SCIAENGINEER

# Steel connections

## Design and checks of steel connections

# Contacts

### *Belgium Headquarters*

SCIA nv
Industrieweg 1007
B-3540 Herk-de-Stad
Tel: +32 13 55 17 75
E-mail: info@scia.net

Support Phone
CAE (SCIA Engineer)
Tel: +32 13 55 09 90

CAD (Allplan)
Tel: +32 13 55 09 80

Support E-mail:
support@scia.net

### *France*

SCIA France sarl

Centre d'Affaires
29, Grand' Rue
FR-59100 Roubaix
Tel.: +33 3.28.33.28.67
Fax: +33 3.28.33.28.69
E-mail: france@scia.net

Agence commerciale
8, Place des vins de france
FR-75012 Paris
Tel.: +33 3.28.33.28.67
Fax: +33 3.28.33.28.69
E-mail: france@scia.net

### *Brazil*

SCIA do Brasil Software Ltda
Rua Dr. Luiz Migliano, 1986 - sala 702 , CEP
SP 05711-001 São Paulo
Tel.: +55 11 4314-5880
E-mail: brasil@scia.net

### *USA*

SCIA North America
7150 Riverwood Drive
21046 Columbia, MD
Tel.; +1 443-542-0638
Fax.:+1 410-290-8050
E-mail: usa@scia.net

### *Netherlands*

SCIA Nederland B.V.
Wassenaarweg 40
NL-6843 NW ARNHEM
Tel.:+31 26 320 12 30
Fax.: +31 26 320 12 39
E-mail: info@scia.net

### *Switzerland*

SCIA Swiss Office
Dürenbergstrasse 24
CH-3212 Gurmels
Tel.: +41 26 341 74 11
Fax: +41 26 341 74 13
E-mail: info@scia.ch

### *Czech Republic*

SCIA CZ s.r.o. Praha
Evropská 2591/33d
160 00 Praha 6
Tel.: +420 226 205 600
Fax: +420 226 201 673
E-mail: info.praha@scia.cz

SCIA CZ s.r.o. Brno
Slavickova 827/1a
638 00 Brno
Tel.: +420 530 501 570
Fax: +420 226 201 673
E-mail: info.brno@scia.cz

### *Slovakia*

SCIA SK s.r.o.
Murgašova 1298/16
SK-010 01 Žilina
Tel.: +421 415 003 070
Fax: +421 415 003 072
E-mail: info@scia.sk

| *Austria* | *Germany* |
|---|---|
| SCIA Datenservice Ges.m.b.H. | SCIA Software GmbH |
| Dresdnerstrasse 68/2/6/9 | Technologie Zentrum Dortmund, Emil-Figge-Strasse 76-80 |
| A-1200 WIEN | D-44227 Dortmund |
| Tel.: +43 1 7433232-11 | Tel.: +49 231/9742586 |
| Fax: +43 1 7433232-20 | Fax: +49 231/9742587 |
| E-mail: info@scia.at | E-mail: info@scia.de |
| Support | |
| Tel.: +43 1 7433232-12 | |
| E-mail: support@scia.net | |




Document created: 27 / 05 / 2016

SCIA Engineer 16.0

# Introduction to connections

Module Connections of SCIA Engineer has been made to design frame connections of steel structures.

While a connection is being entered in the graphical windows of SCIA Engineer, each element of the connection is drawn (connected members, endplates, stiffeners, bolts, etc.). In addition, the program checks detailing requirements laid in the code. What's more, the allowable forces on the connection are calculated and they must be compared with the actual forces acting in the connection. The program also lists the parts that determine the resistance of the connection, thus enabling the user to take appropriate actions.

After design and calculation, the program can generate overview drawings and detail drawings of the connection and the connection elements. A report about the calculation can be printed.

The program is based on EN 1993-1-8.

The algorithms and methods described in these references are used to calculate the limit states of the connection. The capacities of the underlying steel parts are calculated by the formulas given in the respective national codes (EC3, DIN18800 T1 or BS 5950-1:2000), depending on the national code setup. For other codes the design of steel connections is not available.

