IDEA StatiCa Connection

Background Teorico

Ottobre 2016

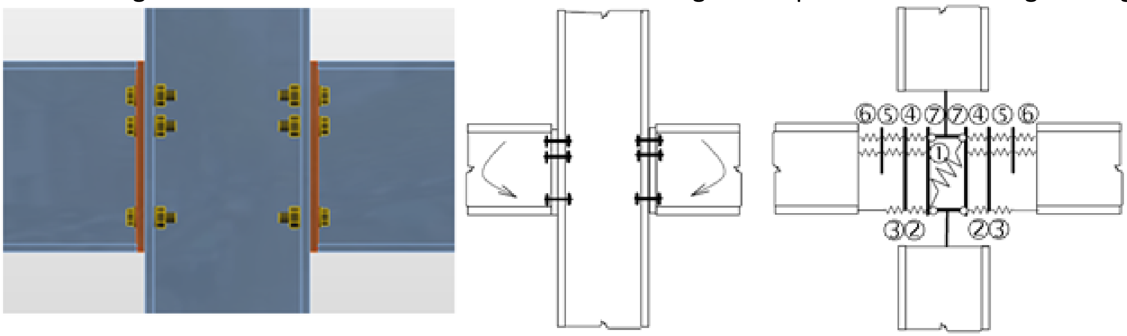
Contenuti

1 Introduzione

L’elemento trave è preferito dagli ingegneri nell’ambito della progettazione di strutture in acciaio. Ci sono però molti punti della struttura in cui la teoria della trave non è valida, ad esempio nei giunti saldati, nelle connessioni bullonate, in presenza di fori nelle pareti, di rastremazioni dell’altezza delle sezioni o di carichi puntuali. L’analisi strutturale in questi punti è difficile e richiede particolare attenzione. Il comportamento infatti è non-lineare e queste non-linearità vanno rispettate, come la plasticizzazione dei materiali che costituiscono le piastre, i punti di contatto tra piastre di estremità o piastre di base e un blocco di cemento, l’irreversibilità delle azioni dei bulloni e degli ancoraggi, le saldature. La normativa nazionale ceca, CSN EN1993-1-8, cosi come la letteratura tecnica, offrono metodi ingegneristici per la soluzione di questi problemi. La loro caratteristica generale è il fatto che siano ricavati da forme strutturali tipiche e casi di carico elementari. Molto utilizzato è il metodo delle componenti.

Metodo delle componenti

Il metodo delle componenti risolve i giunti come un sistema di elementi interconnessi tra loro – le componenti, appunto. Il modello corrispondente è composto per ogni tipo di giunto, in modo da essere in grado di determinare forze e tensioni in ogni componente – vedi figura seguente.



1 – column web in shear, 2 – column web in compression, 3 – beam flange and web in compression, 4 - column flange in bending, 5 – bolts in tension, 6 – end plate in bending and 7 – column web in tension.

Elementi nel nodo con piastre di estremità bullonate, rappresentate con delle molle.

Ogni componente viene controllata separatamente usando i metodi corrispondenti. Poiché ogni tipo di giunto necessita di un proprio modello specifico, l’utilizzo di questo metodo ha dei limiti nei casi in cui si debbano risolvere giunti di geometrie generiche e carichi qualunque.

IDEA RS, assieme al team di lavoro del Dipartimento di Strutture in Acciaio e Legno della Facoltà di Ingegneria Civile di Praga e dell’Istituto di Strutture in Acciaio e Legno della Facoltà di Ingegneria Civile della Brno University of Technology, ha sviluppato un nuovo metodo per la progettazione avanzata dei giunti nelle strutture in acciaio.

Il nome di questo metodo è CBFEM - Component Based Finite Element Model – ed è:

- **Generale** abbastanza da poter essere utilizzato per la maggior parte dei giunti, degli appoggi e dei dettagli nella pratica ingegneristica.
- **Semplice e veloce** a sufficienza nella pratica quotidiana in modo da fornire risultati in tempi comparabili a quelli dei metodi e degli strumenti attualmente in uso.
- **Esauriente** abbastanza da fornire informazioni chiare di ingegneria strutturale riguardo il comportamento del giunto, tensioni, deformazioni e riserve di deformazione dei singoli componenti e infine riguardo la sicurezza e la realizzabilità del sistema complessivo.

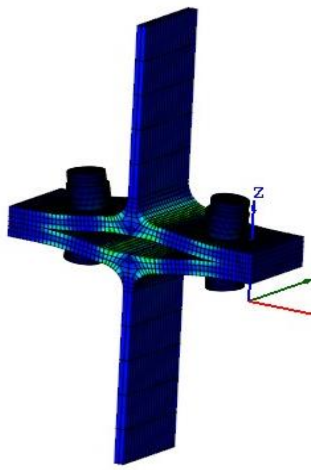
Il metodo CBFEM è basato sull’idea che la maggior parte degli aspetti verificati e molto utili del metodo delle componenti debbano essere conservate. Il punto debole del metodo delle componenti, ossia la sua generalizzazione nell’analizzare le tensioni delle singole componenti, è stato sostituito dalla modellazione e analisi tramite metodo degli elementi finiti.

2 Componenti CBFEM

FEM è un metodo generale ccomune

ly used for structural analysis. Usage of FEM for modelling of joints of any shapes seems to offer directly (Virdi, 1999). The elastic-plastic analysis is required. Steel plasticises ordinarily in the structure. In fact, the results of linear analysis are useless for joints design.

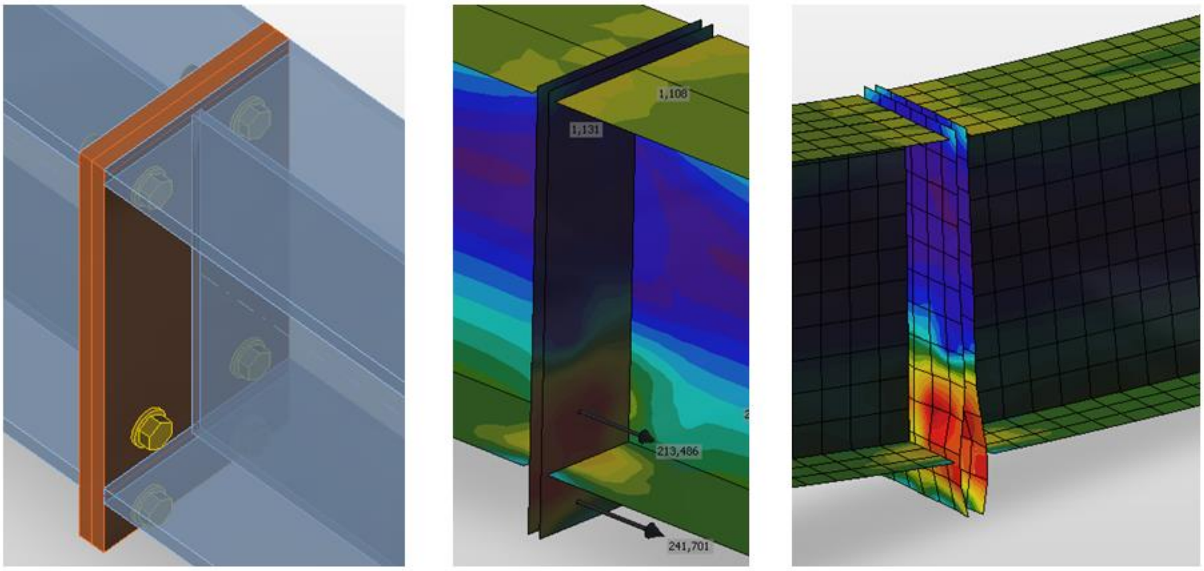
FEM models are used for research purposes of joint behaviour, which usually apply spatial elements and measured values of material properties.



FEM model of joint for research. It uses spatial 3D elements for both plates and bolts.

Both webs and flanges of connected member are modelled using thin plates in CBFEM model, for which the known and verified solution is available.

The fasteners – bolts and welds – are the most difficult in the point of view of the analysis model. Modelling of such elements in general FEM programs is difficult, because the programs do no offer required properties. Thus special FEM components had to be developed to model the welds and bolts behaviour in joint.

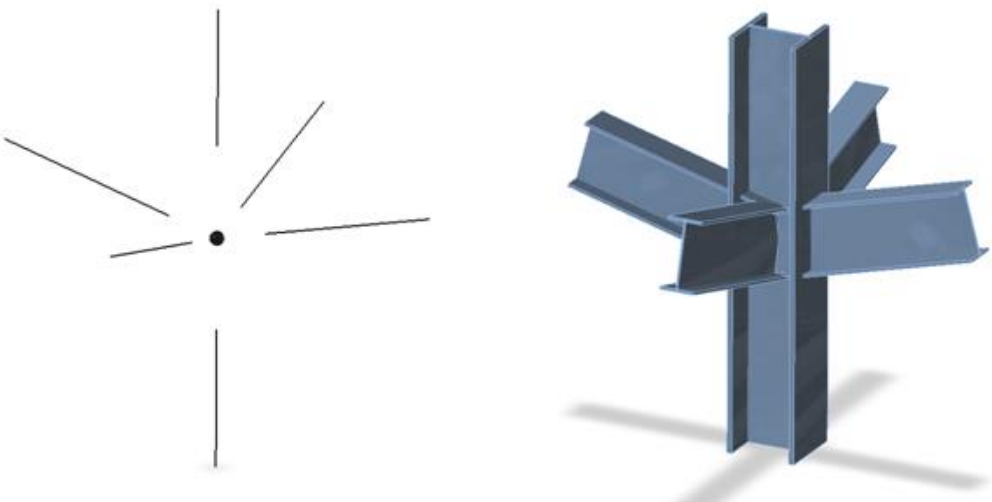


CBFEM model of bolted connection by end plates

Joints of members are modelled as massless points when analysing steel frame or girder structure. Equilibrium equations are assembled in joints and after solving the whole structure internal forces on ends of beams are determined. In fact, the joint is loaded by those forces. The resultant of forces from all members in the joint is zero – the whole joint is in equilibrium.

The real shape of joint is not known in the structural model. The engineer only defines, if the joint is assumed to be rigid or hinged.

It is necessary to create the trustworthy model of joint, which respect the real state, to design the joint properly. Ends of members with length of 2-3 multiple of maximal cross-section height are used in CBFEM method. These segments are modelled using plate/wall elements.



Theoretical (massless) joint and real shape of joint without modified member ends.

For better precision of CBFEM model, the end forces on 1D members are applied as loads on segment ends. Sextuplets of forces from the theoretical joint are transferred to the end of segment –

the values of forces are kept, but the moments are modified by the actions of forces on corresponding arms.

Segment ends at the joint are not connected. The connection must be modelled. So called manufacturing operations are used in CBFEM method to model the connection. Manufacturing operations especially are: cuts, offsets, holes, stiffeners, ribs, end plates and splices, angles, gusset plates and other. Fastening elements are added to them – welds and bolts.

2.1 Material model

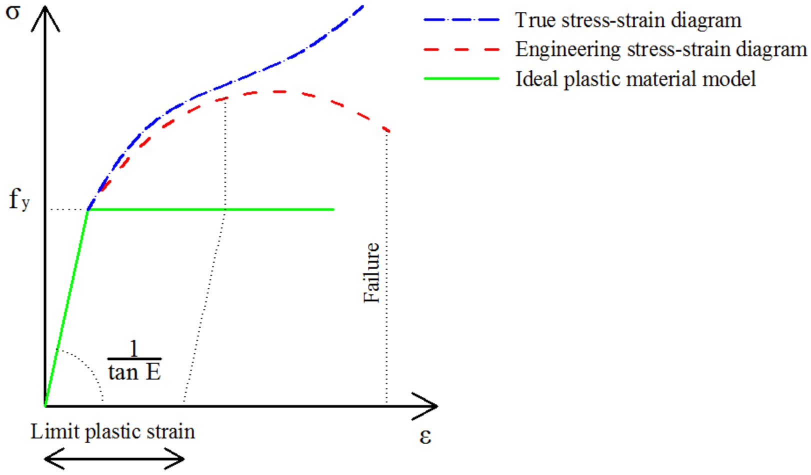
The most common material diagrams, which are used in finite element modelling of structural steel, are the ideal plastic or elastic model with strain hardening and the true stress-strain diagram. The true stress-strain diagram is calculated from the material properties of mild steels at ambient temperature obtained in tensile tests. The true stress and strain may be obtained as follows:

$\sigma_{true} = \sigma\,(1 + \varepsilon)$

$\varepsilon_{true} = \ln(1 + \varepsilon)$

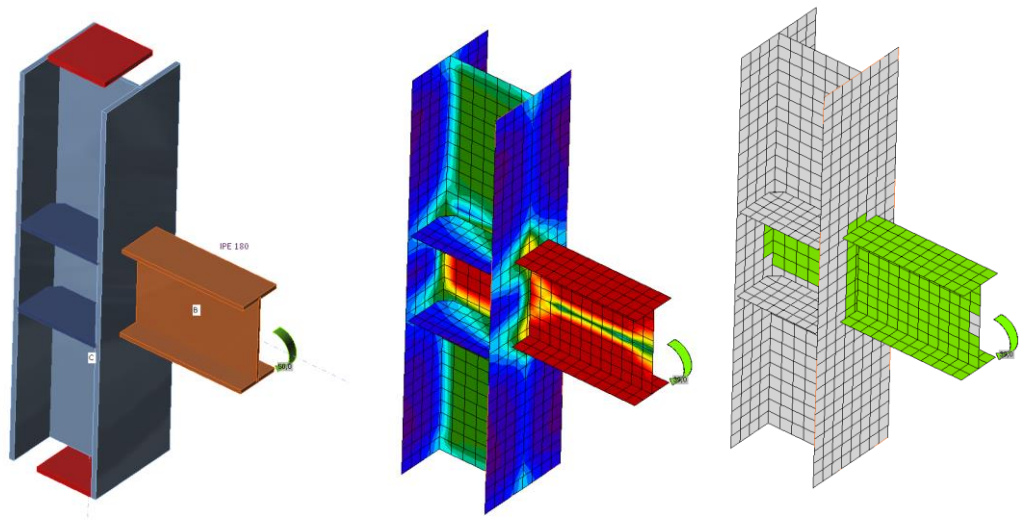
where σ_{true} is true stress, ε_{true} true strain, σ nominal stress and ε nominal strain. The elastoplastic material with strain hardening is modelled according to EN1993-1-5:2005. The material behaviour is based on von Mises yield criterion. It is assumed to be elastic before reaching the yield strength f_y.

The ultimate limit state criteria for regions not susceptible to buckling is reaching of a limiting value of the principal membrane strain. The value of 5% is recommended (e.g. EN1993-1-5 app. C par. C8 note 1).



