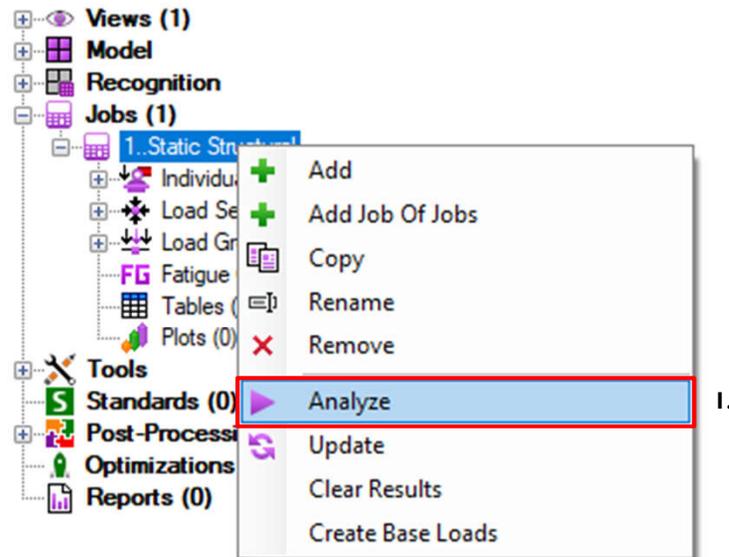


This tutorial uses project with predefined boundary condition, load combinations and load group. The model contains only circular tubes elements.

Analyze Job

1

Execute ► **Analyze** from *Static Structural* context menu



Explanation of Joints

Joint – location where different beam members connect. They are used to recognize beam member length by Beam Member Finder Tool.

There are 6 types of Joints:

1D Joint – 2 beam members that lie on the curve but with different properties;

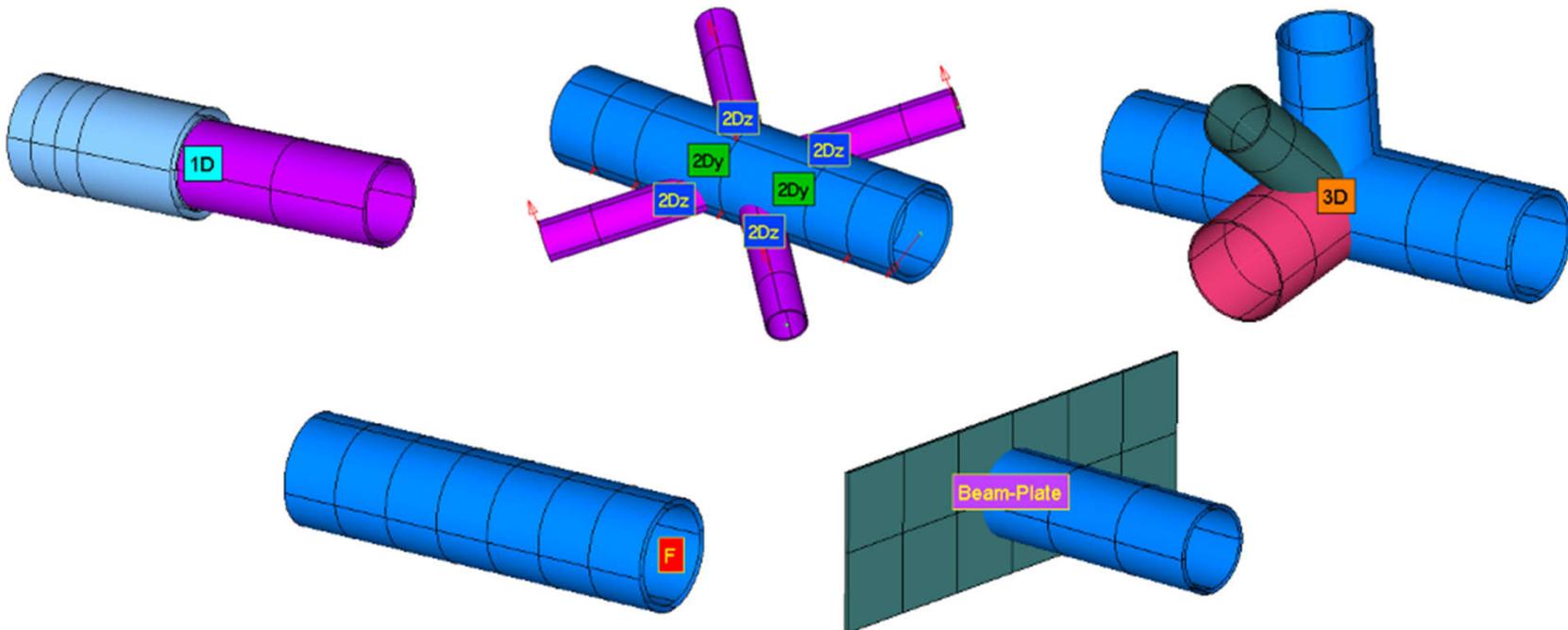
2D Joint – beam members connected in one plane;

3D Joint – beam members connected in space;

Free Joint – node which belongs only to one element (free);

Beam-Plate Joint – beam member connected to plates (perpendicularly);

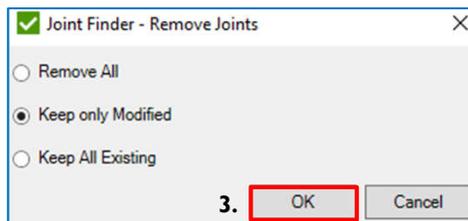
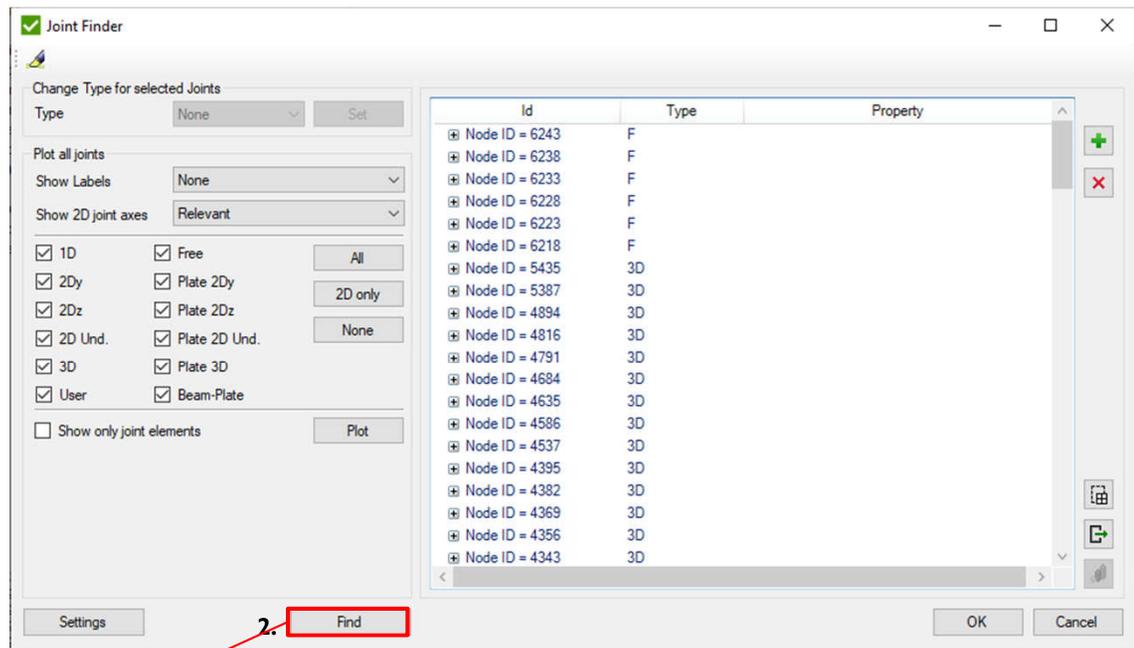
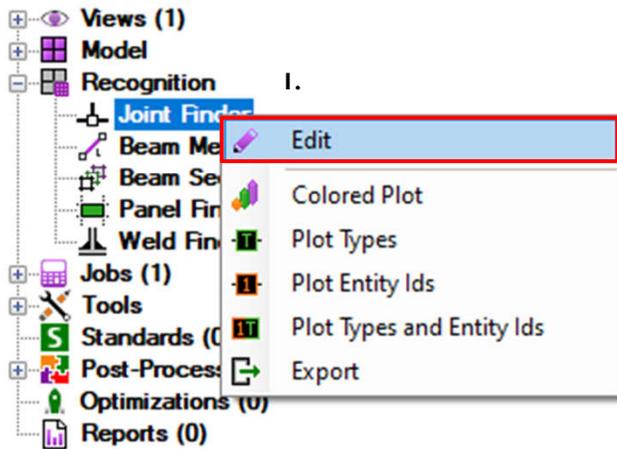
User Defined;



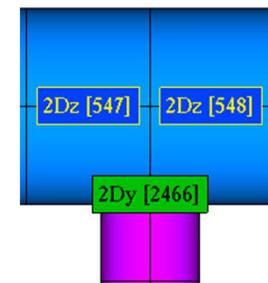
Joint Recognition

- 1 Execute *Edit* from *Joints* context menu
- 2 Press *Find*.
- 3 Press *Ok*.

When performing the joint recognition there are 3 options for existing joints. Default option: Keep only modified– remove all joints except edited by user. Keep all existing options should be used when additional elements were added to the model.