An overview of the used formulas for the specified steel parts is given 'Connect Frame Theoretical Background', Chapter 'Connection analysis according to DIN18800 and BS 5950-1'.

# Connection types

## Introduction to connection types

The program calculates pinned and rigid connections (connections which transfer bending moment) . Rigid connections are often not completely rigid, but allow a certain deformation. The stiffness of this connection is calculated by the program. When the stiffness is low, it has to be taken into account in the calculation model to determine the internal forces in the structure.

The description of connection types that can be analysed in the program is given in chapters:

- Analysis types
- Geometric types

## Analysis types

### Frame bolted and welded connections

For the calculation of (semi)-rigid connections, the following characteristics of connection are introduced in Eurocode 3:

- Moment Resistance
- Rotational Stiffness

This design method allows us to determine a "Moment-rotation characteristic", which in turn allows us to represent the real connection by a rotational spring connection defined in the centre lines of column and connected beam in the point of their intersection (approximation to the real behaviour of the connection).

By using this method, the design of non-stiffened connections can be considered, which results in a reduction of the total cost of structural steelwork.

The principles for the design of these semi-rigid connections are satisfied when the detailed application rules given in the revised Annex J of Eurocode 3, Ref. [1] are followed. For the design of column bases, the application rules given in Annex L of Eurocode 3, Ref. [5] are followed.

The following types of connections are supported :

- Beam-to-column connections : Bolted endplate + welded connections (knee, T, cross - with continuous beam or continuous column)
- Beam-to-beam connections : Endplate type beam splice (plate-to-plate connection).
- Column bases : Bolted base plate connection.

The types "beam-to-beam" and "column bases" are limited to symmetric and asymmetric I beams (including the elements with variable height) and RHS sections, both for major-axis bending configurations.

For the type " beam-to-column", the beam element is limited to symmetric and asymmetric I beams (including the elements with variable height) and RHS sections, both for major-axis bending configuration ; the column element is limited to symmetric I beams (including elements with variable height) in major-axis configuration, and to symmetric I beams in minor-axis bending configuration.

The following types of stiffeners are supported :

Beam-to-column connection:

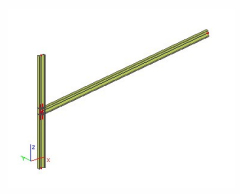
- Haunches : Welded from plate or made from a profile
- Web doublers : Supplementary web plates
- Backing plates : On the flanges
- Stiffeners : Triangular + rectangular, positioned in beam or column

Column base connection:

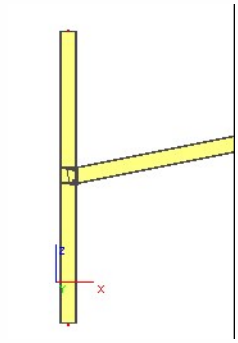
- Haunches : Welded from plate or made from a profile
- Stiffeners : Triangular + rectangular positioned in column at haunches
- Flange wideners
- Shear iron

Note: The frame connection can be defined between two beams (e.g. column and beam) that must be perpendicular to each other in the "direction of connection". This on-the-spot-invented phrase will be best explained on a simple example.

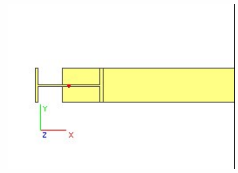
Let us assume a column and a beam made of vertically oriented I-section.



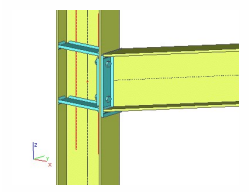
In the side view, the two members are not perpendicular.



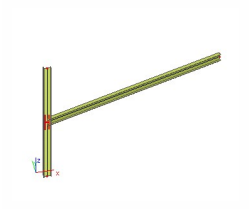
In the plan view, however, they ARE perpendicular to each other. And this is the "direction of connection".



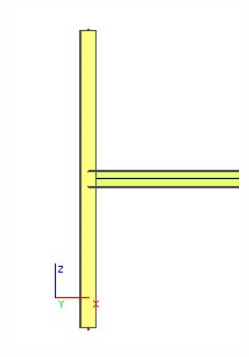
Therefore, the connection can be defined here.