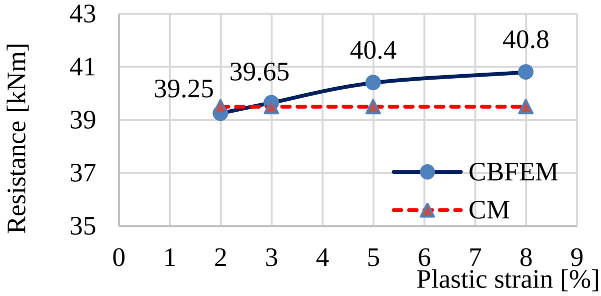
Material diagrams of steel in numerical models

The limit value of plastic strain is often discussed. In fact that ultimate load has low sensitivity to the limit value of plastic strain when ideal plastic model is used. It is demonstrated on the following example of a beam to column joint. An open section beam IPE 180 is connected to an open section column HEB 300 and loaded by bending moment. The influence of the limit value of plastic strain on the resistance of the beam is shown in following picture. The limit plastic strain is changing from 2 % to 8 %, but the change in moment resistance is less than 4 %.



Loads Stress Strain

Example of prediction of ultimate limit state of a beam to column joint



Influence of the limit value of plastic strain on the moment resistance

2.2 Plate model and mesh convergence

2.2.1 Plate model

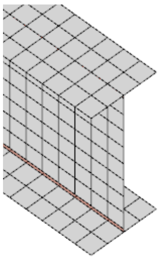
Shell elements are recommended for modelling of plates in design FEA of structural connection. 4-node quadrangle shell elements with nodes at its corners are applied. Six degrees of freedom are considered in every node: 3 translations (u_x, u_y, u_z) and 3 rotations (φ_x, φ_y, φ_z). Deformations of the element are divided into membrane and flexural components.

The formulation of the membrane behaviour is based on the work by Ibrahimbegovic (1990). Rotations perpendicular to the plane of the element are considered. Complete 3D formulation of the element is provided. The out-of-plane shear deformations are considered in the formulation of the flexural behaviour of element based on Mindlin hypothesis. The MITC4 elements are applied, see Dvorkin (1984). The shell is divided into five integration points along the height of the plate and plastic behavior is analyzed in each point. It is called Gaus - Lobatto integration. The nonlinear elastic-plastic stage of material is analyzed in each layer based on the known strains.

2.2.2 Mesh convergence

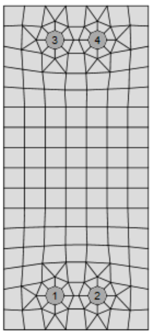
There are some criteria of the mesh generation in the connection model. The connection check should be independent on the element size. Mesh generation on a separate plate is problem-free. The attention should be paid to complex geometries such as stiffened panels, T-stubs and base plates. The sensitivity analysis considering mesh discretisation should be performed for complicated geometries.

All plates of a beam cross-section have common size of elements. Size of generated finite elements is limited. Minimal element size is set to 10 mm and maximal element size to 50 mm. Meshes on flanges and webs are independent on each other. Default number of finite elements is set to 8 elements per cross-section height as shown in figure.



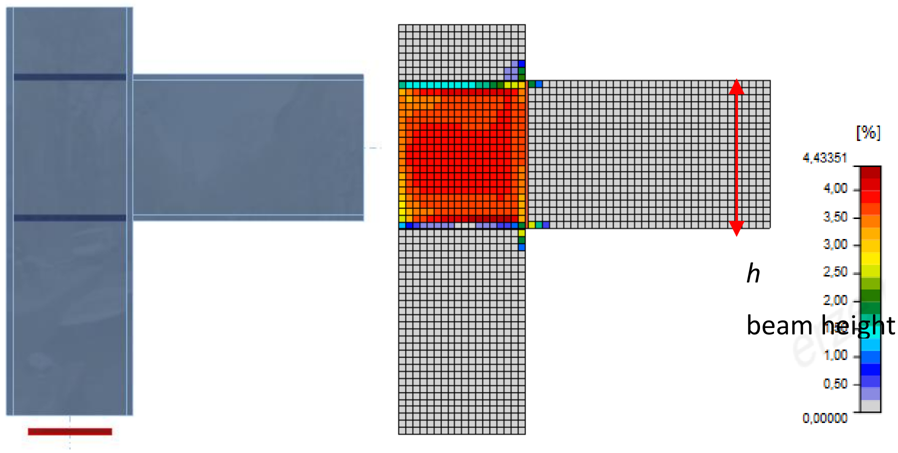
Mesh on beam with constrains between web and flange plate

The mesh of end plates is separate and independent on other connection parts. Default finite element size is set to 16 elements per cross-section height as shown in figure.

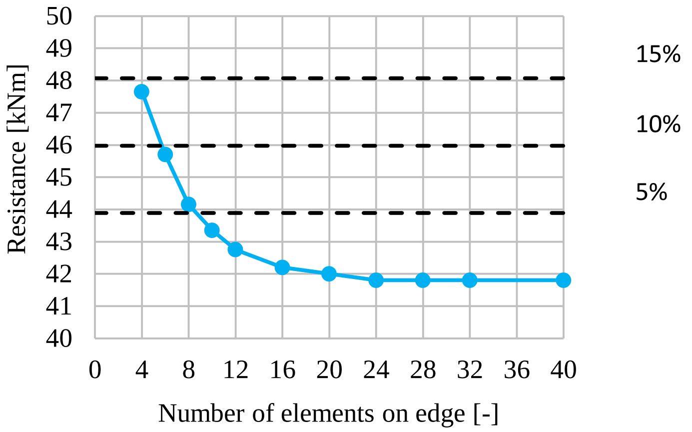


Mesh on end plate, with 7 elements on width

Following example of a beam to column joint shows the influence of mesh size on the moment resistance. An open section beam IPE 220 is connected to an open section column HEA200 and loaded by bending moment, as shown in following picture. The critical component is column panel in shear. The number of finite elements along the cross-section height is changing from 4 to 40 and the results are compared. Dashed lines are representing 5%, 10% and 15% difference. It is recommended to subdivide the cross-section height into 8 elements.

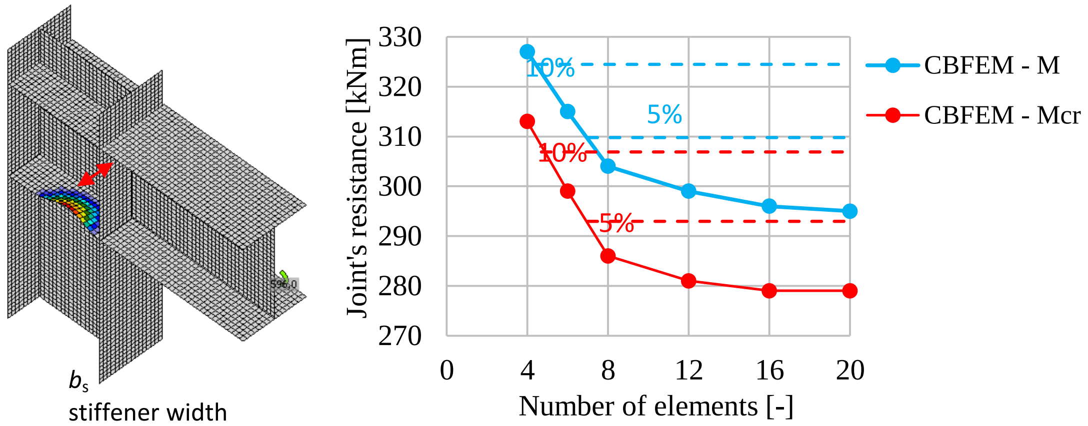


Beam to column joint model and plastic strains at ultimate limit state



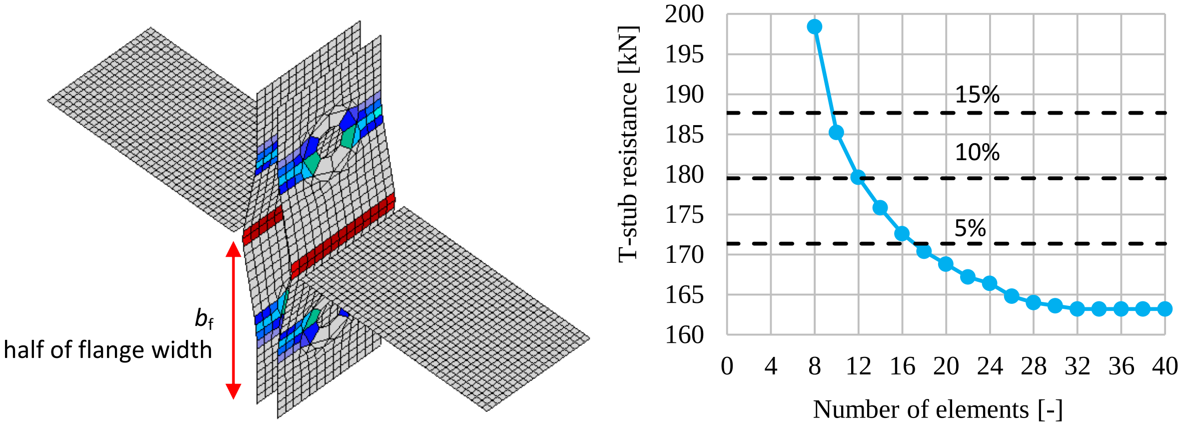
Influence of number of elements on the moment resistance

Mesh sensitivity study of a slender compressed stiffener of column web panel is presented. The geometry of the example is taken from section 6.3. The number of elements along the width of the stiffener is changed from 4 to 20. The first buckling mode and the influence of number of elements on the buckling resistance and critical load are shown in following picture. The difference of 5% and 10% are displayed. It is recommended to use 8 elements along the stiffener width.

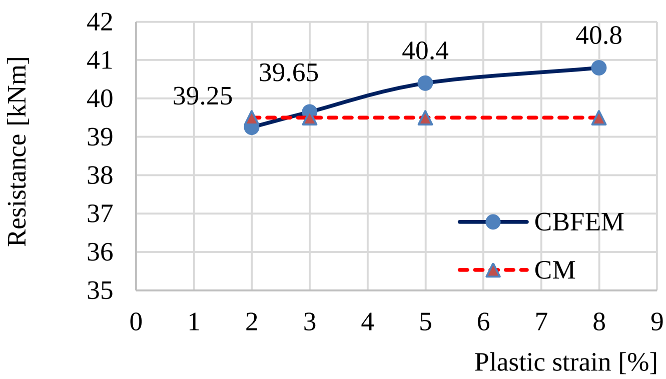


First buckling mode and influence of number of elements along the stiffener on the moment resistance

Mesh sensitivity study of T-stub in tension is presented. The geometry of the T-stub is described in section 5.1. The half of the flange width is subdivided into 8 to 40 elements and the minimal element size is set to 1 mm. The influence of number of elements on the T-stub resistance is shown in following picture. The dashed lines are representing the 5%, 10% and 15% difference. It is recommended to use 16 elements on the half of the flange width.



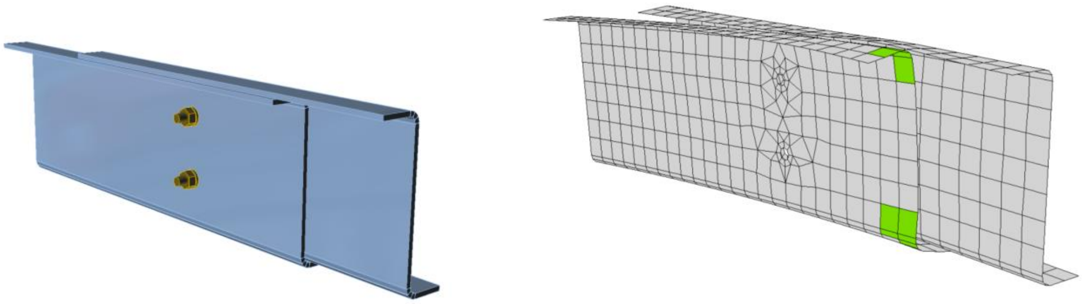
Influence of number of elements on the T-stub resistance



Influence of the limit value of plastic strain on the moment resistance

2.3 Contacts

The standard penalty method is recommended for modelling of a contact between plates. If penetration of a node into an opposite contact surface is detected, penalty stiffness is added between the node and the opposite plate. The penalty stiffness is controlled by heuristic algorithm during nonlinear iteration to get better convergence. The solver automatically detects the point of penetration and solves the distribution of contact force between the penetrated node and nodes on the opposite plate. It allows to create the contact between different meshes as shown. The advantage of the penalty method is the automatic assembly of the model. The contact between the plates has a major impact on the redistribution of forces in connection.



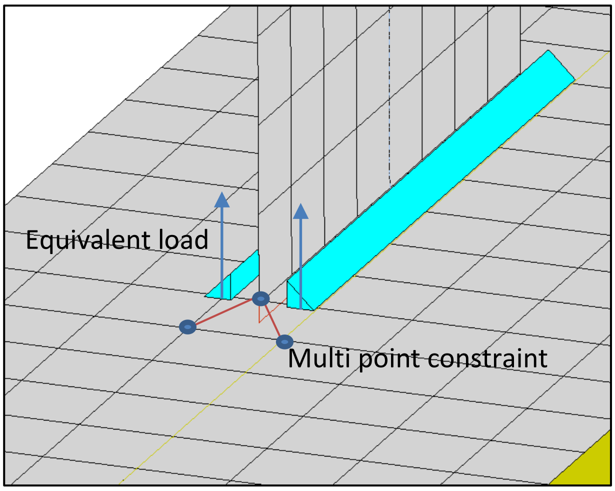
Example of separation plates in contact between web and flanges of two overlapped Z sections purlins

2.4 Welds

There exist several options how to treat welds in numerical models. Large deformations makes the mechanical analysis more complex and it is possible to use different mesh descriptions, different kinetic and kinematic variables, and constitutive models. The different types of geometric 2D and 3D models and thereby finite elements with their applicability for different accuracy levels are generally used. Most often used material model is the common rate-independent plasticity model based on von Mises yield criterion. Two approaches which are used for welds are described.

2.4.1 Direct connection of plates

The first option of weld model between plates is direct merge of meshes. The load is transmitted through a force-deformation constrains based on Lagrangian formulation to opposite plate. The connection is called multi point constraint (MPC) and relates the finite element nodes of one plate edge to another. The finite element nodes are not connected directly. The advantage of this approach is the ability to connect meshes with different densities. The constraint allows to model midline surface of the connected plates with the offset, which respects the real weld configuration and throat thickness. The load distribution in weld is derived from the MPC, so the stresses are calculated in the throat section. This is important for the stress distribution in plate under the weld and for modelling of T-stubs.