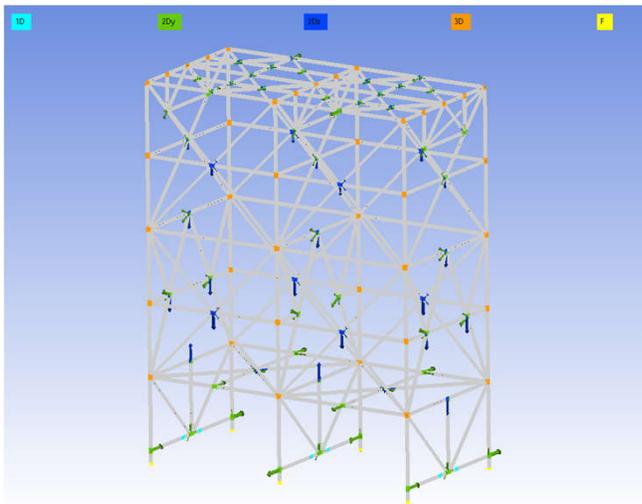


Node ID = 719			
Element ID = 547	2Dz	8..400x19	
Element ID = 548	2Dz	8..400x19	
Element ID = 2466	2Dy	14..200x10	



Joints Plot

- 1 Select All Joints (Ctrl+A).
- 2 Press
- 3 Press Plot Joint Type Labels
- 4 Press OK



Joint Finder

Change Type for selected Joints
 Type: None [Set]

Plot all joints
 Show Labels: None [v]
 Show 2D joint axes: Relevant [v]

1D Free [All]
 2Dy Plate 2Dy [2D only]
 2Dz Plate 2Dz [None]
 2D Und. Plate 2D Und.
 3D Plate 3D
 User Beam-Plate

Show only joint elements [Plot]

Id	Type	Property
Node ID = 1		
Node ID = 2	3D	
Node ID = 27	3D	
Node ID = 52		
Node ID = 53	3D	
Node ID = 78	3D	
Node ID = 103	3D	
Node ID = 104	2Dz	
Node ID = 129	3D	
Node ID = 154		
Node ID = 167		
Node ID = 192	3D	
Node ID = 205	2Dz	
Node ID = 230	3D	
Node ID = 255	2Dz	
Node ID = 280	3D	
Node ID = 305		
Node ID = 318		
Node ID = 343	3D	
Node ID = 356	2Dz	

[Settings] [Find] [OK] [Cancel]

Plot Joints of specific type:

Plot all joints
 Show Labels: None [v]
 Show 2D joint axes: Relevant [v]

1D Free [All]
 2Dy Plate 2Dy [2D only]
 2Dz Plate 2Dz [None]
 2D Und. Plate 2D Und.
 3D Plate 3D
 User Beam-Plate

Show only joint elements [Plot]

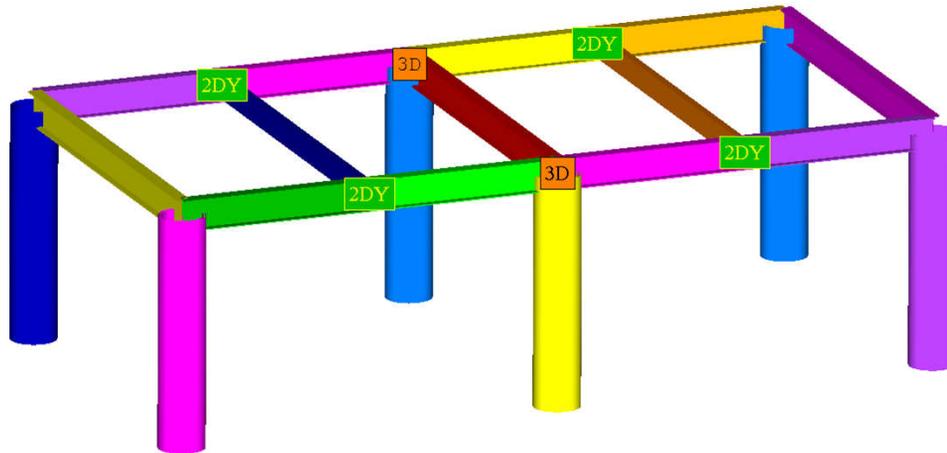
3. Plot Joint Type Labels
- Plot Joint Type in colors

Modify Joint Type:

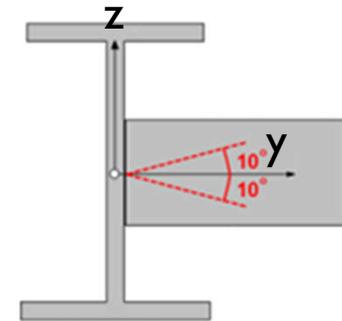
Change Type for selected Joints
 Type: None [v] [Set]

Beam Member Lengths in 2 directions

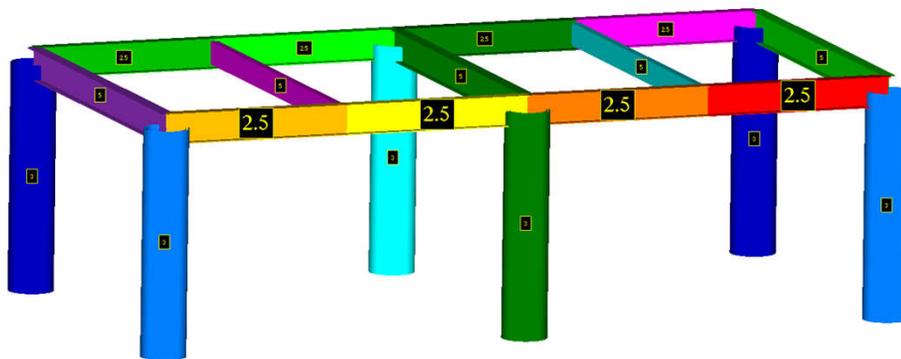
Beam Member Finder recognizes beam members and (buckling) lengths for different directions (Y, Z and Torsional).



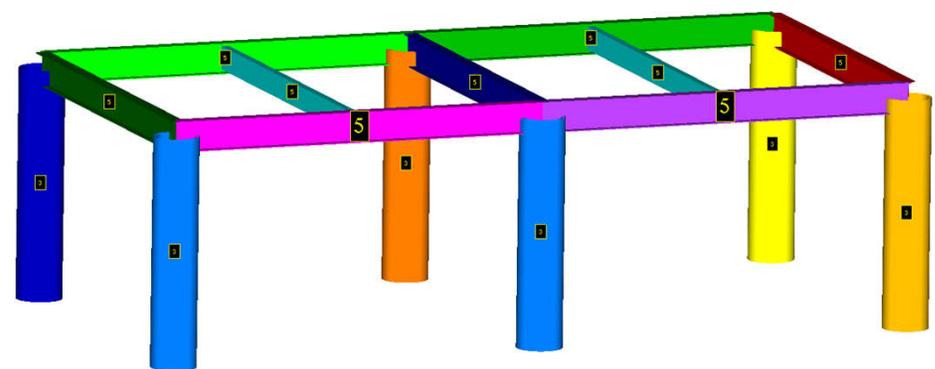
2DY
Joint



Length Y – 4 Beam Members with $L = 2.5$



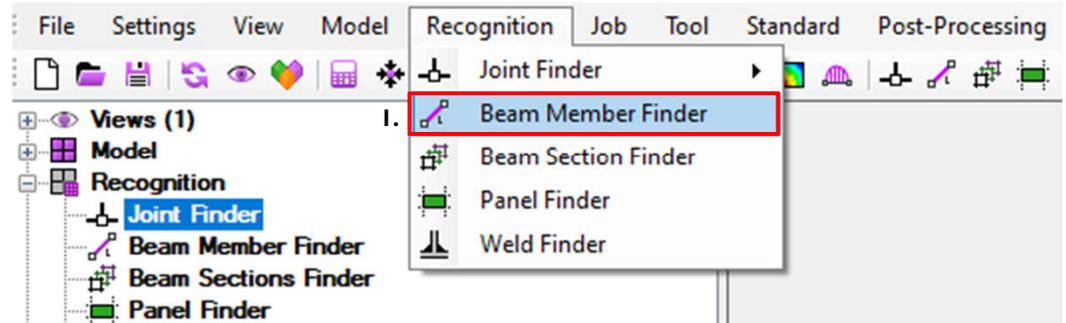
Length Z – 2 Beam Members with $L = 5$



Recognize Length

1 Execute *Recognition – Beam Member Finder*

2 Press *Find*.



Beam Member Finder

Length Y | Length Z | Length Torsional | Edit Member (in all directions) | Filter

Break Joint Options

Length Y:

1D User

2DY Plate 2DY

2DZ Plate 2DZ

2D Und. Plate 2D Und.

3D Plate 3D

Beam-Plate

Set K=0.01 for members fixed by plates

Update Selected Members

Set to Selected

Length [m]: Set

Length Factor: Set

Cm Type: A Set

A = 0.85
B = Calculate
used in API 2A, AISC89, ISO19902
and Norsok N004 standards

ID	Title	Elements	Length [m]	Length Factor	Cm Type	Modified	Joint - NodeID [Joint Type]
1	Beam Member 1 (Y)	50	5				1[3D]
2	Beam Member 2 (Y)	50	5				52[3D]
3	Beam Member 3 (Y)	100	10				458[3D] 405[3D] 507[3D]
4	Beam Member 4 (Y)	100	10				556[3D] 445[3D] 605[3D]
5	Beam Member 5 (Y)	100	10				103[3D] 356[2Dz] 104[2Dz]
6	Beam Member 6 (Y)	51	5				154[3D] 167[3D]
7	Beam Member 7 (Y)	100	10				205[2Dz] 230[3D] 255[2Dz]
8	Beam Member 8 (Y)	51	5				305[3D] 318[3D]
9	Beam Member 9 (Y)	28	2.795	1	A		
10	Beam Member 10 (Y)	56	5.59				318[3D]
11	Beam Member 11 (Y)	56	5.59				305[3D]
12	Beam Member 12 (Y)	50	5				3368[3D]
13	Beam Member 13 (Y)	103	10				1176[3D]
14	Beam Member 14 (Y)	50	5				3518[3D]
15	Beam Member 15 (Y)	103	10				804[3D]
16	Beam Member 16 (Y)	50	5				3668[3D]
17	Beam Member 17 (Y)	28	2.795	1	A		
18	Beam Member 18 (Y)	56	5.59				3825[3D]
19	Beam Member 19 (Y)	56	5.59				3880[3D]
20	Beam Member 20 (Y)	13	1.25	1	A		
21	Beam Member 21 (Y)	28	2.795	1	A		
22	Beam Member 22 (Y)	56	5.59				154[3D]
23	Beam Member 23 (Y)	56	5.59				167[3D]
24	Beam Member 24 (Y)	100	10				4537[3D] 1151[3D] 4586[3D]
25	Beam Member 25 (Y)	100	10				4635[3D] 779[3D] 4684[3D]
26	Beam Member 26 (Y)	13	1.25	1	A		