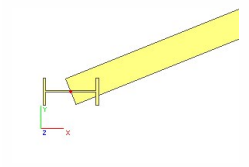
On the other hand, let us assume another column and a beam made of the same vertically oriented I-section. At first sight (in axonometric view) everything seems to be the same.



In the side view, the two members are perpendicular.

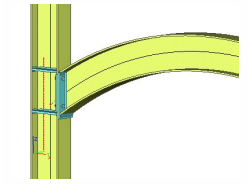


In the plan view, however, they are NOT perpendicular to each other. And this is the "direction of connection".



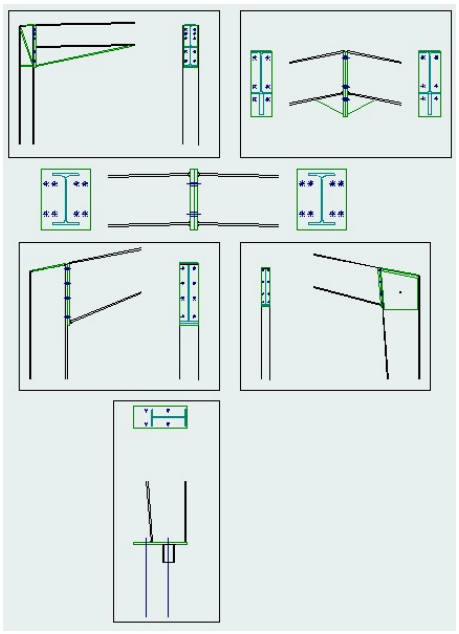
Therefore, the connection cannot be defined here.

The same can be said for curved beams. If the curvature of the beam does not break the condition of perpendicularity in the "direction of connection", the connection can be defined, see e.g. the picture below.

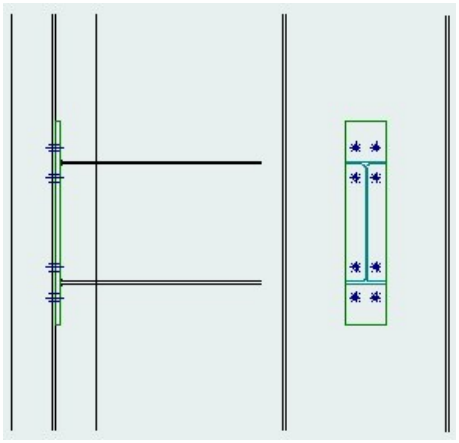


## Strong axis versus weak axis

### Strong axis connection



### Weak axis connection



## Frame pinned connections

Frame pinned connections are connections that do not transfer any moment. This is caused by the gap between the beam flange and the column flange.

The following type of connections are supported:

- Beam-to-column connections (knee, T, cross).

The beam element is limited to I-section and RHS section, both for major-axis bending configuration. The column element is limited to I-section in major-axis and minor-axis bending configuration.

The following types of connection elements are supported:

- plate welded to beam web and welded to column flange,
- plate bolted in beam web and welded to column flange,
- angle section bolted in beam web and bolted in column flange,
- short endplate welded to beam web and bolted in column flange.

## Grid Pinned Connections

Grid Pinned connections are beam-to-beam connections. Three main forms are considered in SCIA Engineer:

- Welded pinned plate
- Bolted pinned plate
- Cleat
- Short end plate

The grid pinned connections are checked for critical shear force and normal force. The following critical situations are considered:

(1) VRd : design shear resistance for the connection element

(2) VRd : design shear resistance of the beam

(3a) VRd : design block shear resistance for beam web

(3b) VRd : design block shear resistance for connection element (beam side)

(3c) VRd : design block shear resistance for connection element (column side)

(3d) VRd : design block shear resistance for endplate (beam side)

(4) VRd : design shear resistance due to the bolt distribution in the beam web

(5) VRd : design shear resistance due to the bolt distribution in the column

(6) VRd : design shear resistance at the notch

(7) NRd : design compression/tension resistance for the connection element

(8) NRd : design compression/tension resistance of the beam

(9) NRd : design tension resistance due to the bolt distribution in the column

[(10) NRd : design compression resistance for column web]

For more information see Theoretical Background Manual.