Constraint between mesh nodes

This model does not respect the stiffness of the weld and the stress distribution is conservative. Stress peaks, which appear at the end of plate edges, in corners and rounding, govern the resistance along the whole length of the weld. To eliminate the effect three methods for evaluation of the weld can be chosen

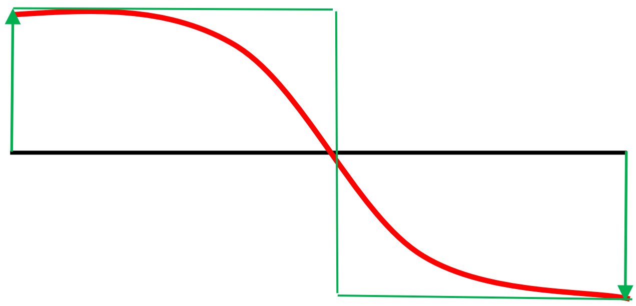
1. Maximal stress (conservative)
2. Average stress on weld
3. Linear interpolation along weld

Weld stress evaluation for direct connection

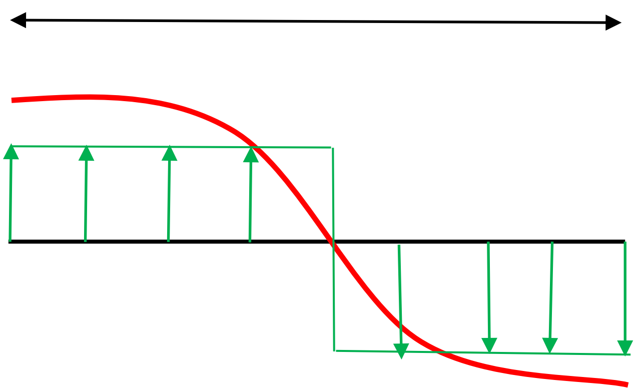
Program calculates precise values in weld link. User can decide how to evaluate the value for the check.

Method 1 can be too conservative in many cases. Method 2 simulates the situation when the whole weld can be plastic. In majority of cases it is close to the reality, but for instance for long welds this method is not appropriate. Similar situation is with method 3.

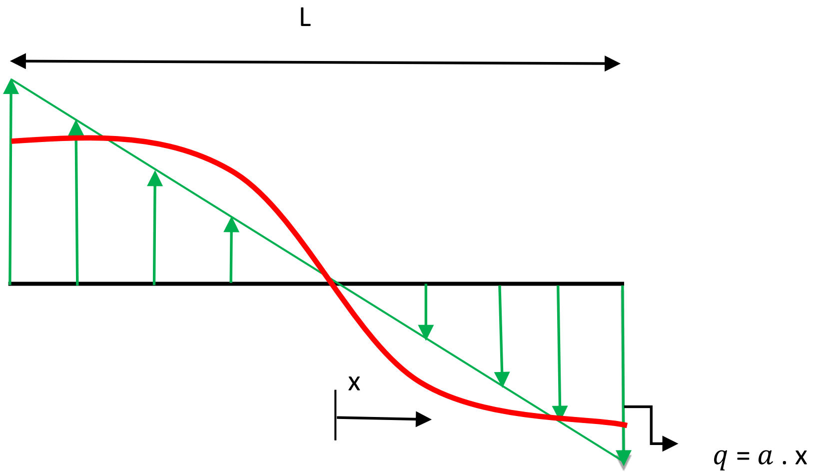
1. Maximal stress



2. Average stress



3. Linear interpolation

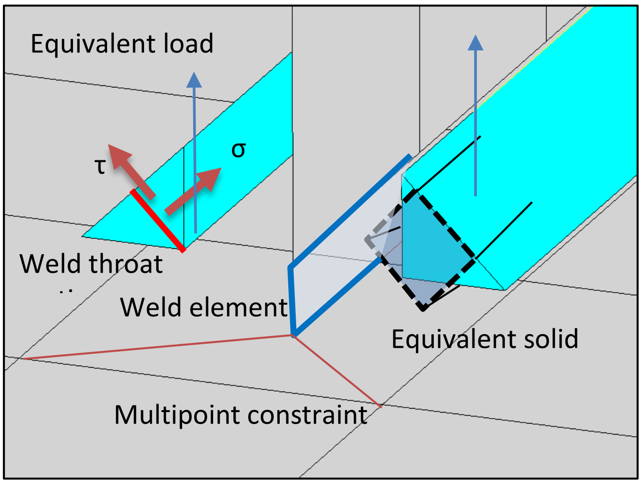


$$M = \int_{-L/2}^{L/2} q\ x\ dx = \frac{L^3\ a}{12}$$

$$a = \frac{12\ \mathrm{M}}{L^3}$$

2.4.2 Plastic welds

To express the weld behavior an improved weld model is applied. A special elastoplastic element is added between the plates. The element respects the weld throat thickness, position and orientation. The equivalent weld solid is inserted with the corresponding weld dimensions. The nonlinear material analysis is applied and elastoplastic behavior in equivalent weld solid is determinate. Ideal plastic model is used and the plasticity state is controlled by stresses in the weld throat section. The plastic strain in weld is limited to 5% as in the plate (e.g. EN1993-1-5 app. C par. C8 note 1). The stress peaks are redistributed along the longer part of the weld length.



Constraint between weld element and mesh nodes

Weld stress evaluation for plastic welds

Fully plastic model of welds gives real values of stress and there is no need to average or interpolate them. Calculated values are directly used for checks.

2.5 Bolts

In the Component based finite element method (CBFEM) is component bolt with its behavior in tension, shear and bearing by the dependent nonlinear springs. The bolt in tension is described by spring with its axial initial stiffness, design resistance, initialization of yielding and deformation capacity. The axial initial stiffness is derived analytically in guideline VDI2230. The model corresponds to experimental data, see (Gödrich et al 2014). For initialization of yielding and deformation capacity is assumed that plastic deformation occurs in the threated part of the bolt shank only. The force at beginning of yielding $F_{y,ini}$ is

$$F_{y,ini} = f_{y,b}\ A_t$$

where, $f_{y,b}$ is yield strength of bolts and A_t tensile area of the bolt. Relation gives for materials with low ratio of the ultimate strength to yield strength higher values than design resistance $F_{t,Rd}$. To assure a positive value of plastic stiffness it should be taken

$$F_{y,ini} \leq F_{t,Rd}$$

Deformation capacity of the bolt δ_c consists of elastic deformation of bolt shank δ_{el} and plastic one of the threated part only δ_{pl}.

$$\delta_c = \delta_{el} + \delta_{pl}$$

$$\delta_{el} = \frac{F_{t,Rd}}{k_{ini}}$$

where k_{ini} is initial deformation stiffness of the bolt in tension according to guideline VDI2230, and

$$\delta_{pl} = \varepsilon_{pl}\ l_t$$

where, ε_{pl} is limiting plastic strain, given by value 5%, and l_t is length of threated part. The tensile force is transmitted to the plates by interpolation links between the bolt shank and nodes in the plate. The transfer area corresponds to the mean value of the bolt shank and the circle inscribed in the hexagon of the bolt head.

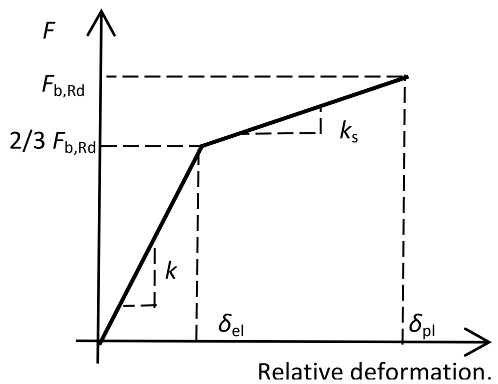
The initial stiffness and design resistance of bolts in shear is in CBFEM modelled according to in cl. 3.6 and 6.3.2 in EN1993-1-8:2006. Linear behavior up to failure is considered.

The spring representing bearing has bi-linear force deformation behavior with initial stiffness and design resistance according to in cl. 3.6 and 6.3.2 in EN1993-1-8:2006. Deformation capacity is considered according to (Wald et al 2002) as

$$\delta_{pl} = 3\ \varepsilon_{el}$$

Initialization of yielding is expected, see following figure, at

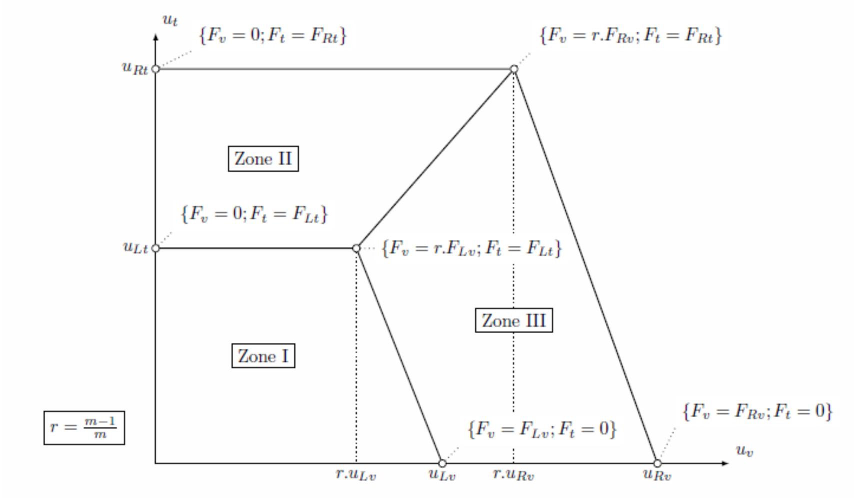
$$\mathrm{F_{ini}} = 2/3\ \mathrm{F_{b,Rd}}$$



Force deformation diagram for bearing of the plate

Interaction of axial and shear force in the bolt is considered according to Tab. 3.4 in EN1993-1-8:2006. Only the compression force is transferred from the bolt shank to the plate in the bolt hole. It is modelled by interpolation links between the shank nodes and holes edge nodes. The deformation stiffness of the shell element, which models the plates, distributes the forces between the bolts and simulates the adequate bearing of the plate.

Interaction of axial and shear force can be introduced in directly the analysis model. Distribution of forces better reflects the reality (see enclosed diagram). Bolts with high tensile force take less shear force and vice versa.



2.6 Preloaded bolts

Preloaded bolts are used in cases when the minimizing of deformation is needed. The tension model of bolt is the same as for standard bolts. Shear force is not transferred by bearing, but by friction between gripped plates.

The design slip resistance of a preloaded class 8.8 or 10.9 bolt is subjected to an applied tensile force, $F_{t,Ed}$

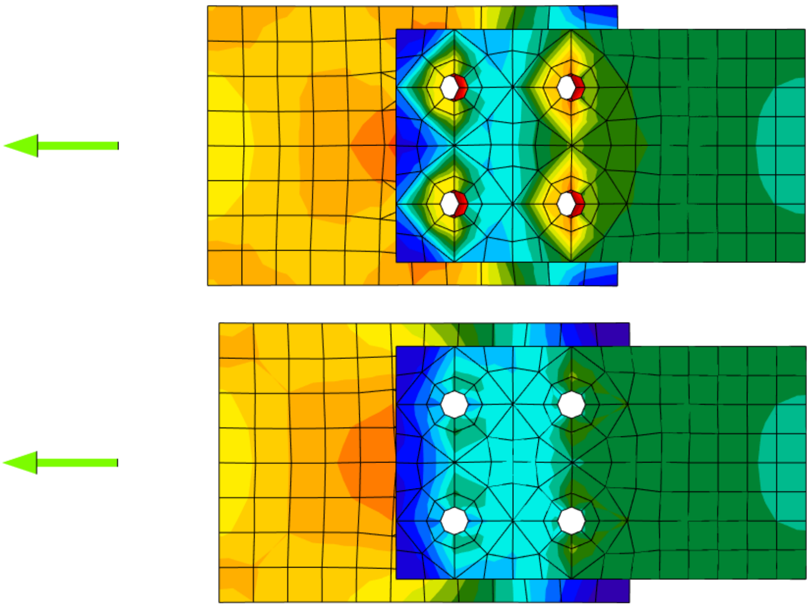
Preloading force of bolt with tensile stress area As to be used EN 1993-1-8 3.9 (3.7)
$F_{p,C}$ = 0,7 fub As.

Design slip resistance per bolt EN 1993-1-8 3.9 (3.8)
$F_{s,,Rd} = ks\ \ n\ \ \mu\ (F_{p,C} - 0{,}8\ F_{t,Ed})\ /\ \gamma\ M_3$
Where k_s is a coefficient given in Table 3.6, μ is slip factor, n is number of the friction surfaces and γM_3 is a safety factor.

IDEA StatiCa Connection checks Service Limit State of preloaded bolts. If there is a slipping effect, bolts do not satisfy the check. Then the Ultimate limit State can be checked as a standard bearing check of bolts.

User can decide which limit state to be checked. Either it is resistance to major slip or ultimate state in shear of bolts. Both checks on one bolt are not combined in one solution. It is assumed, that bolt has standard behaviour after major slip and can be checked by standard bearing procedure.

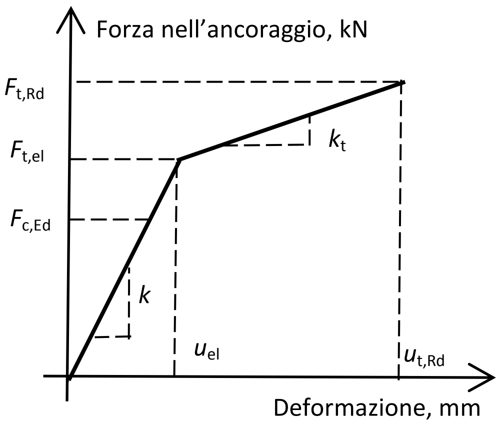
Moment load of connection has small influence to the shear capacity. But, we solved simply friction check on each bolt separately by the equations (3.8). This check is implemented in FEM component of bolt. There is no information in general way, if external tension load of each bolt is from moment or from tension load of connection.



Stress distribution in standard and slip-resistant shear bolt connection

2.7 Anchor bolts

The anchor bolt is modelled with similar procedures as structural bolts. The bolt is on one side fixed to the concrete block. Its length L_b is taken according to EN1993-1-8:2006 as sum of washer thickness t_w, base plate thickness t_{bp}, grout thickness t_g and free length embedded in concrete, which is expected as 8*d*, where *d* is bolt diameter. The stiffness in tension is calculated as $k = E\, A_s/L_b$. The load-deformation diagram of the anchor bolt is shown in following figure. The values according to ISO 898:2009 are summarised in table and in formulas below.



Load–deformation diagram of the anchor bolt

$$F_{t,el} = \frac{F_{t,Rd}}{c_1 \cdot c_2 - c_1 + 1}$$

$$k_t = c_1 \cdot k;\ c_1 = \frac{R_m - R_e}{\left(\frac{1}{4} \cdot A - \frac{R_e}{E}\right) \cdot E}$$

$$u_l = \frac{F_{t,el}}{k}$$

$$u_{t,Rd} = c_2 \cdot u_l;\ c_2 = \frac{A \cdot E}{4 \cdot R_e}$$

Grade	R_m	$R_e = R_{p02}$	A	E	c_1	c_2
	[MPa]	[MPa]	[%]	[MPa]	[-]	[-]
4.8	420	340	14	2,1E+05	0,011	21,6
5.6	500	300	20	2,1E+05	0,020	35,0
5.8	520	420	10	2,1E+05	0,021	12,5
6.8	600	480	8	2,1E+05	0,032	8,8
8.8	830	660	12	2,1E+05	0,030	9,5
10.9	1040	940	9	2,1E+05	0,026	5,0

Anchor bolt parameters, based to ISO 898:2009

The stiffness of the anchor bolt in shear is taken as the stiffness of the structural bolt in shear. The anchor bolt resistance is evaluated according to ETAG 001 Annex C or prEN1992-1-4.2015. Steel failure mode is determined according to cl. 6.2.6.12 in EN 1993-1-8.