Settings Find OK Cancel

Beam Member Finder interface

Break Options define what joints are used to split beam members

Change Length/Length Factor for selected beam members

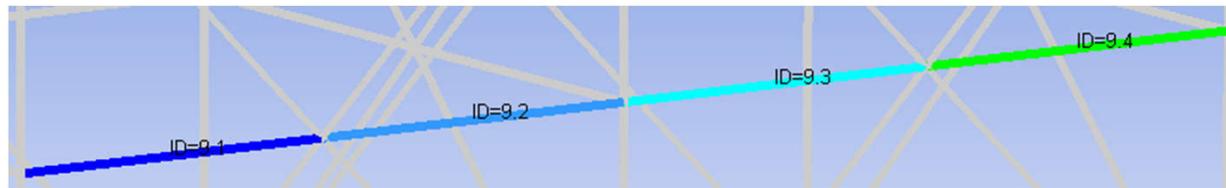
Cm Type is used in API 2A, AISC89, ISO 19902 and Norsok N004 standards

Colored Plot of members with labels (ID, Length, Factor or Cm Type).

- Plot selected members
- Plot Members ID labels
- Plot Full Members ID labels
- Plot Length labels
- Plot Cm Type labels
- Plot Length Factor labels
- Plot Joints for Selected Members
- Plot Members Y and Z axes

Beam Member – straight line. If it contains joints it is split on sub members

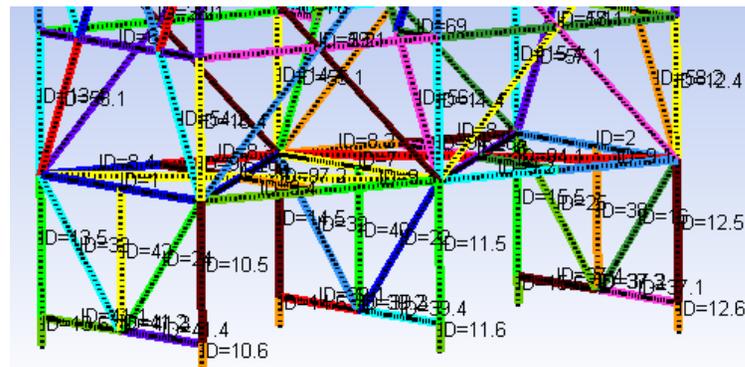
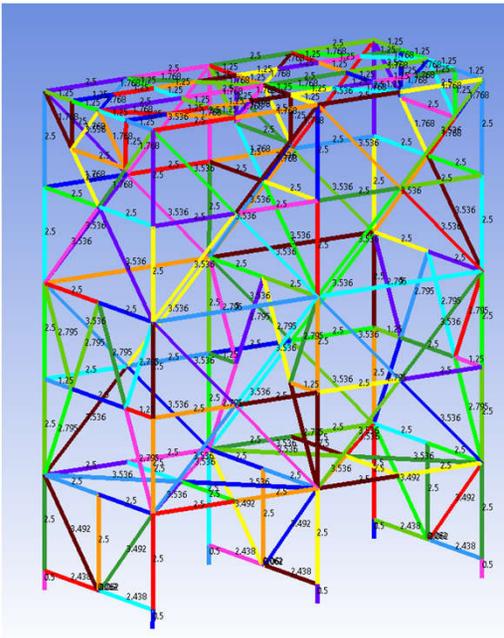
9	Beam Member 9	32	10				1103[2Dz] 201[3D] 2221[2Dz]
9.1	Sub Member 9.1	8	2.5	1	A		
9.2	Sub Member 9.2	8	2.5	1	A		
9.3	Sub Member 9.3	8	2.5	1	A		
9.4	Sub Member 9.4	8	2.5	1	A		



Beam Member's Length Plot

- 1 Select All Beam Members (Ctrl+A)
- 2 Press
- 3 Press Plot Length labels
- 4 Press OK

ID	Title	Elements	Length [m]	Length Factor	Cm Type	Modified	Joint - NodeID [Joint Type]
1	Beam Member 1 (Y)	47	13				923[2Dz] 1107[3D] 155[3D] 1104[3D] 4[3D]
2	Beam Member 2 (Y)	47	13				922[2Dz] 1109[3D] 136[3D] 1108[3D] 117[3D]
3	Beam Member 3 (Y)	32	10				1090[2Dz] 22[3D] 2208[2Dz]
4	Beam Member 4 (Y)	32	10				1100[2Dy] 1117[3D] 2218[2Dy]
5	Beam Member 5 (Y)	18	5				6[3D] 24[3D] 7[3D]
6	Beam Member 6 (Y)	40	10				231[2Dy] 237[2Dy] 243[2Dy] 3[3D] 1361[2Dy] 1...
7	Beam Member 7 (Y)	40	10				251[2Dy] 258[2Dy] 265[2Dy] 5[3D] 1383[2Dy] 1...
8	Beam Member 8 (Y)	18	5				271[2Dz] 237[2Dz] 258[2Dz]
9	Beam Member 9 (Y)	40	10				271[2Dz] 23[3D] 1389[2Dz]
10	Beam Member 10 (Y)	26	7.07				243[2Dz] 23[3D] 1383[2Dz]
11	Beam Member 11 (Y)	13	3.535				231[2Dz]
12	Beam Member 12 (Y)	13	3.535				251[2Dz]
13	Beam Member 13 (Y)	26	7.07				265[2Dz] 23[3D] 1361[2Dz]
14	Beam Member 14 (Y)	45	14.144				1090[2Dz] 1105[3D] 2232[2Dz]
15	Beam Member 15 (Y)	23	7.072				1090[2Dz]
16	Beam Member 16 (Y)	36	10				1105[3D]
17	Beam Member 17 (Y)	32	10				1114[2Dz] 165[3D] 2232[2Dz]
18	Beam Member 18 (Y)	36	10				381[2Dy] 1106[3D] 1500[2Dy]
19	Beam Member 19 (Y)	22	7.072				1114[2Dz]
20	Beam Member 20 (Y)	45	14.144				1114[2Dz] 1105[3D] 2208[2Dz]
21	Beam Member 21 (Y)	5	1.5	1	A		
22	Beam Member 22 (Y)	5	1.5	1	A		
23	Beam Member 23 (Y)	5	1.5	1	A		
24	Beam Member 24 (Y)	5	1.5	1	A		
25	Beam Member 25 (Y)	6	1.803	1	A		



- Plot selected members
- Plot Members ID labels
- Plot Full Members ID labels
- Plot Length labels
- Plot Cm Type labels
- Plot Length Factor labels
- Plot Joints for Selected Members
- Plot Members Y and Z axes

Also it is possible to display beam members IDs by pressing Plot Members ID labels

Cm – reduction factors

Cmy and Cmz reduction factors are used in combined axial and bending check. API RP 2A description:

$$\frac{f_c}{\phi_c F_{cn}} + \frac{1}{\phi_b F_{bn}} \left\{ \left[\frac{C_{my} f_{by}}{\left(1 - \frac{f_c}{\phi_c F_{ey}}\right)} \right]^2 + \left[\frac{C_{mz} f_{bz}}{\left(1 - \frac{f_c}{\phi_c F_{ez}}\right)} \right]^2 \right\}^{0.5} \leq 1.0 \quad \dots\dots (D.3.2-1)$$

**TABLE D.3-1
EFFECTIVE LENGTH AND BENDING
REDUCTION FACTORS FOR
MEMBER STRENGTH CHECKING**

Situation	Effective Length Factor K	Reduction Factor Cm ⁽¹⁾
Superstructure Legs		
Braced	1.0	(a)
Portal (unbraced)	K ⁽²⁾	(a)
Jacket Legs & Piling		
Grouted Composite Section	1.0	(c)
Ungouted Jacket Legs	1.0	(c)
Ungouted Piling Between Shim Points	1.0	(b)
Jacket Braces		
Face-to-face Length of Main Diagonals	0.8	(b) or (c)
Face of Leg to Centerline of Joint Length of K-Braces ⁽³⁾	0.8	(c)
Longer Segment Length of X-Braces ⁽³⁾	0.9	(c)
Secondary Horizontals	0.7	(c)
Deck Truss Chord members	1.0	(a),(b) or (c)
Deck Truss Web Members		
In-Plane Action	0.8	(b)
Out-of-Plane Action	1.0	(a) or (b)

Notes to Table D.3-1:

- (1) Use whichever is more applicable to a specific situation. Values of the reduction factor C_m referred to in the above table are as follows:
 - (a) 0.85
 - (b) 0.6 - 0.4 $\left(\frac{M_1}{M_2}\right)$, no more than 0.85, or less than 0.40, where M₁/M₂ is the ratio of smaller to larger moments at the ends of that portion of the member unbraced in the plane of bending under consideration. M₁/M₂ is positive when the number is bent in reverse curvature, negative when bent in single curvature.
 - (c) 1.0 - 0.4 $\left(\frac{f_c}{\phi_c F_e}\right)$, or 0.85, whichever is less

By default **Cm Type** is equal to A = 0.85 for all members. It is possible to modify Cm Type for selected members:

In SDC Verifier B = minimum from (b) and (c)

Cmy and Cmz factors for ISO 19902 can be found in the Table 13.5-1 and are used in the formula 13.3-3.

