COMPONENT BASED FINITE ELEMENT METHOD

In the following unit, we will shortly introduce the elementary background of the Component-Based Finite Element Method (CBFEM) – the theory behind applications like IDEA StatiCa Connection, HILTI PROFIS Engineering, Advance Design Connection and others.

Design codes, e.g. EN1993-1-8, and also technical literature offer engineering solution methods. Their general feature is derivation for typical structural shapes and simple loadings. The method of components is used very often.

Component method (CM) solves the joint as a system of interconnected items – components. The corresponding model is built per each joint type to be able to determine forces and stresses in each component.

As can be seen in the following picture, each component is checked separately using corresponding formulas. As the proper model must be created for each joint type, the method usage has limits when solving joints of general shapes and general loads.

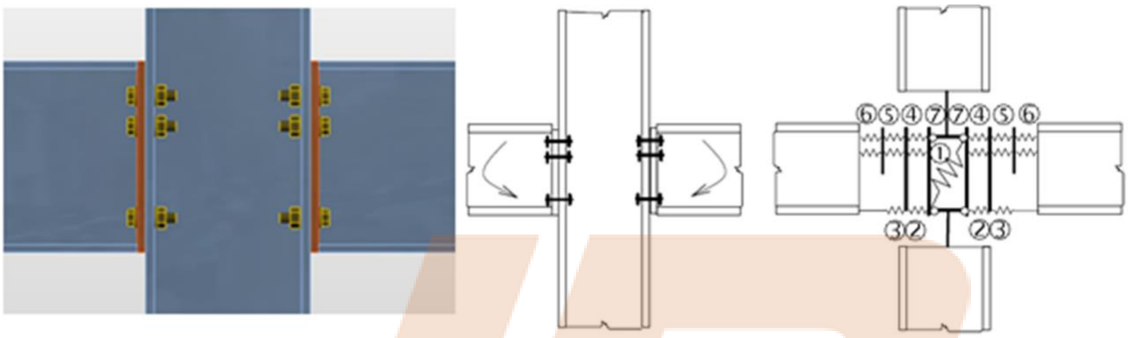

Figure 0.1. The components of a joint with bolted end plates modeled by springs.

Therefore, IDEA StatiCa together with a project team of Czech Technical University in Prague and Brno University of Technology developed a new method for advanced design of steel structural joints.

The new **Component Based Finite Element Model (CBFEM)** method is:

- **General** enough to be usable for most of joints, footings and details in engineering practice.
- **Simple and fast** enough in daily practice to provide results in a time comparable to current methods and tools.
- **Comprehensive** enough to provide structural engineer clear information about joint behavior, stress, strain and reserves of individual components and about overall safety and reliability.

The CBFEM method is based on the idea that the most of the verified and very useful parts of CM should be kept. The weak point of CM – its generality when analyzing stresses of individual components – was replaced by modeling and analysis using Finite Element Method (FEM).

CBFEM COMPONENTS

FEM is a general method commonly used for structural analysis. Usage of FEM for modeling of joints of any shapes seems to be ideal (Virdi, 1999). The elastic-plastic analysis is required, as the steel ordinarily yields in the structure. In fact, the results of the linear analysis are useless for joint design.

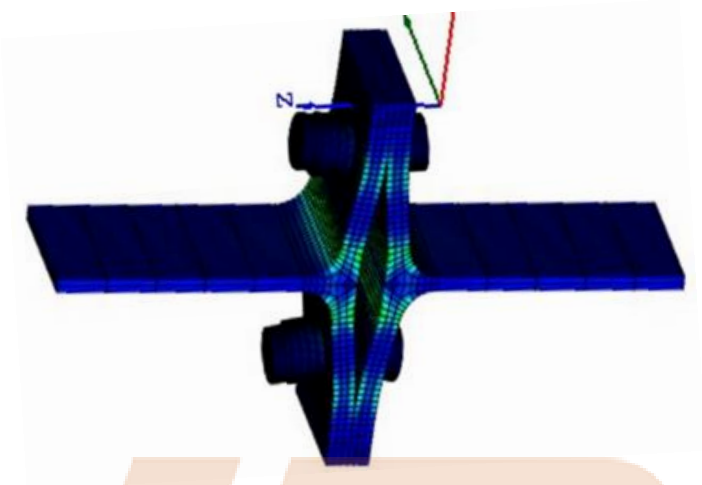

Figure 0.2. FEM model of a joint for research. It uses spatial 3D elements for both plates and bolts.