2.8 Concrete block

2.8.1 Design model

In component based finite element method (CBFEM), it is convenient to simplify the concrete block as 2D contact elements. The connection between the concrete and the base plate resists in compression only. Compression is transferred via Winkler-Pasternak subsoil model, which represents deformations of the concrete block. Tension force between the base plate and concrete block is carried by anchor bolts. Shear force is transferred by friction between a base plate and a concrete block, by shear key, and by bending of anchor bolts and friction. The resistance of bolts in shear is assessed analytically. Friction and shear key are modelled as a full single point constraint in the plane of the base plate-concrete contact.

2.8.2 Resistance

The resistance of concrete in 3D compression is determined based on EN 1993-1-8:2006 by calculating the design bearing strength of concrete in the joint f_{jd} under the effective area A_{eff} of the base plate. The design bearing strength of the joint f_{jd} is evaluated according to Cl. 6.2.5 in EN 1993-1-8:2006 and Cl. 6.7 in EN 1992-1-1:2005. The grout quality and thickness is introduced by the joint coefficient β_{jd}. For grout quality equal or better than quality of the concrete block is expected β_{jd} =

1,0. The effective area A_{eff} under the base plate is estimated to be of the shape of the column cross-section increased by additional bearing width c

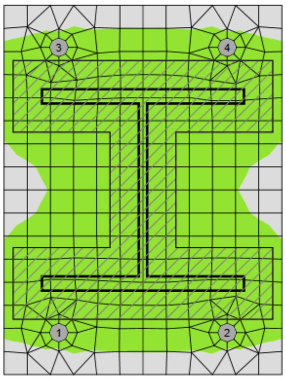
$$c = t\sqrt{\frac{f_{\mathrm{y}}}{3\, f_{\mathrm{j}}\, \gamma_{\mathrm{M0}}}}$$

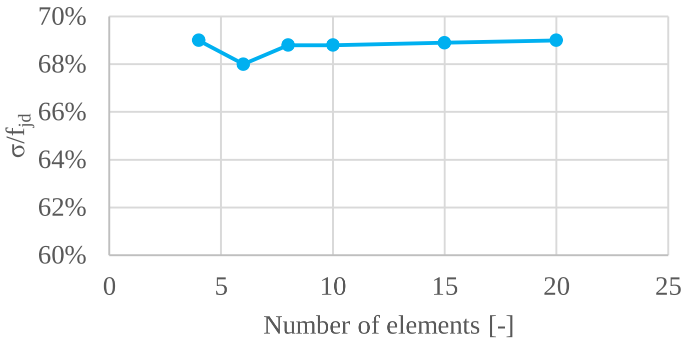
where t is the thickness of the base plate, f_{y} is the base plate yield strength, γ_c is the partial safety factor for concrete and γ_{M0} is the partial safety factor for steel.

The effective area is calculated by iteration until the difference between additional bearing widths of current and previous iteration $|c_i - c_{i-1}|$ is less than 1 mm.

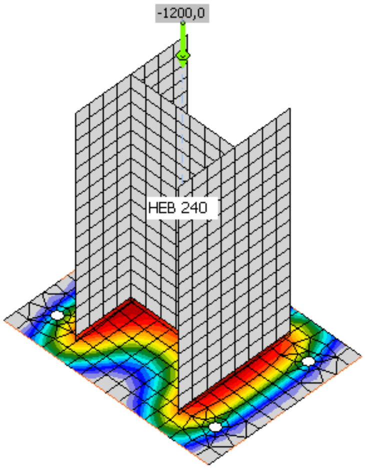
The area where the concrete is in compression is taken from results of FEA. This area in compression A_{com} allows to determine the position of neutral axis. The intersection of the area in compression A_{com} and the effective area A_{eff} allows to assess the resistance for generally loaded column base of any column shape with any stiffeners. The average stress σ on the effective area A_{eff} is determined as the compression force divided by the effective area. Check of the component is in stresses $\sigma \le f_{jd}$.

This procedure of assessing the resistance of the concrete in compression is independent on the mesh of the base plate as can be seen in figures bellow. Two cases were investigated: loading by pure compression 1200 kN, and loading by combination of compressive force 1200 kN and bending moment 90 kN.

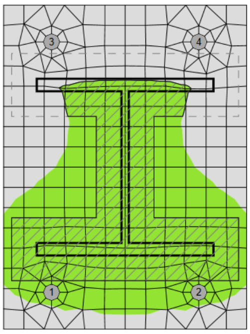


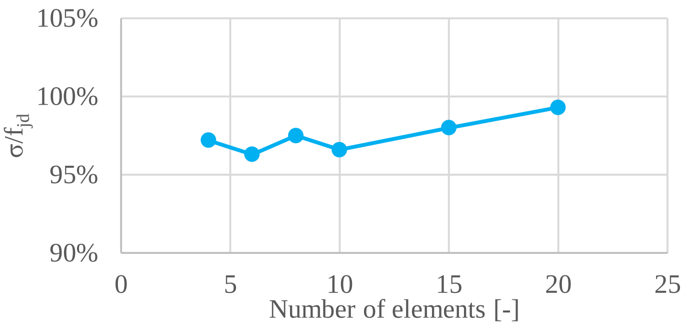


No. of elements	A_{eff} [m^2]	σ [MPa]	f_{jd} [MPa]
4	0,06	18,5	26,8
6	0,06	18,2	26,8
8	0,06	18,5	26,8
10	0,06	18,4	26,8
15	0,06	18,5	26,8
20	0,06	18,5	26,8

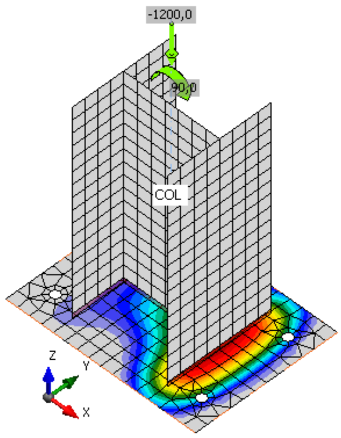


Influence of number of elements on prediction of resistance of concrete in compression in case of pure compression





No. of elements	A_{eff} [m^2]	σ [MPa]	f_{jd} [MPa]
4	0,05	26,0	26,8
6	0,04	25,8	26,8
8	0,04	26,1	26,8
10	0,05	25,9	26,8
15	0,04	26,3	26,8
20	0,04	26,6	26,8



Influence of number of elements on prediction
of resistance of concrete in compression in case of compression and bending

2.8.3 Deformation stiffness

The stiffness of the concrete block may be predicted for design of column bases as elastic hemisphere. A Winkler-Pasternak subsoil model is commonly used for a simplified calculation of foundations. The stiffness of subsoil is determined using modulus of elasticity of concrete and effective height of subsoil as

$$k = \frac{E_c}{(\alpha_1 + \upsilon) \cdot \sqrt{\dfrac{A_{eff}}{A_{ref}}}} \cdot \left(\frac{1}{\dfrac{h}{\alpha_2 \cdot d} + \alpha_3} + \alpha_4 \right) \qquad (3.7.2)$$

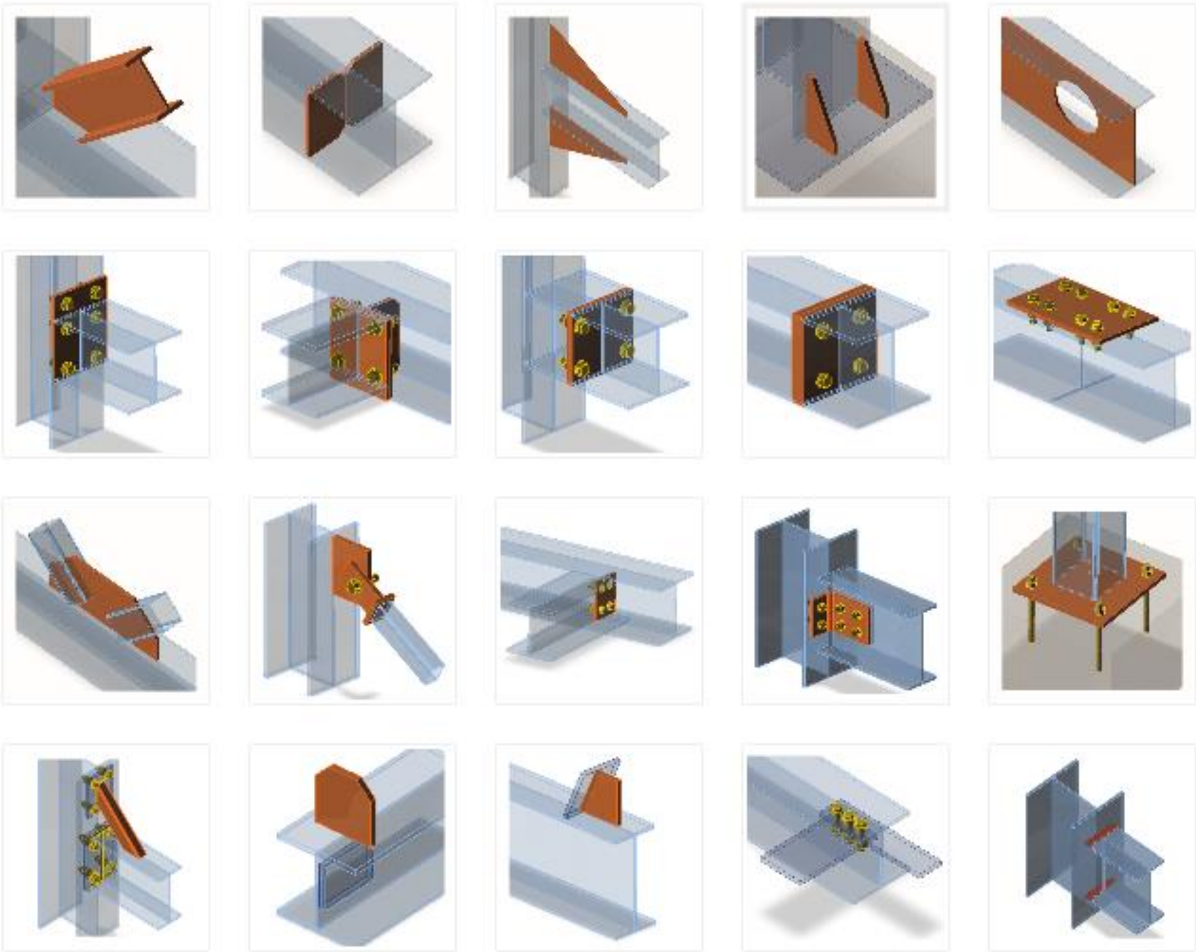
where, k is stiffness in compression, E_c is modulus of elasticity, n is Poisson coefficient of concrete foundation, A_{eff} is effective area, A_{ref} is reference area, d is base plate width, h is column base height, and α_i are coefficients. The following values for coefficient were used: A_{ref} =10 m^2; α_1 = 1,65; α_2 = 0,5; α_3 = 0,3; α_4 =1,0.

3 Analysis

3.1 Analysis model

New components method (CBFEM – Component Based Finite Element Model) enables fast analysis of joints of several shapes and configurations.

The analysis FEM model is generated automatically. The designer does not create the analysis FEM model, he creates the joint using manufacturing operations – see the picture.



Manufacturing operations/items which can be used to construct the joint

Each manufacturing operation adds new items into the connection – cuts, plates, bolts, welds.

3.2 Bearing member and supports

One member of joint is always set as “bearing”. All other members are “connected”. The bearing member can be chosen by designer. The bearing member can be “continuous” or “ended” in the joint. “Ended” members are always terminated in the joint.

Connected members can be of several types, according to the load, which the member can take:

- Type N-Vy-Vz-Mx-My-Mz – member is able to transfer all 6 components of internal forces.
- Type N-Vy-Mz – member is able to transfer only loading in XY plane – internal forces N, Vy, Mz.
- Type N-Vz-My – member is able to transfer only loading in XZ plane – internal forces N, Vz, My.
- Type X – member is able to transfer only loading in X direction– normal force N.

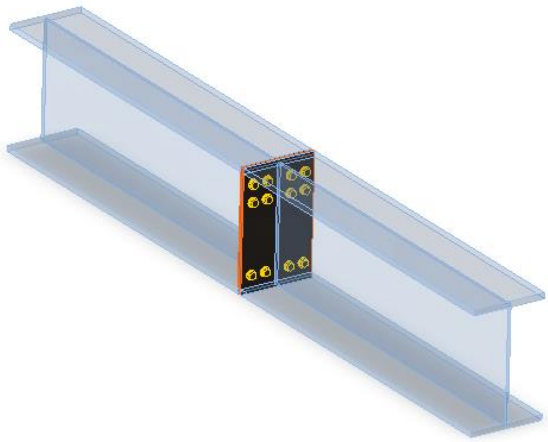
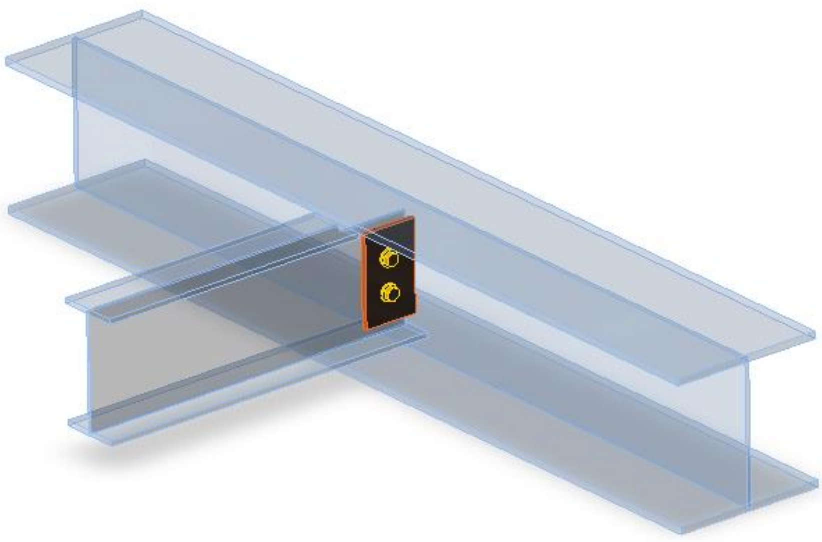
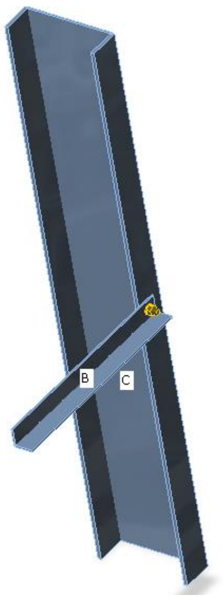


Plate to plate connection transfers all components of internal forces



Fin plate connection. The connection can transfer only loads in XZ plane – internal forces N, Vz, My.