Cmy and Cmz factors for Norsok N004 Rev3 can be found in the Table 6-2 and are used in the formula 6.27

API RP 2A-LRFD



API RP*2A-LRFD 93 ■ 0732290 0507613 001 ■

Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms — Load and Resistance Factor Design

API RECOMMENDED PRACTICE 2A-LRFD (RP 2A-LRFD)
FIRST EDITION, JULY 1, 1993

American Petroleum Institute
1220 L Street, Northwest
Washington, DC 20005



This “Recommended Practice for Planning, Designing, and Constructing Fixed Offshore Platforms — Load and Resistance Factor Design” (LRFD) contains the engineering design principles and good practices that have been the basis of the API RP2A working strength design (WSD) recommended practice, now in its 20th Edition. The LRFD provisions have been developed from the WSD provisions using reliability based calibration.

API RP*2A-LRFD 93 ■ 0732290 0507613 T48 ■

Issued by
AMERICAN PETROLEUM INSTITUTE
Production Department

FOR INFORMATION CONCERNING TECHNICAL CONTENTS OF
THIS PUBLICATION CONTACT THE API PRODUCTION DEPARTMENT,
1201 MAIN STREET, SUITE 2535, DALLAS, TX 75202-3994 — (214) 748-3841.
SEE BACK SIDE FOR INFORMATION CONCERNING HOW TO OBTAIN
ADDITIONAL COPIES OF THIS PUBLICATION.

Users of this publication should become familiar with its scope
and content. This publication is intended to supplement rather
than replace individual engineering judgment.

OFFICIAL PUBLICATION

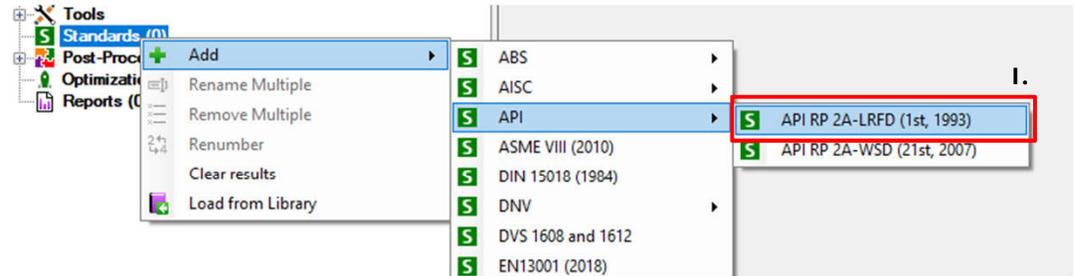


REG. U.S. PATENT OFFICE

Copyright © 1993 American Petroleum Institute

Add API RP 2A-LRFD standard

1 Execute *Standards* => *Add* => *API RP 2A-LRFD (1st, Jul 1993)*.



SECTION D CYLINDRICAL MEMBER DESIGN

C_x = critical elastic buckling coefficient

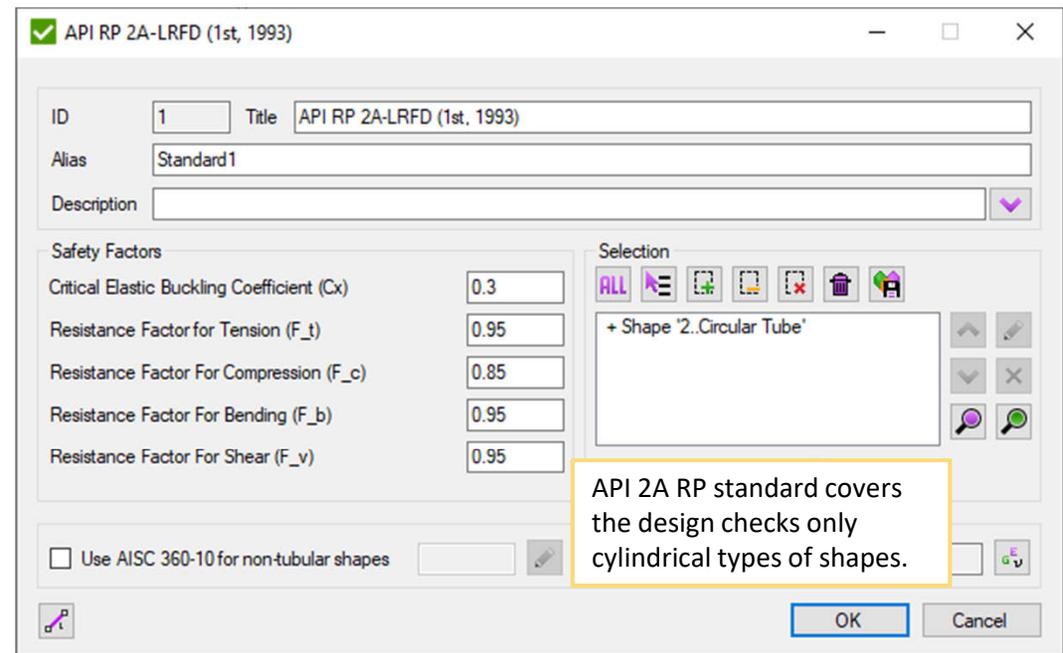
The theoretical value of C_x is 0.6. However, a reduced value of $C_x = 0.3$ is recommended for use in Equation D.2.2-3 to account for the effect of initial geometric imperfections within API Spec 2B tolerance limits, Reference D2.

ϕ_t = resistance factor for axial tensile strength, 0.95

ϕ_c = resistance factor for axial compressive strength, 0.85

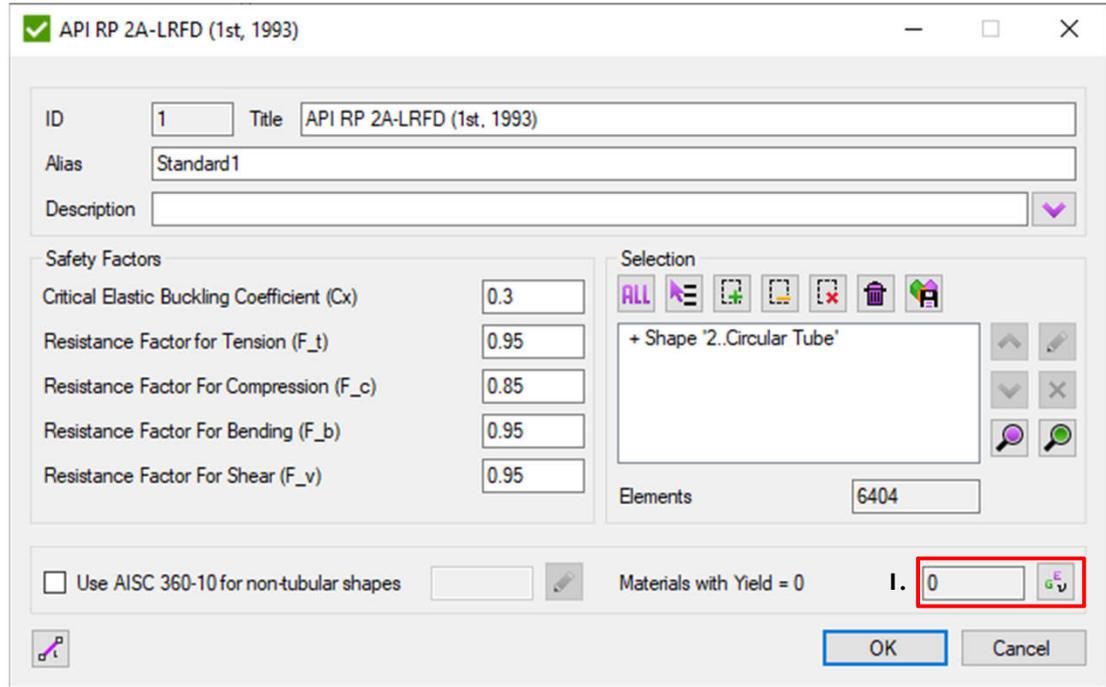
ϕ_b = resistance factor for bending strength, 0.95.

ϕ_v = resistance factor for beam shear strength, 0.95



Define Material Characteristics

- 1 Press  to set the material yield stress and tensile strength
- 2 Tensile Strength: **360e6**
- 3 Yield Stress: **240e6**
- 4 Press *Set*
- 5 Press *OK*



API RP 2A-LRFD (1st, 1993)

ID: 1 Title: API RP 2A-LRFD (1st, 1993)

Alias: Standard1

Description:

Safety Factors

Critical Elastic Buckling Coefficient (Cx): 0.3

Resistance Factor for Tension (F_t): 0.95

Resistance Factor For Compression (F_c): 0.85

Resistance Factor For Bending (F_b): 0.95

Resistance Factor For Shear (F_v): 0.95

Selection

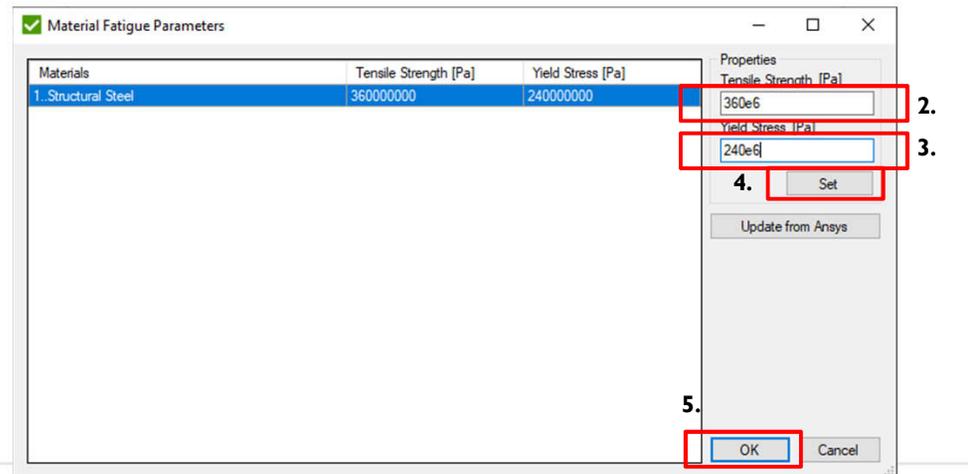
+ Shape '2..Circular Tube'