Both webs and flanges of connected members are modeled using shell elements 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. Modeling of such elements in general FEM programs is difficult because the programs do not offer required properties. Thus, special FEM components had to be developed to model the welds and bolts behaviour in a joint.

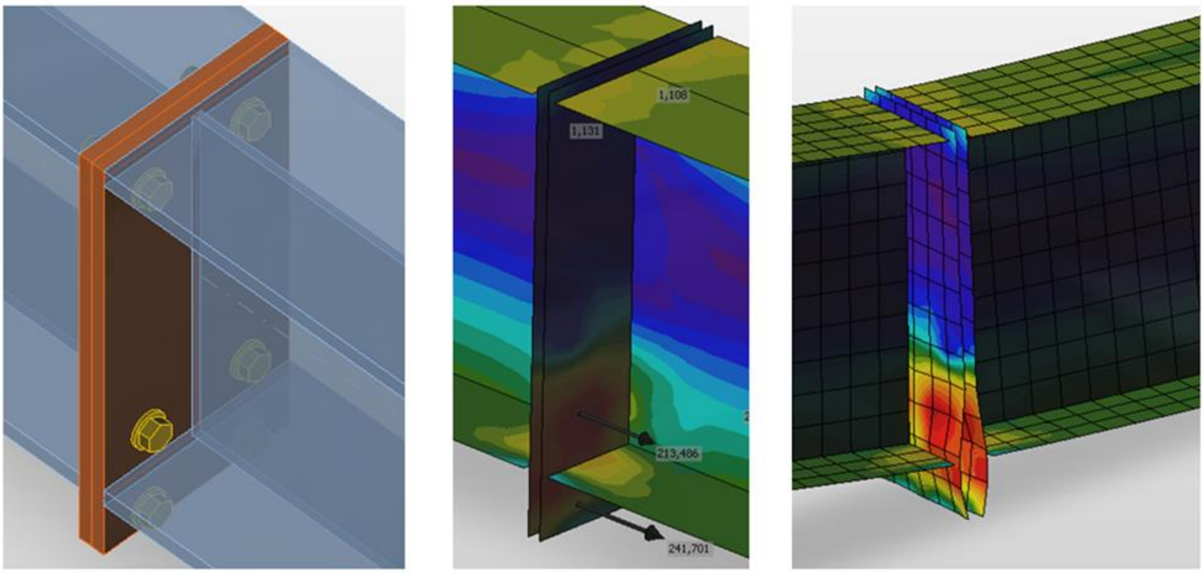

Figure 0.3. CBFEM model of bolted connection by end plates.

Joints of members are modeled as massless points when analyzing steel frame or girder structure. The resultant of forces from all members in the joint is zero – the whole joint is in equilibrium.

The real shape of a joint is not known in the structural model. The engineer only defines whether 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. The ends of members with the length of a 2–3 multiple of maximal cross-section height are used in the CBFEM method. These segments are modeled using shell elements.