Gusset connection – connection of truss member. The connection can transfer only axial force N.

Each joint is in the state of equilibrium during analysis of the frame structure. If the end forces of individual members are applied to detailed CBFEM model, the state of equilibrium is met too. Thus it would be not necessary to define supports in analysis model. However, for practical reasons, the support resisting all translations is defined in the first end of bearing member. It does influence neither state of stress nor internal forces in the joint, only the presentation of deformations.

Appropriate support types respecting the type of individual members are defined at ends of connected members to prevent occurrence of instable mechanisms.

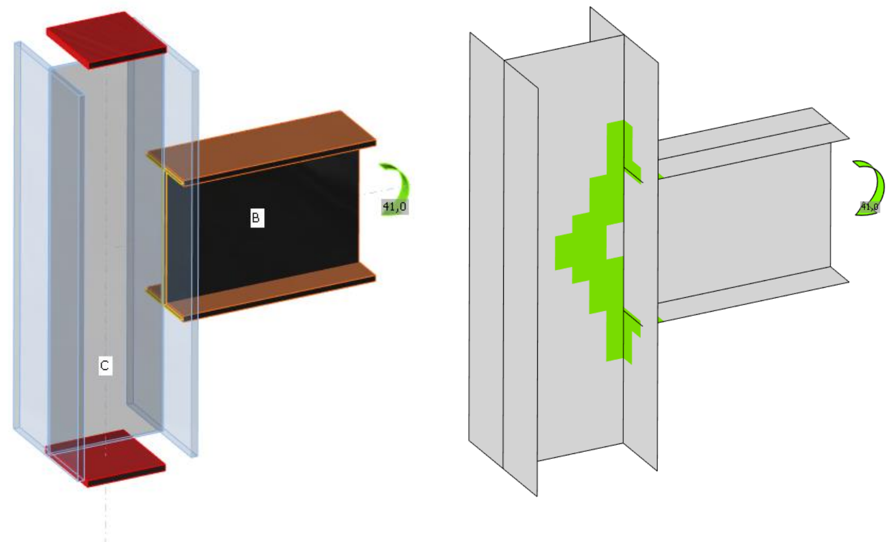
3.3 Equilibrium in node

Each node of 3D FEM model must be in equilibrium. The equilibrium requirement is correct, but it is not necessary for design of simple joints. One member of joint is always „bearing“ and the others are connected. If only the connection of connected members is checked, it is not necessary to keep the equilibrium. Thus there are available two modes of loads input:

- **Simplified** – for this mode the bearing member is supported (continuous member on both sides) and the load is not defined on the member.
- **Advanced** (exact with equilibrium check). The bearing member is supported on one end, the loads are applied on all members and the equilibrium must be found.

The mode can be switched in ribbon group **Advanced mode**.

The difference between modes is shown on following example of T-connection. Beam has end bending moment 41kNm. There is also pressure normal force 100kN in column. In case of simplified mode the normal force is not taken into account because the column is supported on both ends. Program shows only effect of bending moment of beam. Effects of normal force are analysed only in full mode and they are shown in results.



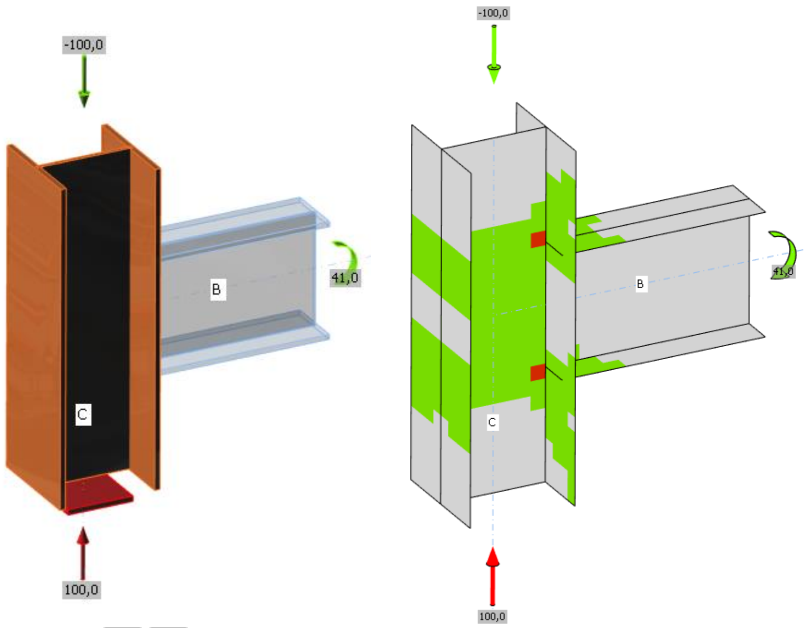
Load effects Import Export

Name	Description
LE1	

Internal forces Clean Copy X position

Member	Position	X [mm]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B	End	0	0,0	0,0	0,0	0,0	41,0	0,0

Simplified input, normal force in column is NOT taken into account



Load effects Import Export

Name	Description
LE1	

Internal forces Clean Copy X position

Member	Position	X [mm]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
C	Begin	0	100,0	0,0	0,0	0,0	0,0	0,0
C	End	0	-100,0	0,0	0,0	0,0	0,0	0,0
B	End	0	0,0	0,0	0,0	0,0	41,0	0,0

Unbalanced forces

X [kN]	Y [kN]	Z [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
0,0	0,0	0,0	0,0	41,0	0,0

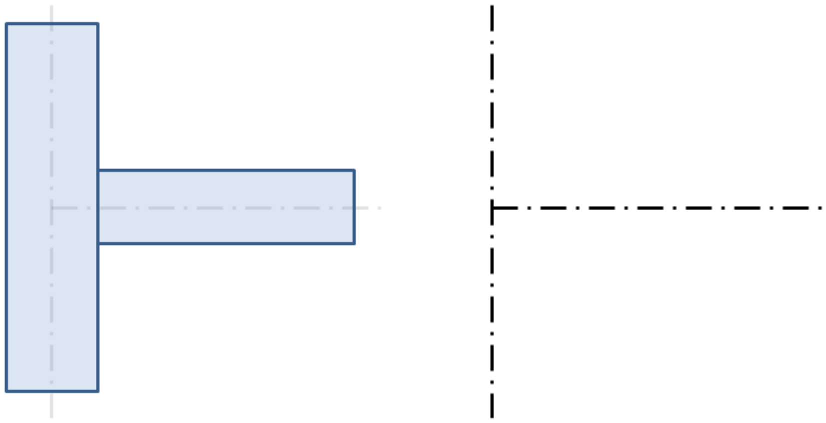
Advanced input, normal force in column is taken into account

Simplified method is easier for user, but it can be used only in case, when user is interested in studying of connection items and not the behaviour of whole joint.
For cases where the bearing member is heavily loaded and close to its limit capacity the advanced mode with respecting of all internal forces in the joint is necessary.

3.4 Loads

End forces of member of the frame analysis model are transferred to the ends of member segments. Eccentricities of members caused by the joint design are respected during transfer.

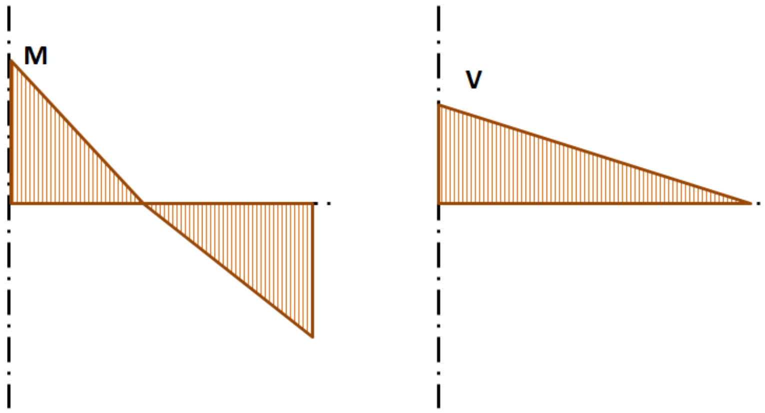
The analysis model created by CBFEM method corresponds to the real joint very precisely, whereas the analysis of internal forces is performed on very idealised 3D FEM 1D model, where individual beams are modelled using centrelines and the joints are modelled using immaterial nodes.



Real shape of joint Theoretical shape in 3D FEM model

Joint of vertical column and horizontal beam

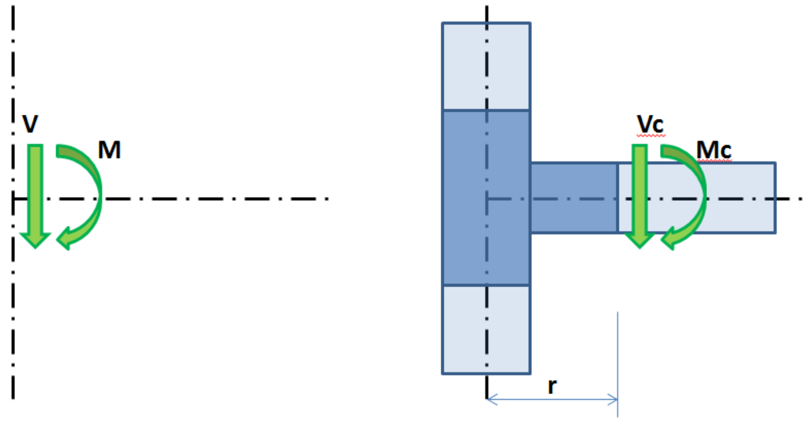
Internal forces are analysed using 1D members in 3D model. There is an example of courses of internal forces in the following picture.



Bending moment Shear force

Course of internal forces on horizontal beam. M and V are the end forces at joint.

The effects caused by member on the joint are important to design the joint (connection). The effects are illustrated in the following picture:



1D member model CBFEM model – dark blue color

Effects of member on the joint. CBFEM model is drawn in dark blue color.

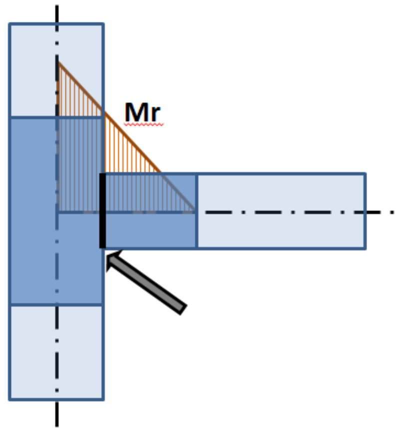
Moment M and force V act in theoretical joint. The point of theoretical joint does not exist ni CBFEM model, thus the load cannot be applied here. The model must be loaded by actions M and V, which have to be transferred to the end of segment in the distance r

Mc = M – V . r

Vc = V

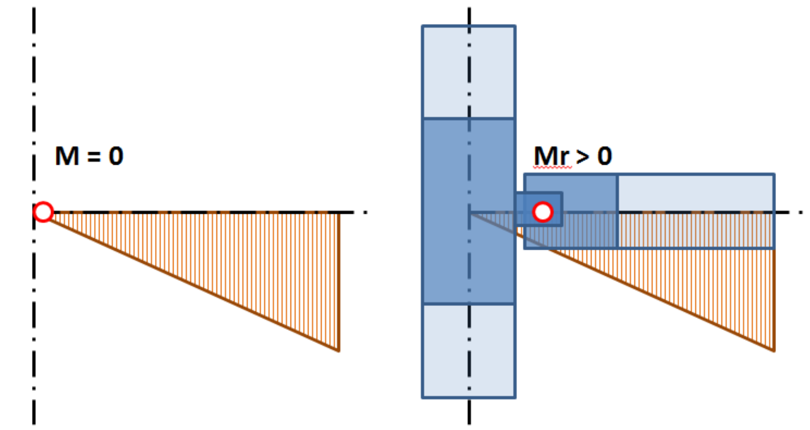
In CBFEM model, the end section of segment is loaded by moment Mc and force Vc.

When designing the joint, its real position relative to the theoretical point of joint must be determined and respected. The internal forces in the position of real joint are mostly different to the internal forces in the theoretical point of joint. Thanks to the precise CBFEM model the design is performed on reduced forces – see moment Mr in the following picture:.



Course of bending moment on CBFEM model. The arrow points to the real position of joint.

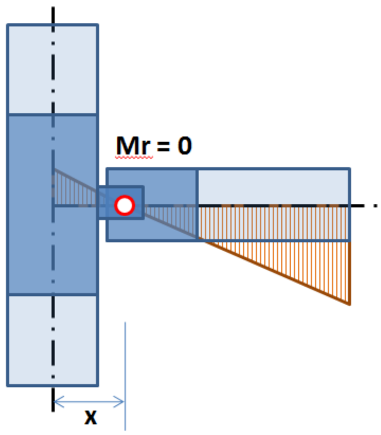
When loading the joint, it must be respected, that the solution of real joint must correspond to the theoretical model used for calculation of internal forces. This is fulfilled for rigid joints, but the situation may be completely different for hinges.



1D members model CBFEM model

Position of hinge in theoretical 3D FEM model and in the real structure

It is illustrated in the previous picture, that the position of hinge in the theoretical 1D members model differs from the real position in the structure. The theoretical model does not correspond to the reality. When applying the calculated internal forces, significant bending moment is applied into the shifted joint and the designed joint is overlarge or cannot be designed either. The solution is simple – both models must correspond. Either the hinge in 1D members model must be defined in the proper position or the courses of internal forces must be shifted to get the zero moment in the position of hinge.



Shifted course of bending moment on beam. Zero moment is at the position of hinge.

The shift of internal force course can be defined in the table for internal forces definition.

Load effects

	Name	Description
>	LE1	

Internal forces Clean Copy X position

	Member	Position	X [m]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
	B	End	0,15	0,0	0,0	-15,0	0,0	25,0	0,0

3.4.1 Import loads from FEA programs

IDEA StatiCa takes during the import from third-party FEA programs calculated results (internal forces, deformations, reactions). Description of load combinations is taken as well. List and content of combinations is shown in wizard (or in BIM application).

FEA programs use to work with envelope combinations. IDEA StatiCa Connection is a program which resolves steel joints nonlinearly (elastic/plastic material model). It means that envelope combinations cannot be used. IDEA StatiCa searches for extremes of internal forces (N, Vy, Vz, Mx, My, Mz) in all combinations at the ends of all members connected to the joint. For each such extreme value are determined also all related internal forces on all remaining members. This set of internal forces is used as a load case for the model of joint in IDEA StatiCa Connection.

User can modify this list of load cases. He can work with combinations in wizard (or BIM) or he can delete some cases directly in IDEA StatiCa Connection.

3.5 Strength analysis

The analysis of joint is non-linear. The load increments are applied gradually and the state of stress is searched. There are two optional analysis modes in IDEA Connection:

- **Response of structure (joint) to the overall load**. All defined load (100%) is applied in this mode and the corresponding state of stress and deformation is calculated.