Elements: 6404

Use AISC 360-10 for non-tubular shapes

Materials with Yield = 0: 0 

OK Cancel



Material Fatigue Parameters

Materials	Tensile Strength [Pa]	Yield Stress [Pa]
1. Structural Steel	360000000	240000000

Properties

Tensile Strength [Pa]: 360e6

Yield Stress [Pa]: 240e6

Set

Update from Ansys

OK Cancel

Extreme table for bending check

1 Execute *Table* from the **Bending Stress Check** context menu

2 Press to select load

3 Load Type: **Load Group**

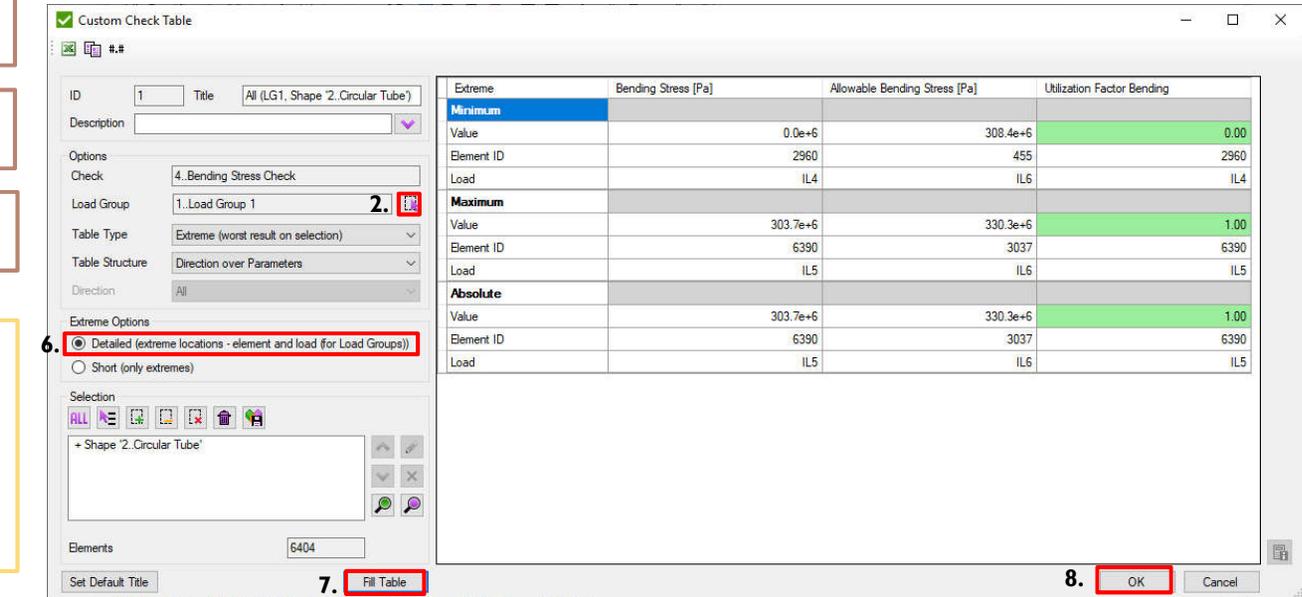
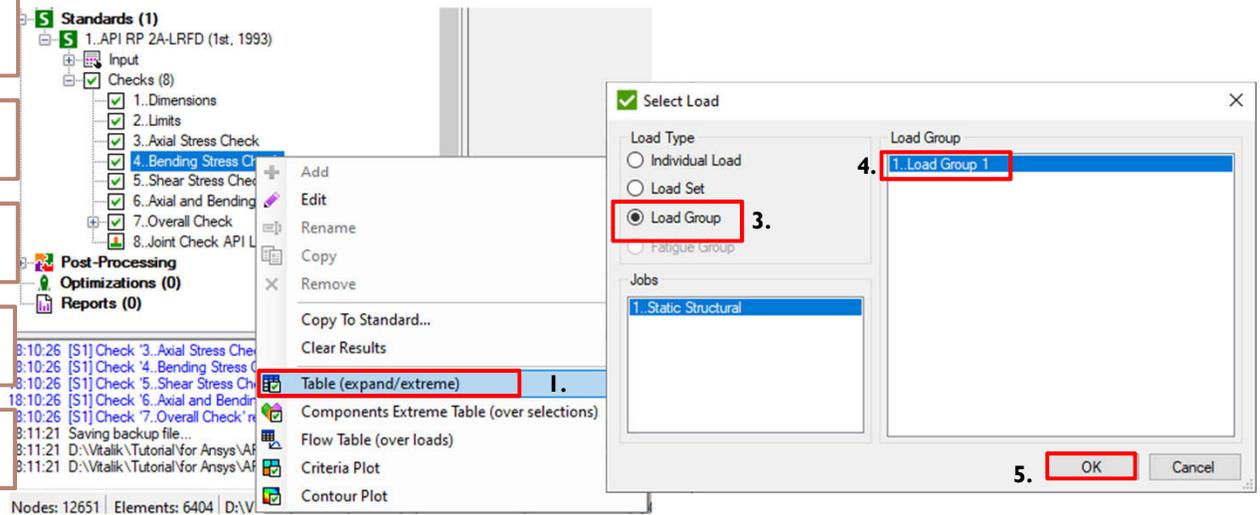
4 Load: **1..Load Group 1**

5 Press *OK*

6 Extreme Options: **Detailed**

7 Press *Fill Table*

8 Press *OK*



Overall Check contains results from all checks. With the help of one table/plot it is possible to verify if the model passes the checks (< 1). Overall Utilization Factor = worst U_f among all checks.

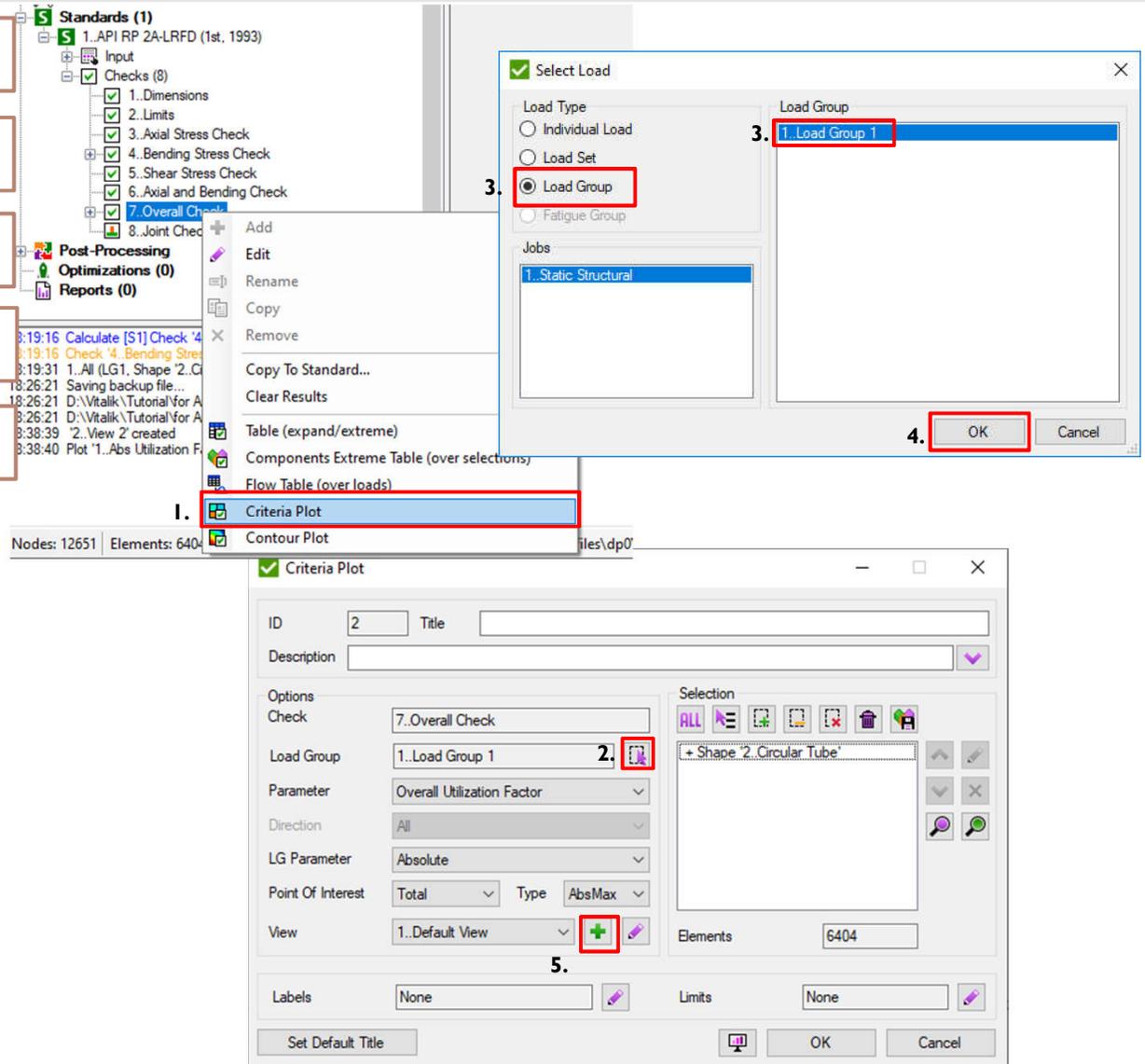
Criteria Plot for Bending Stress Check

- 1 Execute *Criteria Plot* from **Bending Stress Check** context menu
- 2 Press to select load
- 3 Load Type: **Load Group**
- 4 Load: **1..Load Group 1**
- 5 Press **OK**
- 6 Press to add view

The screenshot shows the software interface with several windows and menus. The 'Standards (1)' tree on the left has '4..Bending Stress Check' selected, with a context menu open. The 'Select Load' dialog box is open, showing 'Load Group' selected as the Load Type and '1..Load Group 1' selected in the Load Group list. The 'Criteria Plot' dialog box is also open, showing '4..Bending Stress Check' as the Check, '1..Load Group 1' as the Load Group, and 'Utilization Factor Bending' as the Parameter. The 'View' field is set to '1..Default View' with a '+' icon next to it. The 'Elements' field shows '6404'. The 'OK' button is highlighted.