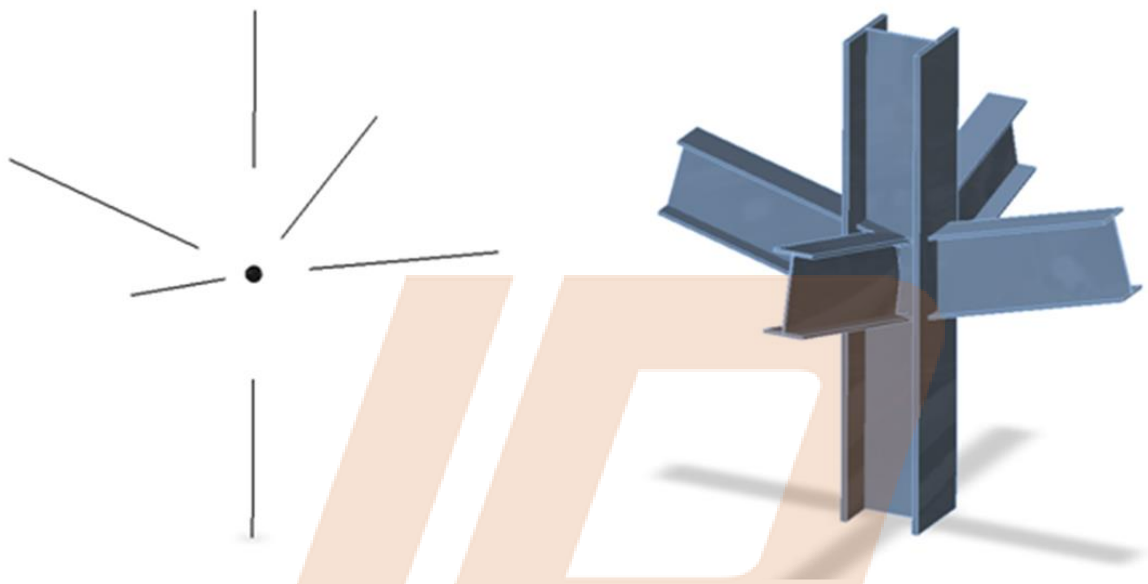

Figure 0.4. A theoretical (massless) joint and a real shape of the joint without modified member ends.

For better precision of CBFEM model, the end forces on 1D members are applied as loads on the 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.

CBFEM ANALYSIS

The newly developed method (CBFEM – Component Based Finite Element Model) enables fast analysis of joints of several shapes and configurations. The model consists of members, to which the load is applied, and manufacturing operations (including stiffening members), which serve to connect members to each other. The analyzed FEM model is generated automatically. The designer does not create the FEM model, he creates the joint using manufacturing operations.

MATERIAL AND PLATE MODEL. FINITE ELEMENT MESH

MATERIAL MODEL

The plates in IDEA StatiCa Connection are modelled with elastic-plastic material with a nominal yielding plateau slope according to EN1993-1-5, Par. C.6, (2), $\tan^{-1}(E/1000)$. The material behaviour is based on von Mises yield criterion. It is assumed to be elastic before reaching the design yield strength, f_{yd}.

The ultimate limit state criterion for regions not susceptible to buckling is reaching the limiting value of the principal membrane strain. The value of 5% is recommended (e.g. EN1993-1-5, App. C, Par. C.8, Note 1).