  Status of FE analysis

	Loads	Applied [%]	Status
>	LE1	100,0	

- **Analysis termination at reaching the ultimate limit state**. The state is found in which all checks of structure are still satisfactory. In the case that the defined load is higher than the calculated capacity, the analysis is marked as non-satisfying and the percentage of used load is printed.

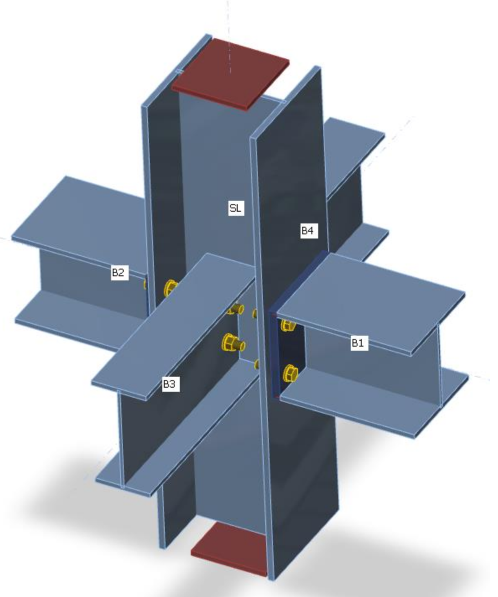
  Status of FE analysis

	Loads	Applied [%]	Status
>	LE1	84,0	

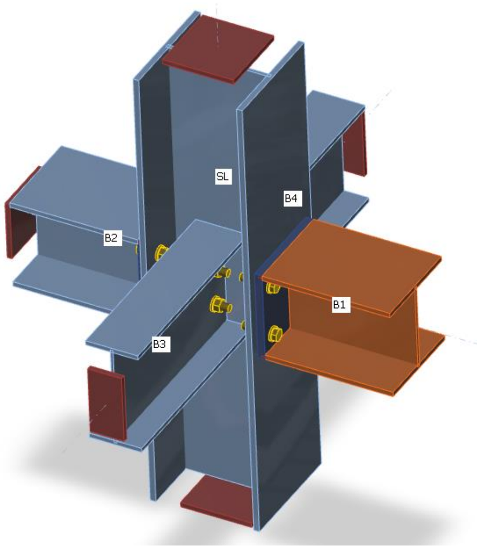
The second mode is more suitable for practical design. The first one is preferable for detailed analysis of complex joints.

3.6 Stiffness analysis

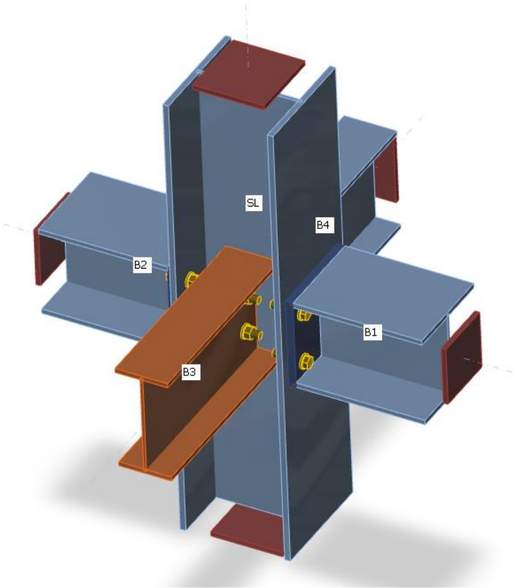
The CBFEM method enables to analyse the stiffness of connection of individual joint members. For the proper stiffness analysis, the separate analysis model must be created for each analysed member. Than the stiffness analysis is not influenced by stiffness of other members of joint, but only by the node itself and the construction of connection of the analysed member. Whereas the bearing member is supported for the strength analysis (SL in), all members except the analysed one are supported for the stiffness analysis (**Errore. L'origine riferimento non è stata trovata.** and **Errore. L'origine riferimento non è stata trovata.** for members B1 and B3).



Supports on members for strength analysis



Supports on members for stiffness analysis of member B1



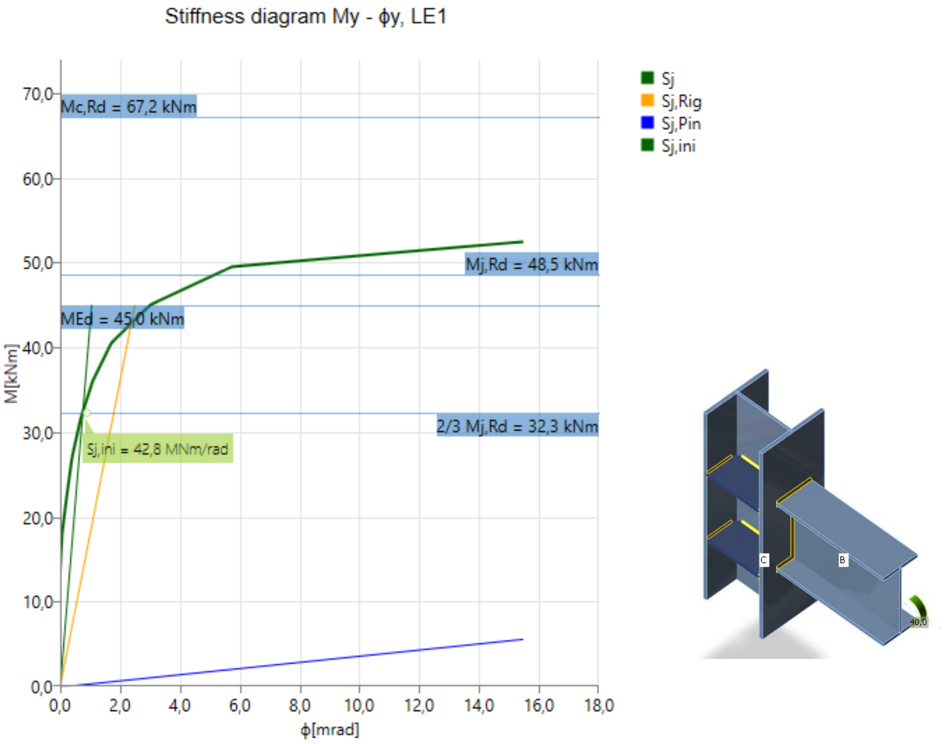
Supports on members for stiffness analysis of member B3

Loads can be applied only on the analysed member. If bending moment My is defined, the rotational stiffness about y-axis is analysed. If bending moment Mz is defined, the rotational stiffness about z-axis is analysed. If axial force is defined, the axial stiffness of connection is analysed.

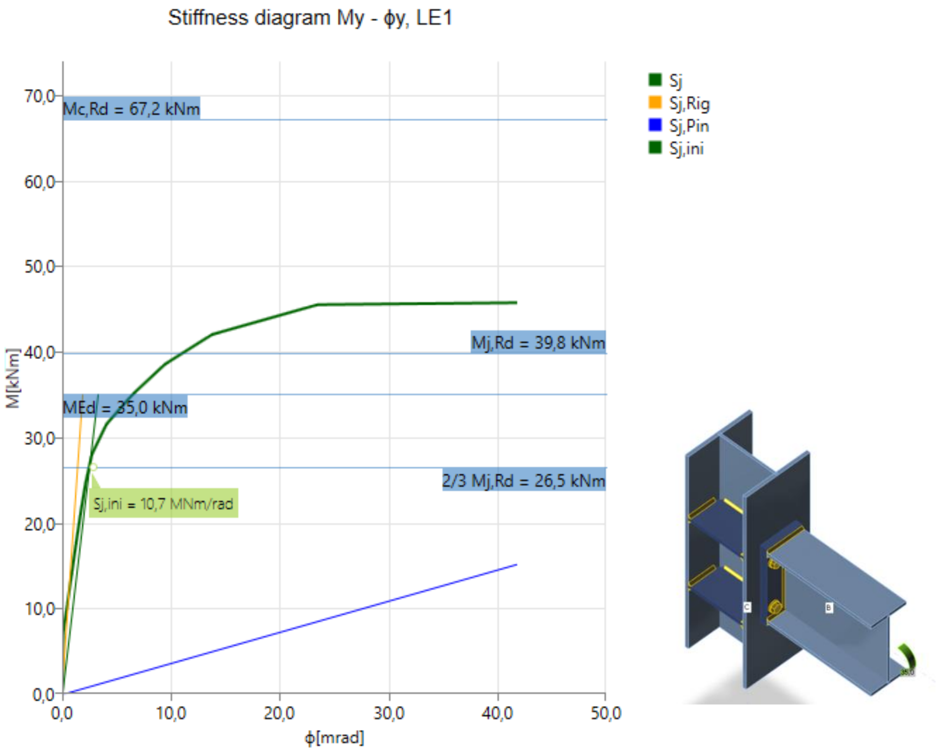
Program generates complete diagram automatically, is directly displayed in GUI and can be added into the output report. Rotational or axial stiffness can be studied for specific design load. IDEA StatiCa Connection can also deal with the influence of other internal forces.

Diagram shows:

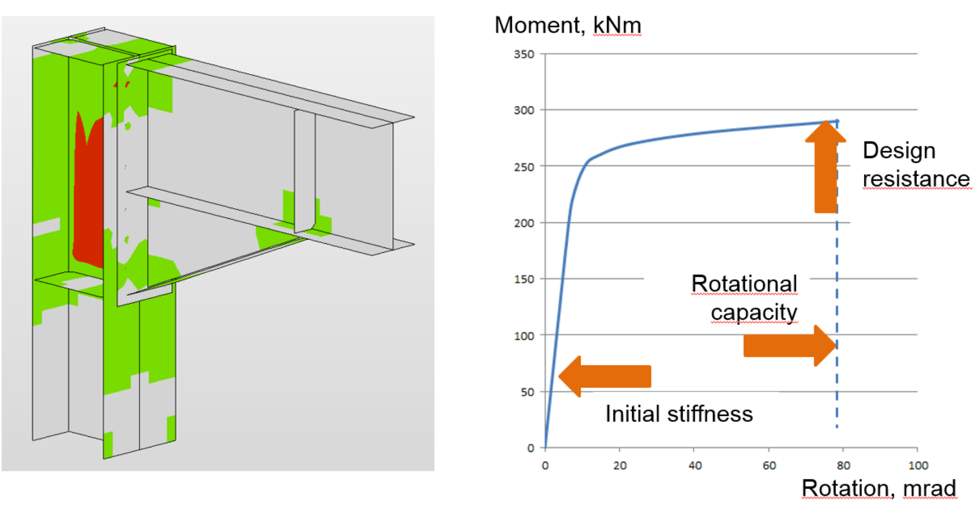
- Level of design load
- Limit value of capacity of connection for 5% equivalent strain
- Limit value of capacity of connected member (useful also for seismic design)
- 2/3 of limit capacity for calculation of initial stiffness
- Value of initial stiffness
- Limits for the classification of connection – rigid and pinned



Rigid welded connection



Semi-rigid bolted connection



After reaching the 5 % strain in the column web panel in shear the plastic zones propagate rapidly

3.7 Stability analysis

The design codes EN 1993-1-5 and EN 1993-1-6 provides five categories of finite element analysis with following assumptions:

1. Linear material, geometrically linear
2. Nonlinear material, geometrically linear
3. Linear material, linear loss of stability – buckling
4. Linear material, geometrically nonlinear using imperfections
5. Nonlinear material, geometrically nonlinear using imperfections

A design procedure, which combines approaches 2 and 3 – material nonlinearity and stability analysis – is mentioned in chapter 8 of EN 1993-1-6. The verification of buckling resistance based on the obtained FEM results is described in Annex B of EN 1993-1-5. This procedure is used for wide range of structures except very slender shells, where geometrically nonlinear analysis with initial imperfections is more suitable (4 and 5) and is currently being implemented into the developed software.

The procedure uses load amplifiers α, which are obtained as results of FEM analysis and allows to predict the post buckling resistance of the joints.

The load coefficient $\alpha_{ult,k}$ is determined, which is caused by reaching of the plastic capacity without considering of the geometrical nonlinearity. The check of plastic capacity has been already implemented into the software. The general automatic determination of $\alpha_{ult,k}$ is currently being implemented into the developed software.

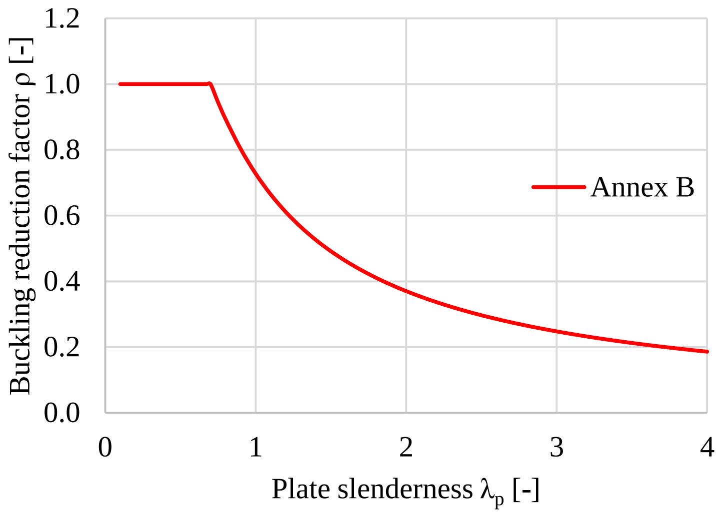
The critical buckling factor α_{cr} is determined, which is obtained using FEM analysis of linear stability. It is determined automatically in the developed software using the same FEM model as for calculation of $\alpha_{ult,k}$. It should be noted that the critical point in terms of plastic resistance is not necessary assessed in first critical buckling mode. More buckling modes need to be assessed in a complex joint, because they are related to different parts of the joint.