Criteria plot for overall check

- 1 Execute  from **Overall Check** context menu
- 2 Press  to select load
- 3 Load Type: **Load Group**. Load: **1..Load Group 1**
- 4 Press **OK**
- 5 Press  to add view



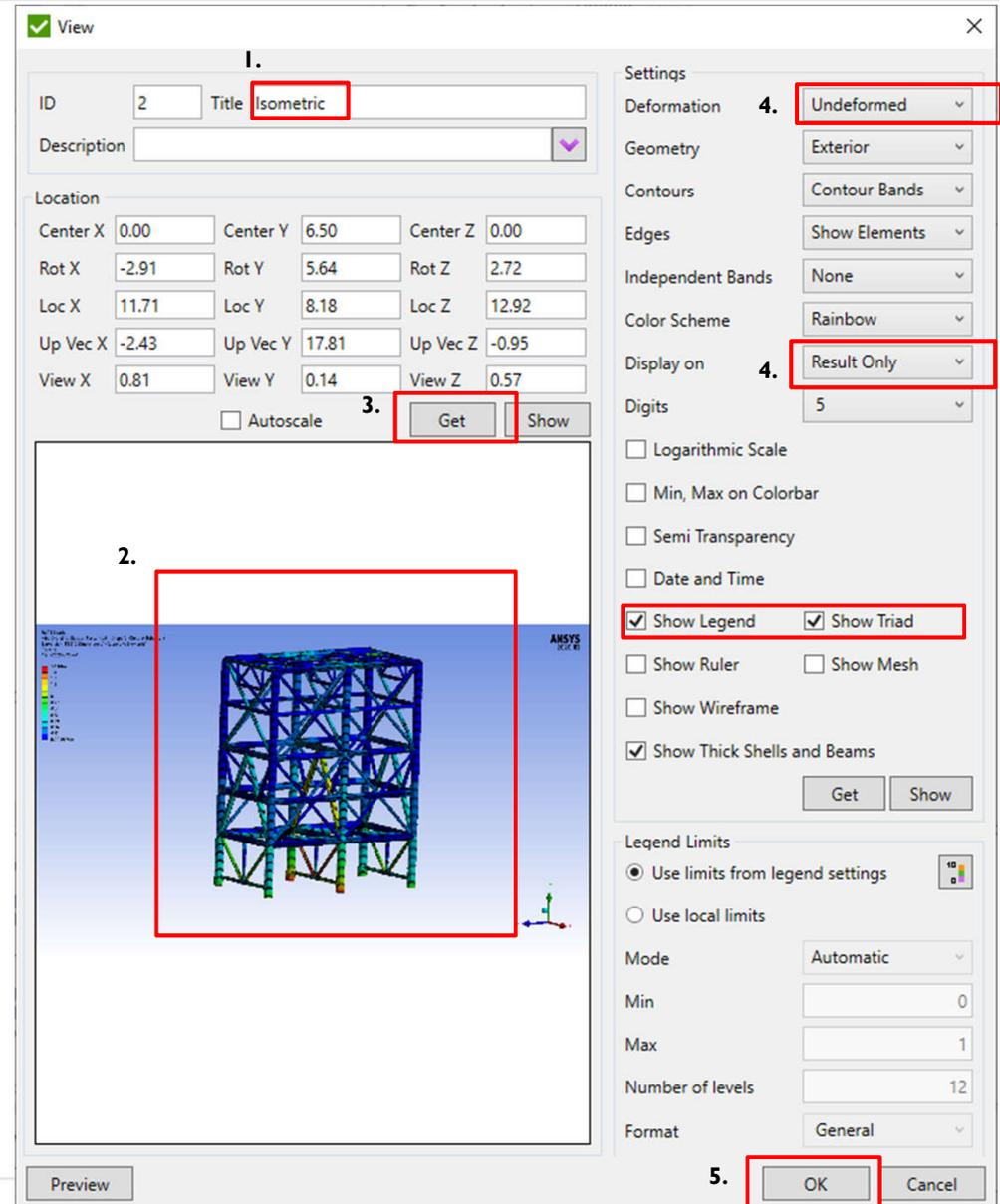
The screenshot illustrates the steps to create a criteria plot for an overall check in SDC Verifier. The background shows the software interface with a tree view of standards and a context menu. The 'Criteria Plot' option is highlighted in the context menu. The 'Select Load' dialog is open, showing 'Load Group' selected as the Load Type and '1..Load Group 1' selected as the Load Group. The 'Criteria Plot' dialog is also open, showing the configuration for the plot. The 'Criteria Plot' dialog has the following settings:

- ID: 2
- Title: (empty)
- Description: (empty)
- Options Check: 7..Overall Check
- Load Group: 1..Load Group 1
- Parameter: Overall Utilization Factor
- Direction: All
- LG Parameter: Absolute
- Point Of Interest: Total, Type: AbsMax
- View: 1..Default View
- Labels: None
- Limits: None

The 'Criteria Plot' dialog also shows a selection of elements: '+ Shape 2..Circular Tube'.

Plot. Create View

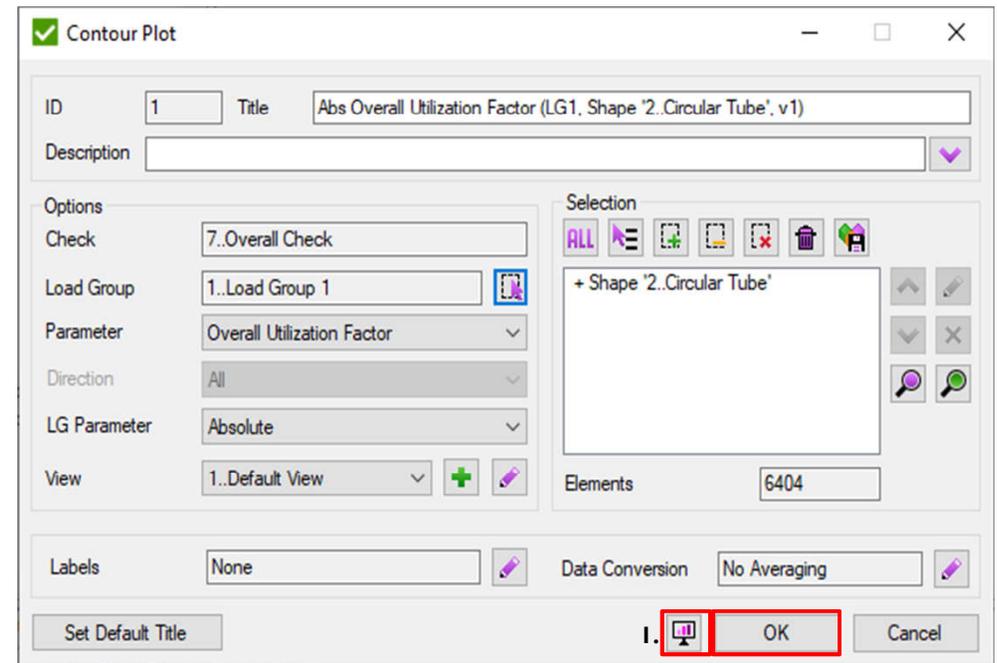
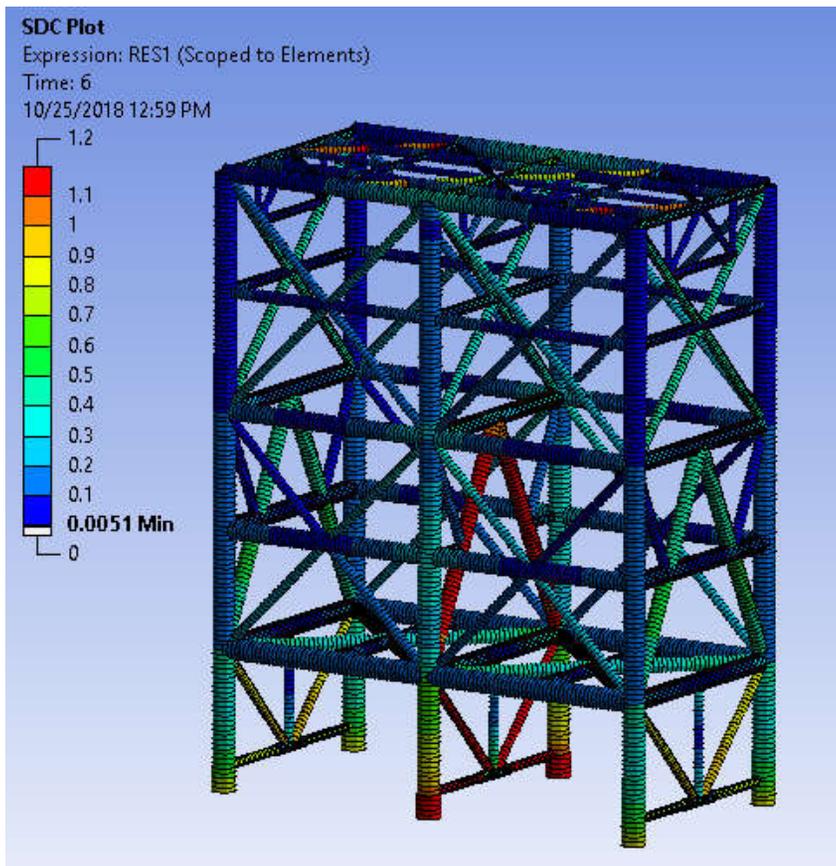
- 1 Title: **Isometric**
- 2 Orient model in Ansys Mechanical as shown on the picture
- 3 Press *Get*
- 4 Deformation: **Undeformed**
Display on: **Result Only**
Show Legend: **On**
Show Triad: **On**
- 5 Press *OK*



Display Plot

1 Press  to display plot

2 Press *OK*



2.