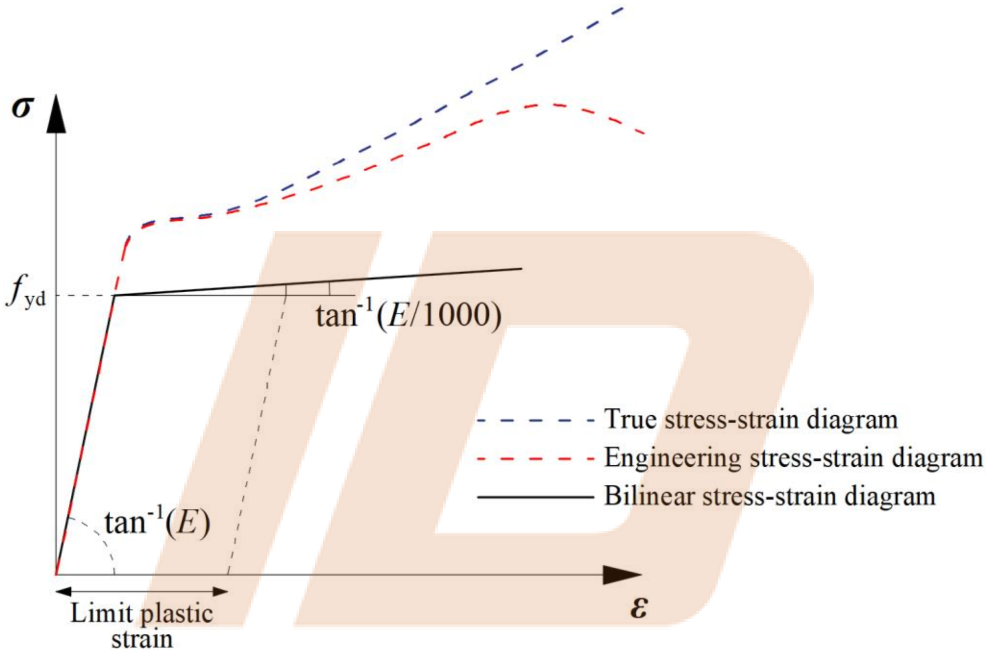

Figure 0.5. Material diagrams of steel in numerical models.

PLATE MODEL AND MESH

Shell elements are recommended for modeling of plates in FEA of structural connection. 4-node quadrangle shell elements with nodes at its corners are applied. Six degrees of freedom are considered in each 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 the membrane and the flexural components.

The shell is divided into five integration layers through thickness of the plate at each integration point and plastic behaviour is analyzed in each point.

Only the maximum stresses and strains of all layers are shown.

All plates of a beam cross-section have a common division into elements. The size of generated finite elements is limited. The minimal element size is set to 10 mm and the maximal element size to 50 mm (can be set in Code setup). Meshes on flanges and webs are independent of each other. The default number of finite elements is set to 8 elements per cross-section height as shown in the following figure. The user can modify the default values in Code setup.

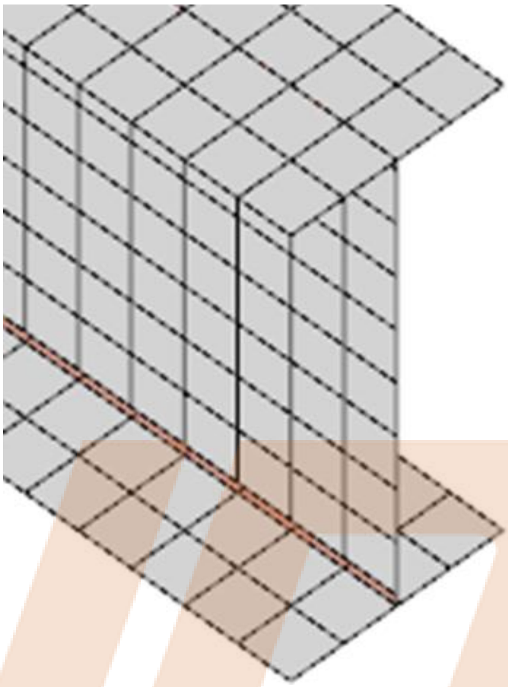

Figure 0.6. The mesh on a beam with constraints between the web and the flange plate.

CONTACTS

If penetration of a node into an opposite contact surface is detected, penalty stiffness is added between the node and the opposite plate. The contact between the plates has a major impact on the redistribution of forces in connection.