The non-dimensional plate slenderness λ_p of the examined buckling mode is determined:

$$\overline{\lambda_p} = \sqrt{\frac{\alpha_{ult,k}}{\alpha_{cr}}} \quad (1)$$

The reduction buckling factor ρ is determined according to Annex B of EN 1993-1-5. The reduction factor is depending on the plate slenderness. The used buckling curve shows the influence of reduction factor on the plate slenderness. The provided buckling factor applicable to non-uniform members is based on the buckling curves of a beam. The verification is based on the von-Mises yield criterion and reduced stress method. Buckling resistance is assessed as:

$$\frac{\alpha_{ult,k} \cdot \rho}{\gamma_{M1}} \geq 1 \quad (2)$$

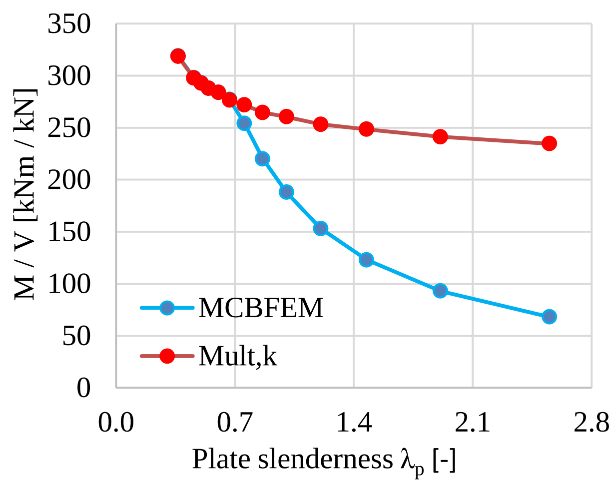


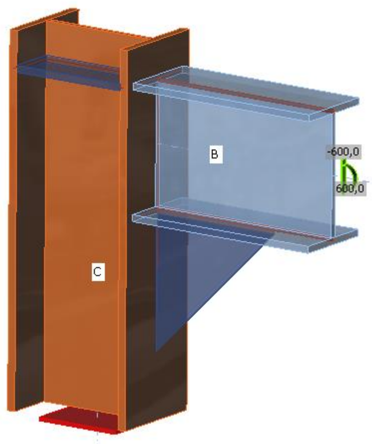
Buckling reduction factor *ρ* according to EN 1993-1-5 Annex B

Although the process seems to be trivial it is general, robust and easily automated. The advantage of the procedure is the advanced FEM analysis of the whole joint, which can be applied to general geometry. Moreover it is included in valid Eurocode standards. The advanced numerical analysis gives quick overview of the global behaviour of the structure and its critical parts and allows fast stiffening to prevent instabilities.
The limit slenderness λ_p is provided in Annex B of EN 1993-1-5 and sets all cases which must be assessed according to previous procedure. The resistance is limited by buckling for plate slenderness higher than 0.7. With the decreasing slenderness is the resistance governed by plastic strain. The limit critical buckling factor for plate slenderness equal to 0.7 and buckling resistance equal to plastic resistance may be obtained as follows:

$$\alpha_{cr} = \frac{\alpha_{ult,k}}{\overline{\lambda_p}^2} = \frac{1}{0{,}7^2} = 2.04$$

It is recommended to check the buckling resistance for critical buckling resistance smaller than 3. The influence of plate slenderness on the plastic resistance $M_{ult,k}$ and buckling resistance M_{CBFEM} is shown in figure bellow. The diagram shows the results of a numerical study of a triangular stiffener in a portal frame joint.





The influence of plate slenderness on the resistance of portal frame joint with slender stiffener

3.8 Deformation capacity

The deformation capacity/ductility δ_{Cd} belongs with resistance and stiffness to the three basic parameters describing the behaviour of connections. In moment resistant connections is achieved the ductility by a sufficient rotation capacity φ_{Cd}. The deformation/rotation capacity is calculated for each connection in the joint separatelly. The prediction of deformation capacity δ_{Cd} of connections is currently studied by component method (CM), but is not offered as standardised procedure. Compare to well accept methods for determination of the initial stiffness and resistance of many types' structural joints, there are no generally accepted standardised procedures for the determination of the rotation capacity. The deemed to satisfy criteria are selected to help the engineers in cl 6.4.2 of EN1993-1-8:2006.

A beam-to-column joint in which the design moment resistance of the joint $M_{j,Rd}$ is governed by the design resistance of the column web panel in shear, may be assumed to have adequate rotation capacity for plastic global analysis, provided that:

$$d/t_w \leq 69\ \varepsilon \qquad (3.11.1)$$

where d the column web panel width, t_w is the web thickness and $\varepsilon \leq \sqrt{235/f_y}$ is the steel yield strength ratio.

In cl 6.4.2(2) is limited the plastic distribution between the bolt rows, for joints with a bolted end-plate connection provided that the design moment resistance of the joint is governed by the design resistance of the column flange or the beam end-plate in bending or the thickness t of either the column flange or the beam end-plate or tension flange cleat satisfies:

$$t \leq 0{,}36\, d \sqrt{f_{ub}/f_y} \qquad (3.11.2)$$

where d and $f_{u.b}$ are the diameter and strength of the bolt and f_y is the yield strength of the relevant plate.

The rotation capacity φ_{Cd} of a welded beam-to-column connection may be assumed to be not less that the value given by the following expression provided that its column web is stiffened in compression but unstiffened in tension, and its design moment resistance is not governed by the design shear resistance of the column web panel, see 6.4.2(1):

$$\varphi_{Cd} = 0{,}025\, h_c / h_b\ ... \qquad (3.11.3)$$

where h_b is the depth of the beam and h_c is the depth of the column. An unstiffened welded beam-to-column joint designed in conformity with the provisions of this section, may be assumed to have a rotation capacity φ_{Cd} of at least 0,015 radians.

The estimation of the rotation capacity is important in connections exposed to seismic, see (Gioncu and Mazzolani, 2002) and (Grecea 2004), and extreme loading, see (Sherbourne AN, Bahaari, 1994 and 1996). The deformation capacity of components has been studied from end of last century (Foley and Vinnakota, 1995). Faella et al (2000) carried out tests on T-stubs and derived for the deformation capacity the analytical expressions. Kuhlmann and Kuhnemund (2000) performed tests on the column web subjected to transverse compression at different levels of compression axial force in the column. Da Silva et al (2002) predicted deformation capacity at different levels of axial force in the connected beam. Based on the test results combined with FE analysis deformation capacities are established for the basic components by analytical models by Beg et al (2004). In the work are represented components by non-linear springs, and appropriately combined in order to determine the rotation capacity of the joint for the end-plate connections, with an extended or flush end-plate, and welded connections. For these connections, the most important components that may significantly contribute to the rotation capacity column were recognised as the web in compression, column web in tension, column web in shear, column flange in bending, and end-plate in bending. Components related to the column web are relevant only when there are no stiffeners in the column that resist compression, tension or shear forces. The presence of a stiffener eliminates the corresponding component, and its contribution to the rotation capacity of the joint can be therefore neglected. End-plates and column flanges are important only for end-plate connections, where the components act as a T-stub, where also the deformation capacity of the bolts in tension is included. The questions and limits of deformation capacity of connections of high strength steel was studied by Girao at al (2004).

4 Check of components according to Eurocode

CBFEM method combines advantages of general finite elements method and standard method of components. The stresses and internal forces calculated on the accurate CBFEM model are used in checks of all components.

Individual components are checked according to Eurocode EN 1993-1-8.

4.1 Plates

The resulting equivalent stress (HMH, von Mieses) and plastic strain are calculated on plates. The stress check cannot be performed, because the stress reaches the yield strength only. Thus the check of equivalent plastic strain is performed. The limit value 5% is suggested in Eurocode (EN1993-1-5 app. C par. C8 note 1), this value can be modified in project settings.

Plate element is divided to 7 layers and elastic/plastic behaviour is investigated in each of them. Program shows the worst result from all of them.

Check of members and steel plates

Item	Th [mm]	Loads	σ,Ed [MPa]	ε,Pl [%]	Status
COL-bfl	21,5	LE1	222,6	0,1	
COL-tfl	21,5	LE1	155,0	0,0	
COL-web	12,0	LE1	106,1	0,0	
BP1	30,0	LE1	159,6	0,0	

Design data

Material	Fy[MPa]	ε,lim [%]
S 235	235,0	5,0

CBFEM method can provide stress rather higher than yield strength. The reason is the slight inclination of plastic branch of stress-strain diagram, which is used in analysis to improve the stability of interaction calculation. This is not problem for practical design. The equivalent plastic strain is exceeded at higher stress and the joint does not satisfy anyway.

4.2 Welds

4.2.1 Fillet welds

Design resistance

The stress in the throat section of fillet weld is determined according to art. 4.5.3. Stresses are calculated from shear forces in weld links. Bending moment round the weld longitudinal axis is not taken into account.

$\sigma_{w,Ed} = [\sigma_\perp^2 + 3\,(\tau_\perp^2 + \tau_{||}^2)]^{0.5}$

$\sigma_{w,Rd} = f_u / (\beta_W \ \gamma_{M2})$

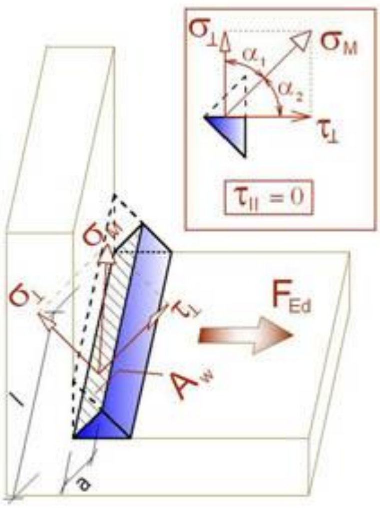
$0.9_{\sigma w,Rd} = f_u / \gamma_{M2}$

Weld utilisation

$U_t = \min(\sigma_{w,Ed}/\sigma_{w,Rd}, \sigma_\perp/0.9\sigma_{w,Rd})$

where:

- β_W - correlation factor tab 4.1



Stresses in the weld

All values required for check are printed in tables.

Check of welds for extreme load effect (Average stress)

Item	Edge	Th [mm]	L[mm]	Loads	σw,Ed [MPa]	σ⊥ [MPa]	τ|| [MPa]	τ⊥ [MPa]	Ut [%]	Status
EP1	C-bfl 1	◢5,9◣	166	LE1	12,9	5,7	0,0	6,7	3,6	✓
EP1	C-tfl 1	◢5,9◣	166	LE1	55,8	-27,0	0,0	28,2	15,5	✓
EP1	C-w 1	◢3,4◣	295	LE1	76,4	-38,1	2,4	38,2	21,2	✓
SPL1	B-bfl 1	◢10,7	126	LE1	9,3	2,7	0,0	5,1	2,6	✓
SPL1	B-tfl 1	◢10,7	126	LE1	3,9	-0,6	0,0	-2,2	1,1	✓
SPL1	B-w 1	◢6,6	343	LE1	46,7	-0,7	27,0	-0,7	13,0	✓

Design data

Material	βw	σw,Rd [MPa]	0.9 σw,Rd [MPa]
S 235	0,8	360,0	259,2

4.2.2 Butt welds

Welds can be specified as butt welds. Complete joint penetration is considered for butt welds, thus such weld are not checked.

4.3 Bolts

Design tension resistance of bolt:

$F_{tRd} = 0.9\ f_{ub}\ A_s / \gamma_{M2}$

Design shear resistance at punching of bolt head or nut EN 1993-1-8:

$B_{pRd} = 0.6\ \pi\ d_m\ t_p\ f_u / \gamma_{M2}$

Design shear resistance per one shear plane:

$F_{vRd} = \alpha_v\ f_{ub}\ A_s / \gamma_{M2}$

Design bearing resistance of plate EN 1993-1-8:

$F_{bRd} = k_1\ a_b\ f_u\ d\ t / \gamma_{M2}$

Utilisation in tension [%]:

$U_{tt} = F_{tEd} / \min (F_{tRd,} B_{pRd})$

Utilisation in shear [%]:

$U_{ts} = V / \min (F_{vRd,} F_{bRd})$

Interaction of shear and tension [%]:

$U_{tts} = V / F_{vRd} + F_{tEd} / 1.4 F_{tRd}$

where:

- A_s - tensile stress area of the bolt,
- f_{ub} - ultimate tensile strength,
- d_m - bolt head diameter,
- d - bolt diameter,
- t_p - plate thickness under the bolt head/nut,
- f_u - ultimate steel strength,
- α_v = 0,6 for classes (4.6, 5.6, 8.8) ,
- α_v = 0,5 for classes (4.8, 5.8, 6.8, 10,9),
- k_1 = 2.5,
- a_b = 1.0,
- F_{tEd} - design tensile force in bolt,
- V - resultant of shear forces in bolt.

Check of bolts and anchors

Item	Material	Loads	Ft,Ed [kN]	V [kN]	Utt [%]	Uts [%]	Utts [%]	Status
B1	M16 - 8.8 - 1	LE1	4,285	12,585	4,7	20,9	24,3	✓
B2	M16 - 8.8 - 1	LE1	4,277	12,581	4,7	20,9	24,2	✓
B3	M16 - 8.8 - 1	LE1	2,037	12,425	2,3	20,6	22,2	✓
B4	M16 - 8.8 - 1	LE1	2,212	12,411	2,4	20,6	22,3	✓
B5	M12 - 6.8 - 2	LE1	0,963	4,757	2,7	23,6	25,5	✓
B6	M12 - 6.8 - 2	LE1	0,972	4,758	2,7	23,6	25,5	✓
B7	M12 - 6.8 - 2	LE1	0,224	4,745	0,6	23,5	24,0	✓
B8	M12 - 6.8 - 2	LE1	0,253	4,740	0,7	23,5	24,0	✓

Design data

Item	Ft,Rd [kN]	Bp,Rd [kN]	Fv,Rd [kN]	Fb,Rd [kN]
M16 - 8.8 - 1	90,432	130,288	60,288	115,200
M12 - 6.8 - 2	36,288	103,145	20,160	86,400

4.4 Preloaded bolts

The design slip resistance of a preloaded class 8.8 or 10.9 bolt is subjected to an applied tensile force, $F_{t,Ed}$

Preloading force to be used EN 1993-1-8 3.9 (3.7)
$F_{p,C}$ = 0,7 fub As

Design slip resistance per bolt EN 1993-1-8 3.9 (3.8)
$F_{s,,Rd} = ks\ \ n\ \ \mu\ (F_{p,C} – 0{,}8\ F_{t,Ed})\ /\ \gamma\ M_3$

Utilisation in shear [%]:
$U_{ts} = V / F_{s,,Rd}$ where

- A_s - tensile stress area of the bolt,
- f_{ub} - ultimate tensile strength,
- k_s – coefficient given in Table 3.6; k_s = 1,
- μ - slip factor obtained,
- n - number of the friction surfaces. Check is calculated for each friction surface separately,
- $\gamma\ M_3$ - safety factor,
- V - shear force,
- $F_{t,Ed}$ - design tensile force in bolt.

4.5 Anchors

Concrete cone failure resistance of anchor or group of anchors ETAG-001 5.2.2.4:

$N_{Rkc} = N^0_{Rkc} \ A_{cN} / A^0_{cN} \ \Psi_{sN} \ \Psi_{reN}$

Initial value of characteristic resistance:

$N^0_{Rkc} = 7.2 \ f_{ck}^{0.5} \ hef^{1.5}$

where:

- A^0_{cN} - area of concrete cone of an individual anchor. Circle of radius 1.5 * hef,
- hef - length of anchor in concrete,
- f_{ck} - characteristic concrete compressive strength,
- A_{cN} - actual area of concrete cone of the anchorage at the concrete surface respecting influence of edges and adjoining anchors,
- $\Psi_{sN} = 1$,
- $\Psi_{reN} = 1$.

Anchors shear resistance in case of transfer of shear forces. Friction is not taken into account. Valid in case, that the anchor failure precedes the concrete failure ETAG-001 5.2.3.2:

$V_{Rks} = 0.5 \ f_y \ A_s$

Concrete pry-out failure ETAG-001 5.2.3.3:

$V_{Rkcp} / \gamma_{Mc} <= V$

$V_{Rkcp} = k * N_{Rkc}$

where:

- V - shear force,
- k = 1 for hef < 60,
- k = 2 for hef >= 60.