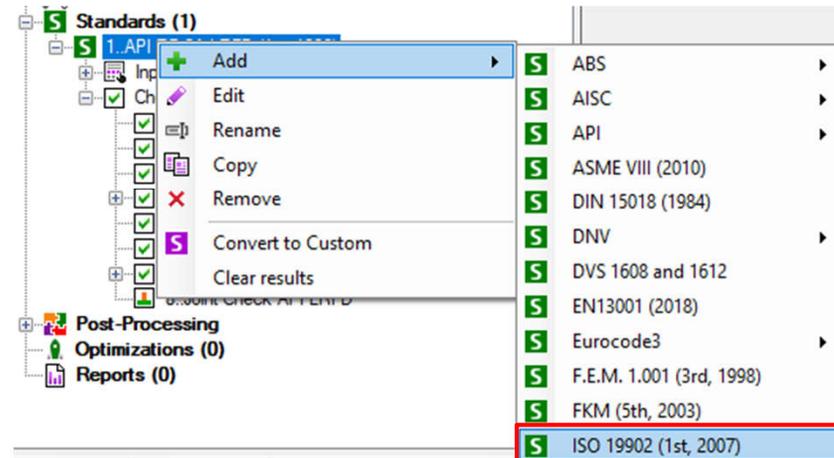
Add ISO 19902 standard

1

Execute *Standards* => *Add* => *ISO 19902* (1st, Dec 2007).

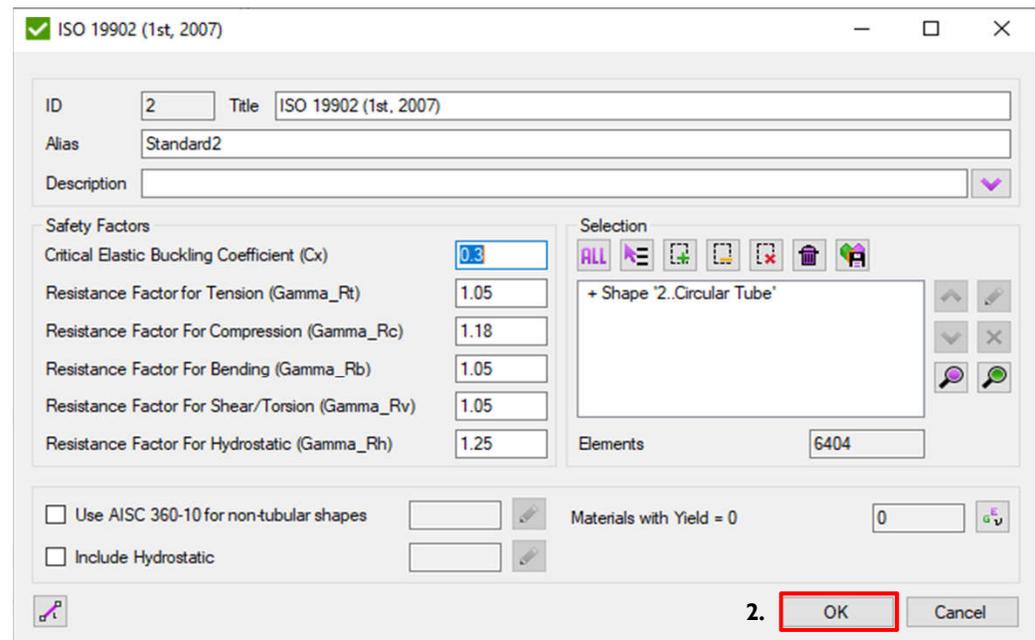
2

Press *OK*



1.

ISO 19902 procedure is similar to API 2A RP. In overall check it is possible to verify if the structure passes all checks : Overall Utilization Factor = worst U_f among all checks < 1.
Add extreme table and criteria plot for '1..Load Group 1' the same as for API standard.



2.

ISO 19902 standard

C_x is the elastic critical buckling coefficient, see below;

The theoretical value of C_x for an ideal tubular is 0,6. However, a reduced value of $C_x = 0,3$ should be used in Equation (13.2-10) to account for the effect of initial geometric imperfections within the tolerance limits given in Clause 21. A reduced value of $C_x = 0,3$ is implicit in the value of f_{xe} used in Equations (13.2-8) and (13.2-9).

$\gamma_{R,t}$ is the partial resistance factor for axial tensile strength, $\gamma_{R,t} = 1,05$.

$\gamma_{R,c}$ is the partial resistance factor for axial compressive strength, $\gamma_{R,c} = 1,18$.

$\gamma_{R,b}$ is the partial resistance factor for bending strength, $\gamma_{R,b} = 1,05$;

$\gamma_{R,v}$ is the partial resistance factor for shear strength, $\gamma_{R,v} = 1,05$;

$\gamma_{R,h}$ is the partial resistance factor for hoop buckling strength, $\gamma_{R,h} = 1,25$.

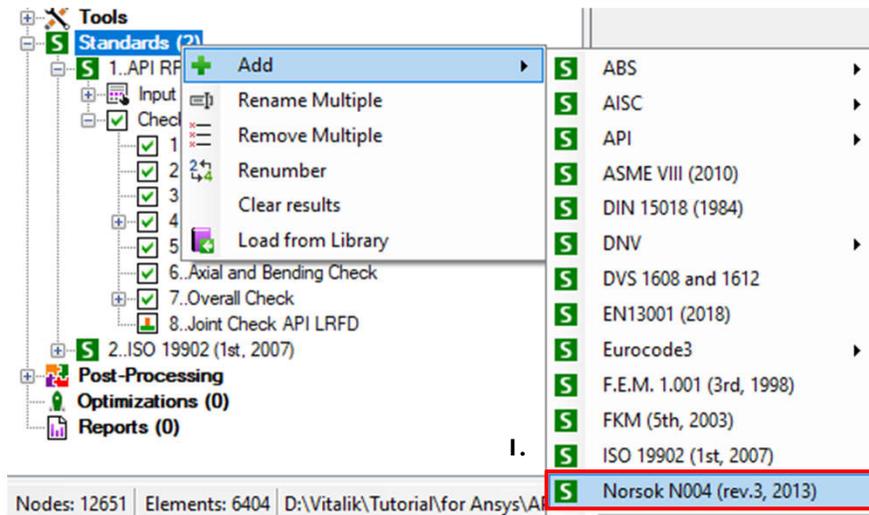
Add Norsok N004 standard

1

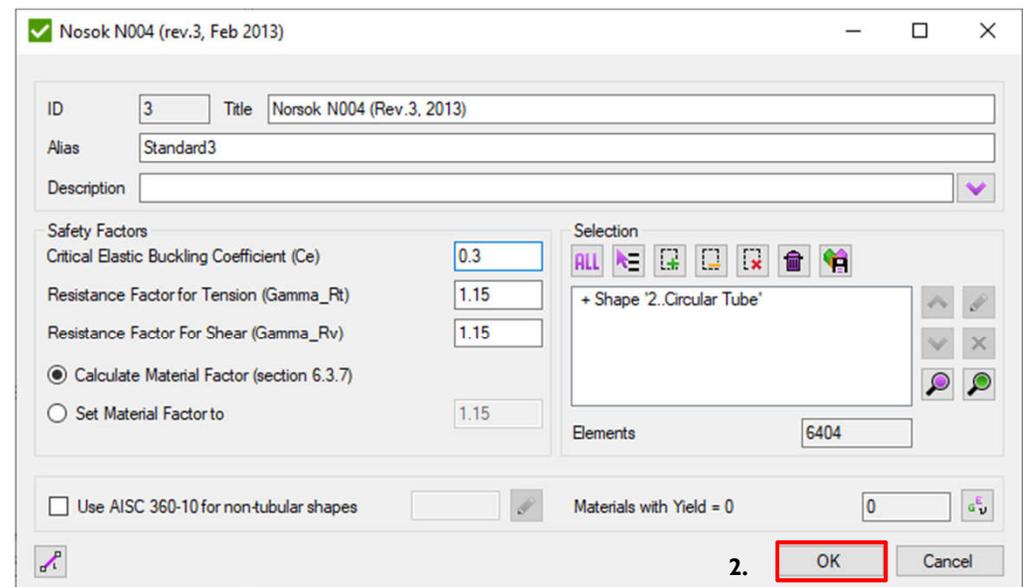
Execute *Standards* => *Add* => *Norsok N004 (rev.3, Feb 2013)*.

2

Press *OK* twice



Norsok N004 procedure is similar to API 2A RP. In overall check it is possible to verify if the structure passes all checks : Overall Utilization Factor = worst U_f among all checks < 1.



Add Norsok N004 standard

$$\begin{aligned} C_e &= \text{critical elastic buckling coefficient} = 0.3 \\ \gamma_{R,t} &= \text{material factor for tension} = 1.15 \\ \gamma_{R,v} &= \text{material factor for shear} = 1.15 \\ \gamma_M &= \text{see section 6.3.7} \\ \gamma_M &= 1.15 \quad \text{for } \bar{\lambda}_s < 0.5 \\ \gamma_M &= 0.85 + 0.60\bar{\lambda}_s \quad \text{for } 0.5 \leq \bar{\lambda}_s \leq 1.0 \\ \gamma_M &= 1.45 \quad \text{for } \bar{\lambda}_s > 1.0 \end{aligned} \tag{6.22}$$

where

$$\bar{\lambda}_s = \frac{|\sigma_{c,Sd}|}{f_{cl}} \cdot \lambda_c + \left(\frac{\sigma_{p,Sd}}{f_h} \right)^2 \cdot \lambda_h \tag{6.23}$$

where f_{cl} is calculated from Equation (6.6) or Equation (6.7) whichever is appropriate and f_h from Equation (6.17), Equation (6.18), or Equation (6.19) whichever is appropriate.

$$\lambda_c = \sqrt{\frac{f_y}{f_{cle}}}, \text{ and } \lambda_h = \sqrt{\frac{f_y}{f_{he}}} \tag{6.24}$$

f_{cle} and f_{he} is obtained from Equation (6.8), and Equation (6.20) respectively.