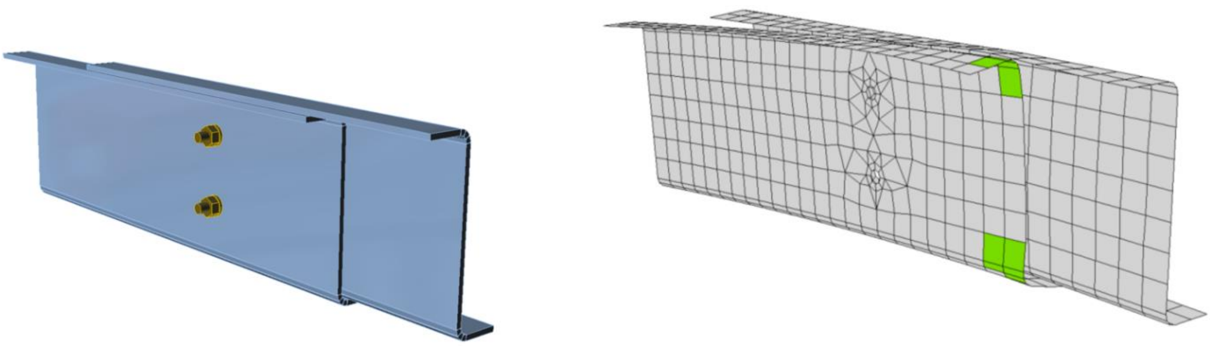

Figure 0.7. An example of separation of plates in contact between the web and flanges of two overlapped Z sections purlins.

It is possible to add contact between:

- two surfaces,
- two edges,
- edge and surface.

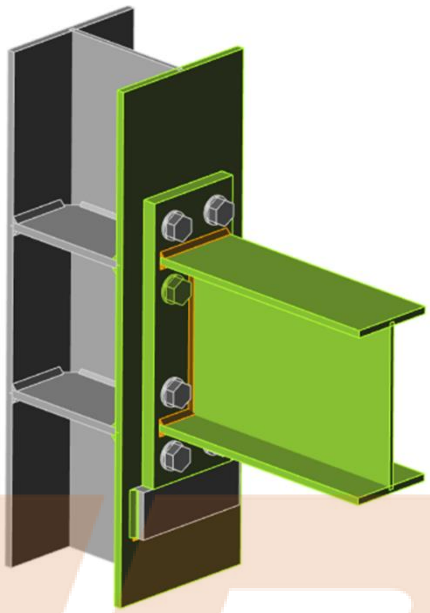

Figure 0.8. An example of edge to edge contact between the seat and the end plate.

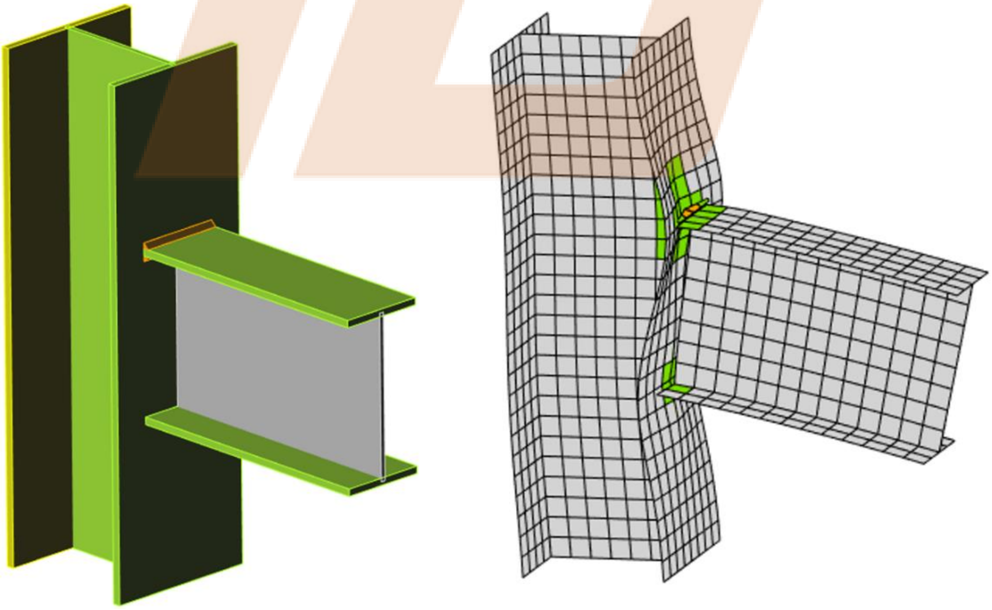

Figure 0.9. An example of edge to surface contact between the lower flange of the beam and the column flange.