Concrete edge failure ETAG-001 5.2.3.4:

$V_{Rkc} / \gamma_{Mc} <= V$

$V_{Rkc} = V^0_{Rkc}\ A_{cV}/ A^0_{cV}\ \Psi_{sV}\ \Psi_{reV}$

$V^0_{Rkc} = 1.7\ d^{\alpha}\ lf^{\beta}\ f_{ck}^{0.5}\ c_1^{1.5}$

$\alpha = 0.1\ (lf / c_1)^{0.5}$

$\beta = 0.1\ (d / c_1)^{0.2}$

where:

- lf = hef,
- c_1 - edge distance,
- d - anchor diameter,
- $\Psi_{sV} = 1$,
- $\Psi_{reV} = 1$,
- A^0_{cV} - area of concrete cone of an individual anchor at the lateral concrete surface not affected by edges ($4.5\ c_1^2$),
- A_{cV} - actual area of concrete cone of anchorage at the lateral concrete surface.

Check of bolts and anchors

	Item	Loads	Ft,Ed [kN]	Nrkc [kN]	Utt [%]	Status
>	B1		0,000	0,000	0,0	✓
	B2		0,000	0,000	0,0	✓
	B3	LE1	81,329	87,670	92,8	✓
	B4	LE1	20,507	87,670	23,4	✓

Design data

	Item	Ft,Rd [kN]	Bp,Rd [kN]	Fv,Rd [kN]	Fb,Rd [kN]
>	M22 - 10.9 - 1	185,436	553,725	121,200	475,200

4.6 Concrete block

Concrete resistance at concentrated compression:

$F_{jd} = \beta j \;\; kj \;\; fck \,/\, \gamma_C$

Average stress under the base plate:

$\sigma = N \,/\, A_{eff}$

Utilisation in compression [%]

$U_t = \sigma \,/\, F_{jd}$

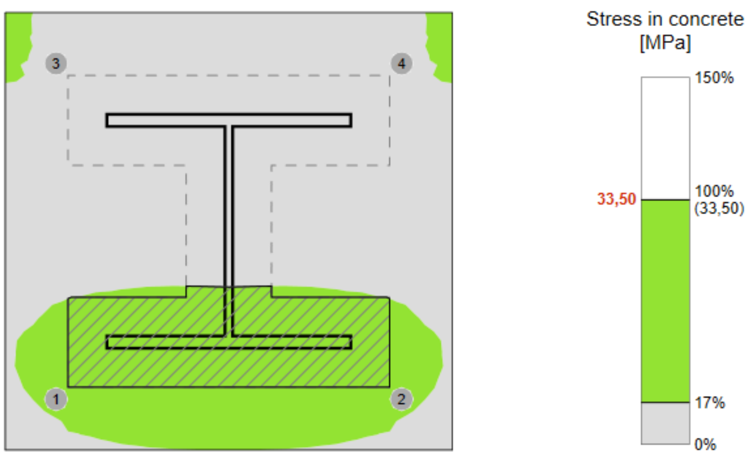
where:

- f_{ck} - characteristic compressive concrete strength,
- $\beta j = 0.67$,
- kj - concentration factor,
- γ_C - safety factor,
- A_{eff} - effective area, on which the column force N is distributed.

Check of contact stress in concrete for extreme load effect

	Item	Loads	c [mm]	Aeff [m2]	σ [MPa]	Kj	Fjd [MPa]	Ut [%]	Status
>	C25/30	LE1	38	0,03	5,7	3,00	33,5	17,2	✓

Effective area is calculated according to the real course of contact stress and assumptions defined in Eurocode. Graphical representation shows the way of checking. Calculated effective area is marked as green. Final effective area for contact stress check is highlighted as shaded.



Effective area of contact stress

4.7 Shear in concrete block

Shear forces are evaluated in this table only in case of shear transfer by friction or shear iron.

1. Shear is transferred only by friction

   $V_{Rdy} = N\ C_f$

   $V_{Rdz} = N\ C_f$

2. Shear is transferred by shear iron and friction

   $V_{Rdy} = N\ C_f + A_{vy}\ f_y / (3^{0.5}\ \gamma_{M0})$

   $V_{Rdz} = N\ C_f + A_{vz}\ f_y / (3^{0.5}\ \gamma_{M0})$

Utilisation in shear [%]

$U_t = \min(V_y/V_{Rdy}, V_z/V_{Rdz})$

where:

- A_{vy} - shear area Ay of shear iron cross-section,
- A_{vz} - shear area Az of shear iron cross-section,
- f_y - yield strength,
- γ_{M0} - safety factor,
- V_y - shear force component in the base plate plane in y-direction,
- V_z - shear force component in the base plate plane in z-direction,
- N - force perpendicular to the base plate,
- C_f - friction coefficient.

Shear in contact plane

Item	Loads	Vy [kN]	Vz [kN]	Vrdy [kN]	Vrdz [kN]	Ut [%]	Status
BP1	LE1	14,900	-29,739	384,560	662,947	4,5	✓

Design data

Friction	Css
0,25	HEB140

5 Check of components according to AISC

CBFEM method combines advantages of general finite elements method and standard method of components. The stresses and internal forces calculated on the accurate CBFEM model are used in checks of all components.

Individual components are checked according to AISC 360-10

5.1 Plates

The resulting equivalent stress (HMH, von Mieses) and plastic strain are calculated on plates. The stress check cannot be performed, because the stress reaches the yield strength only. Thus the check of equivalent plastic strain is performed. The limit value 5% is suggested in Eurocode (EN1993-1-5 app. C par. C8 note 1), this value can be modified in project settings.

Plate element is divided to 7 layers and elastic/plastic behaviour is investigated in each of them. Program shows the worst result from all of them.

Check of members and steel plates for extreme load effect

	Item	Th [mm]	Loads	σ,Ed [MPa]	ε,Pl [%]	Status
>	C-bfl 1	10,0	LE1	248,4	0,08	✓
	C-tfl 1	10,0	LE1	22,1	0,00	✓
	C-w 1	6,5	LE1	213,8	0,01	✓
	B-bfl 1	9,2	LE1	139,2	0,00	✓
	B-tfl 1	9,2	LE1	153,7	0,00	✓
	B-w 1	5,9	LE1	125,3	0,00	✓
	EP1	10,0	LE1	225,3	0,02	✓

Design data

	Material	Fy[MPa]	ε,lim [%]
>	A36	248,2	5,0

CBFEM method can provide stress rather higher than yield strength. The reason is the slight inclination of plastic branch of stress-strain diagram, which is used in analysis to improve the stability of interaction calculation. This is not problem for practical design. The equivalent plastic strain is exceeded at higher stress and the joint does not satisfy anyway.

5.2 Welds

5.2.1 Fillet welds

The **design strength**, ΦRn and the **allowable strength**, Rn/Ω of welded joints are evaluated in connection weld check.

$\Phi = 0.75$ (LRFD)

$\Omega = 2.00$ (ASD)

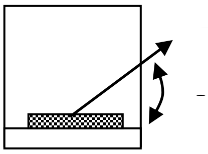
Available strength of welded joints is evaluated according to AISC 360-10 table J2,5

$Rn = Fnw \ Awe$

$Fnw = 0.60FEXX\ (1.0 + 0.50 \sin 1.5\ \Theta)$

where:

- Fnw - nominal stress of weld material,
- Awe - effective area of the weld,
- FEXX - electrode classification number, i.e., minimum specified tensile strength,
- Θ - angle of loading measured from the weld longitudinal axis, degrees.



For end-loaded fillet welds with a length up to 100 times the weld size, it is permitted to take the effective length equal to the actual length. When the length of the end-loaded fillet weld exceeds 100 times the weld size, the effective length shall be determined by multiplying the actual length by the reduction factor, β, determined as follows:

$\beta = 1.2 - 0.002\ (l / w)$

where:

- l - weld length,
- w - size of weld leg.

When the length of the weld exceeds 300 times the leg size, *w*, the effective length

is taken as 180*w*.

All values required for check are printed in tables.

Check of welds for extreme load effect (Average stress)

	Item	Edge	Material	Th [mm]	Ls [mm]	L [mm]	Loads	Fn [kN]	Rn/Ω [kN]	Ut [%]	Status
>	C-bfl 1	B1-bfl 1	E60xx	◢5,0◣	◢7,1◣	110,0	LE1	158,5	204,4	77,5	✔
	C-bfl 1	B1-tfl 1	E60xx	◢5,0◣	◢7,1◣	110,0	LE1	158,5	204,4	77,5	✔
	C-bfl 1	B1-w 1	E60xx	◢4,0◣	◢5,7◣	210,8	LE1	38,1	115,5	32,9	✔
	C-bfl 1	VÝZT1a	E60xx	◢5,0◣	◢7,1◣	78,8	LE1	58,7	146,6	40,1	✔
	C-w 1	VÝZT1a	E60xx	◢3,3◣	◢4,6◣	134,0	LE1	12,9	20,5	62,9	✔
	C-tfl 1	VÝZT1a	E60xx	◢5,0◣	◢7,1◣	78,8	LE1	12,0	146,5	8,2	✔
	C-bfl 1	VÝZT1b	E60xx	◢5,0◣	◢7,1◣	78,7	LE1	58,8	146,6	40,1	✔
	C-w 1	VÝZT1b	E60xx	◢3,3◣	◢4,6◣	134,0	LE1	12,9	20,5	62,9	✔
	C-tfl 1	VÝZT1b	E60xx	◢5,0◣	◢7,1◣	78,7	LE1	12,0	146,5	8,2	✔
	C-bfl 1	VÝZT1c	E60xx	◢5,0◣	◢7,1◣	78,8	LE1	58,7	146,6	40,1	✔
	C-w 1	VÝZT1c	E60xx	◢3,3◣	◢4,6◣	134,0	LE1	12,9	20,5	62,9	✔
	C-tfl 1	VÝZT1c	E60xx	◢5,0◣	◢7,1◣	78,8	LE1	12,0	146,5	8,2	✔
	C-bfl 1	VÝZT1d	E60xx	◢5,0◣	◢7,1◣	78,7	LE1	58,8	146,6	40,1	✔
	C-w 1	VÝZT1d	E60xx	◢3,3◣	◢4,6◣	134,0	LE1	12,9	20,5	62,9	✔
	C-tfl 1	VÝZT1d	E60xx	◢5,0◣	◢7,1◣	78,7	LE1	12,0	146,5	8,2	✔

5.2.2 CJP groove welds

AISC Specification Table J2.5 identifies four loading conditions that might be associated with JP groove welds, and shows that the strength of the joint is either controlled by the base metal or that the loads need not be considered in the design of the welds connecting the parts. Accordingly, when CJP groove welds are made with matching-strength filler metal, the strength of a connection is governed or controlled by the base metal, and no checks on the weld strength are required.

5.3 Bolts

5.3.1 Tensile and shear strength of bolts

The design tensile or shear strength, ΦRn, and the allowable tensile or shear strength, Rn/Ω of a snug-tightened bolt is determined according to the limit states of tension rupture and shear rupture as follows:

Rn = Fn Ab

Φ = 0.75 (LRFD)

Ω = 2.00 (ASD)

where:

- Ab - nominal unthreaded body area of bolt or threaded part, in^2 (mm^2)
- Fn - nominal tensile stress, Fnt, or shear stress, Fnv, from Table J3.2, ksi (MPa)

The required tensile strength includes any tension resulting from prying action produced by deformation of the connected parts.

5.3.2 Combined Tension and shear in bearing type connection

The available tensile strength of a bolt subjected to combined tension and shear is determined according to the limit states of tension and shear rupture as follows:

Rn = F'nt Ab (AISC 360-10 J3-2)

Φ = 0.75 (LRFD)

Ω = 2.00 (ASD)

Fn't = 1,3Fnt - frv Fnt/ ΦFnv (AISC 360-10 J3-3a LRFD)

Fn’t = 1,3Fnt - frv Ω Fnt/Fnv (AISC 360-10 J3-3b ASD)

where:

- Fn’t - nominal tensile stress modified to include the effects of shear stress
- Fnt - nominal tensile stress from AISC 360-10 Table J3.2
- Fnv - nominal shear stress from AISC 360-10 Table J3.2
- frv - required shear stress using LRFD or ASD load combinations. The available shear stress of the fastener shall be equal or exceed the required shear stress, frv.

5.3.3 Bearing strength in bolt holes

The available bearing strength, ΦRn and Rn/Ω at bolt holes is determined for the limit state of bearing as follows:

Φ = 0.75 (LRFD)

Ω = 2.00 (ASD)

The nominal bearing strength of the connected material, Rn, is determined as follows:

For a bolt in a connection with standard, oversized and short-slotted holes, independent of the direction of loading, or a long-slotted hole with the slot parallel to the direction of the bearing force

When deformation at the bolt hole at service load is a design consideration

$Rn = 1.2\ lc\ t\ Fu \leq 2.4\ d\ t\ Fu$ (AISC 360-10 J3-6a)

When deformation at the bolt hole at service load is not a design consideration

$Rn = 1.5\ lc\ t\ Fu \leq 3.0\ d\ t\ Fu$ (AISC 360-10 J3-6b)

where:

- Fu - specified minimum tensile strength of the connected material,
- d - nominal bolt diameter,
- lc - clear distance, in the direction of the force, between the edge of the hole and the edge of the adjacent hole or edge of the material,
- t - thickness of connected material.

5.4 Preloaded bolts

The design slip resistance of a preloaded class A325 or A490 bolt without of effect of tensile force $F_{t,Ed}$

Preloading force to be used AISC 360-10 tab. J3.1.
$Tb = 0{,}7\ fub\ As$

Design slip resistance per bolt AISC 360-10 par. 3.8
$Rn = 1.13\ \mu\ Tb\ Ns$

Utilisation in shear [%]:
$U_{ts} = V / Rn$

- A_s - tensile stress area of the bolt,
- f_{ub} - ultimate tensile strength,
- μ - slip factor obtained,
- Ns - number of the friction surfaces. Check is calculated for each friction surface separately.
- V - shear force.

5.5 Anchors

5.5.1 Concrete cone pull out strength Appendix D of ACI 318-02

Concrete Capacity Design (CCD). In the CCD method, the concrete cone is considered to be formed at an angle of approximately 34° (1 to 1.5 slope). For simplification, the cone is considered to be square rather than round in plan. The concrete breakout stress in the CCD method is considered to decrease with an increase in size of the breakout surface. Consequently, the increase in strength of the breakout in the CCD method is proportional to the embedment depth to the power of 1.5

$\Phi\ N_{cbg} = \Phi\ \psi_3\ 24\ \sqrt{fc}\ \ hef^{1,5}\ A_n/A_{n0}$ for $h_{ef} < 11$ in

$\Phi\ N_{cbg} = \Phi\ \psi_3\ 16\ \sqrt{fc}\ \ hef^{1,66}\ A_n/A_{n0}$ for $h_{ef} >= 11$ in

where:

- $\Phi = 0.70$,
- $\psi_3 = 1.25$ considering the concrete to be uncracked at service loads, otherwise =1.0,
- hef - depth of embedment,
- A_n - concrete breakout cone area for group,
- A_{n0} - concrete breakout cone area for single anchor.