$\sigma_{p,Sd}$ is obtained from Equation (6.16) and

$$\sigma_{c,Sd} = \frac{N_{Sd}}{A} + \frac{\sqrt{M_{y,Sd}^2 + M_{z,Sd}^2}}{W} \tag{6.25}$$

N_{Sd} is negative if in tension.

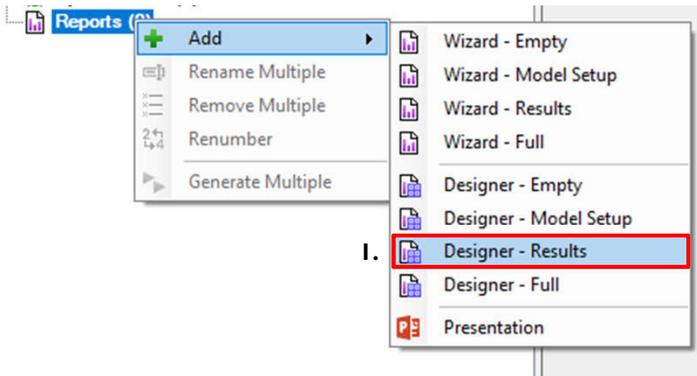
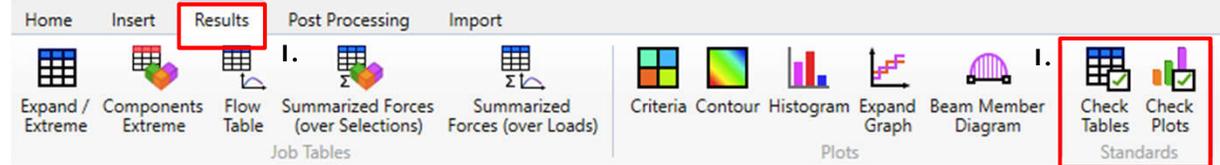
Report

1

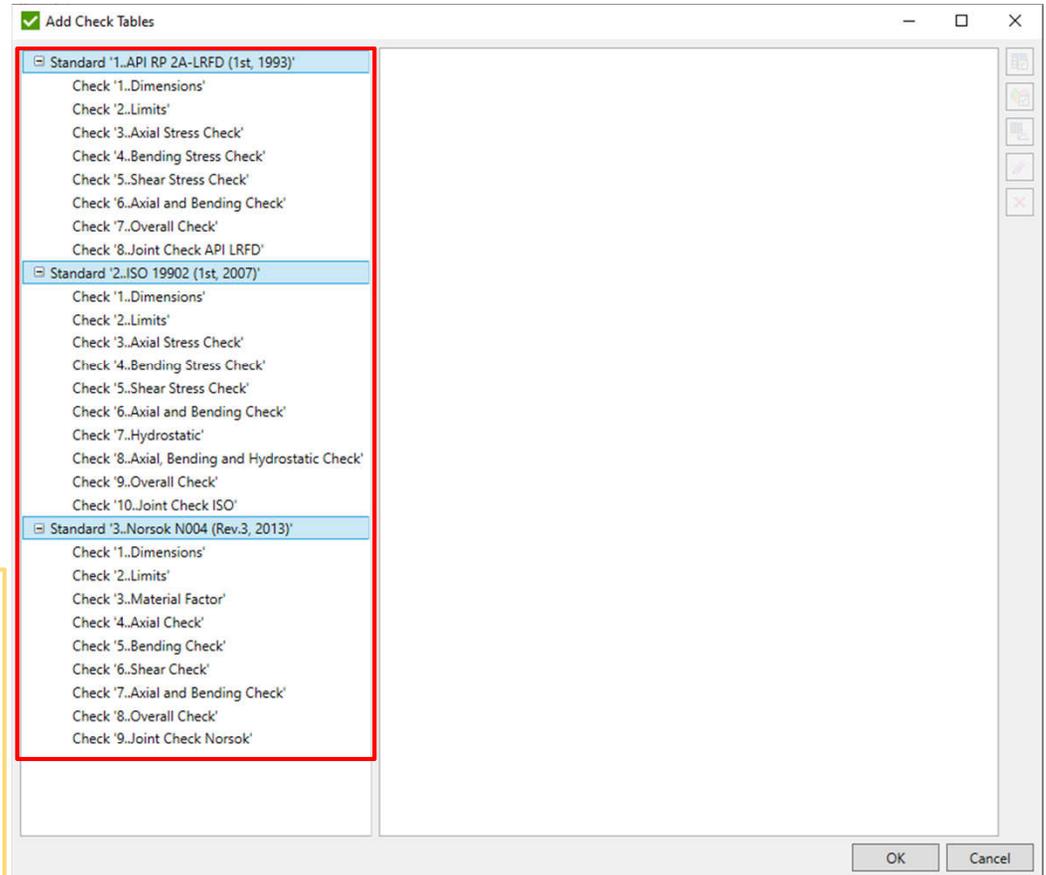
Execute *Reports – Add – Designer – Results*.

2

Results – Check Tables/Check Plots - all checks except the '**8.Overall check**' in each standard.



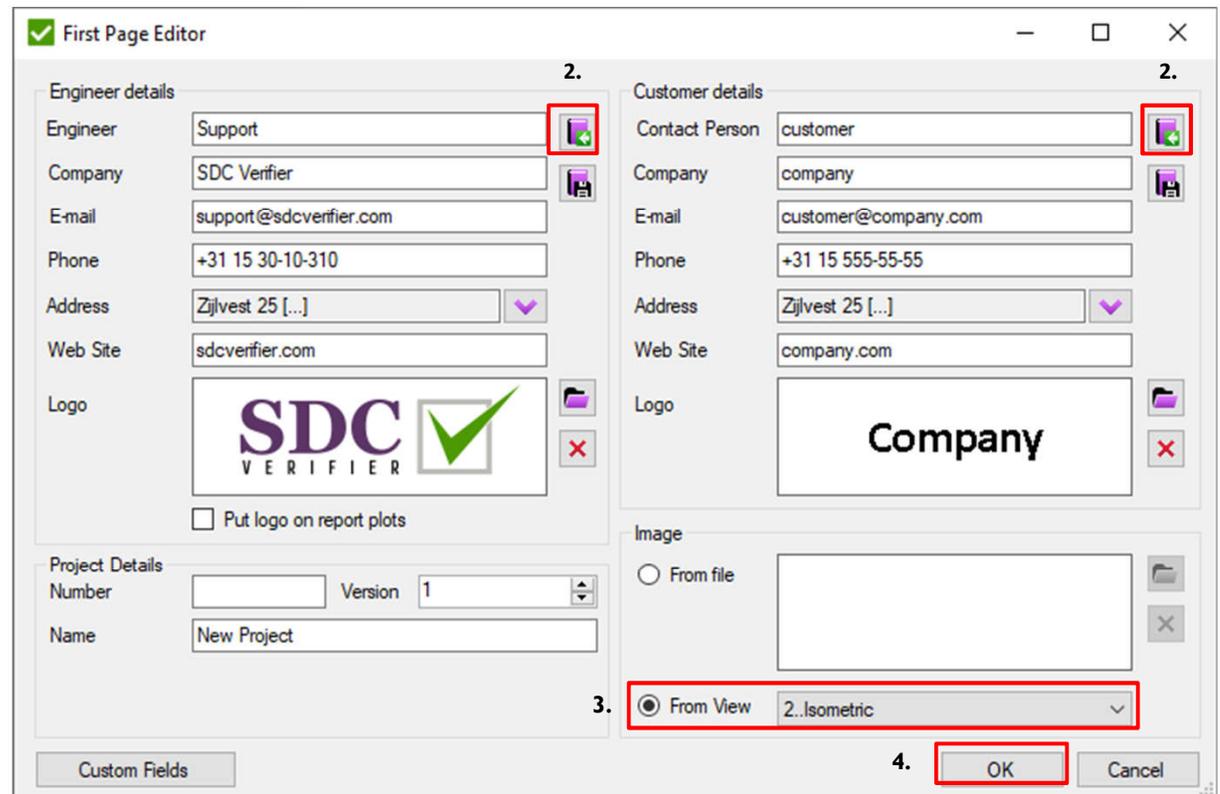
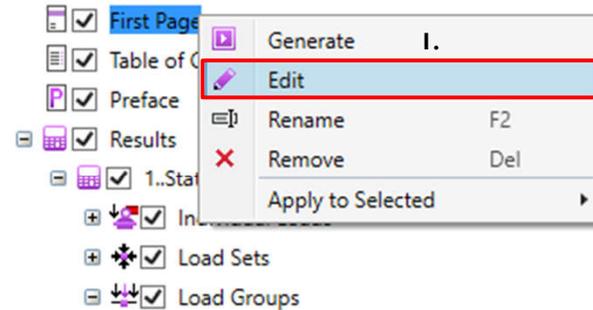
2.



There are 4 templates of the reports:
Empty – only first page and preface items are included;
Model Setup – description of the model data (materials, properties, components, boundary conditions) is included;
Results – for each load extreme displacement tables, stress and displacement plots are included. Predefined tables: sum of reaction forces, stresses/displacements summary tables;
Full – Model Setup + Results + all tables created in Job.

Report. First Page

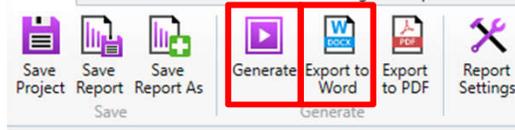
- 1 Right click on *First Page* => *Edit*.
- 2 Press  to load engineer and customer info from library
- 3 Select Image *From View* and pick '*2..Isometric*'.
- 4 Press *OK*.



Report exported to Microsoft Word



Press to generate complete report.



Press to export to Word.

First page

API 2A RP

ISO 19902

Norsok N004