# Bolted diagonal connections

## Bolted diagonal connections

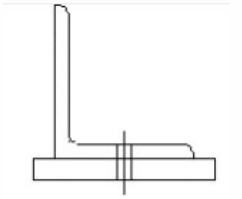
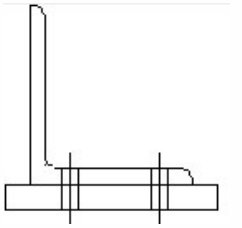
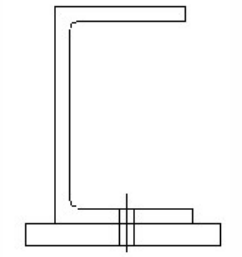
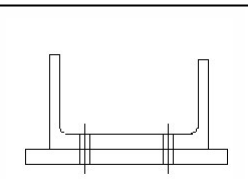
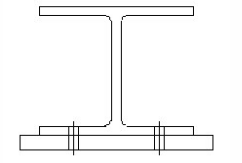
This chapter deals with the design and check of a bolted connection, where the member is subject to normal force. There are 2 ways of connection of the diagonal to another member (to a column):

- bolting the diagonal on a gusset plate,
- bolting the diagonal member directly to the column member.

## Diagonal element bolted on a gusset plate

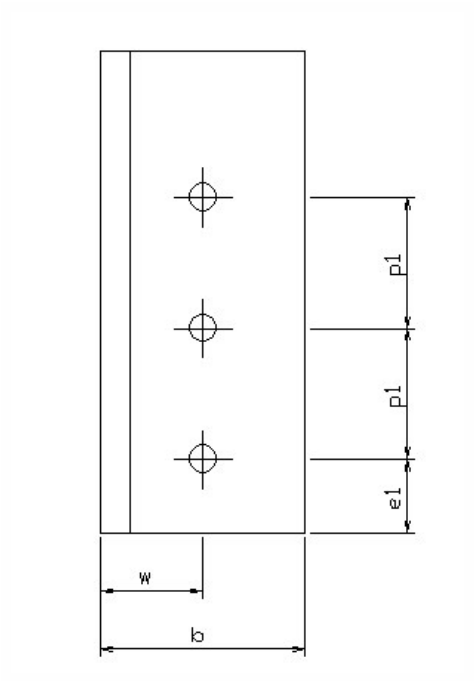
For both ends of the selected diagonal elements, gusset plate connection can be designed.

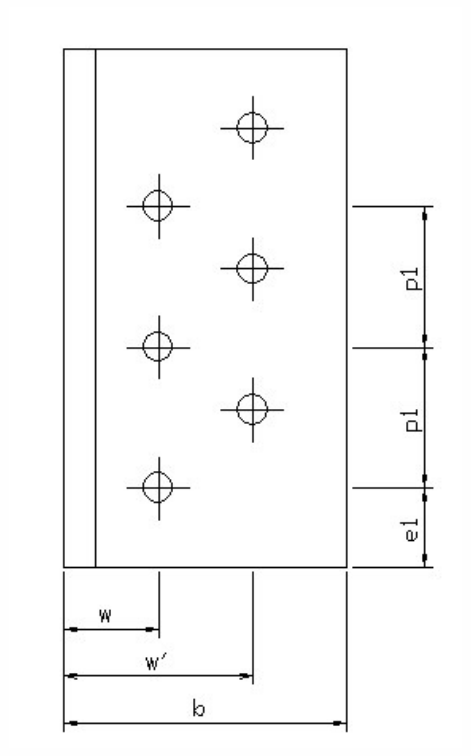
Possible configurations are:

| | |
|---|---|
| angle diagonal with single bolt row |  |
| angle diagonal with double staggered bolt row |  |
| channel diagonal bolted through flange with single bolt row |  |
| channel diagonal bolted through web with double (non) staggered bolt row |  |
| I section diagonal bolted through flange with double (non) staggered bolt row |  |

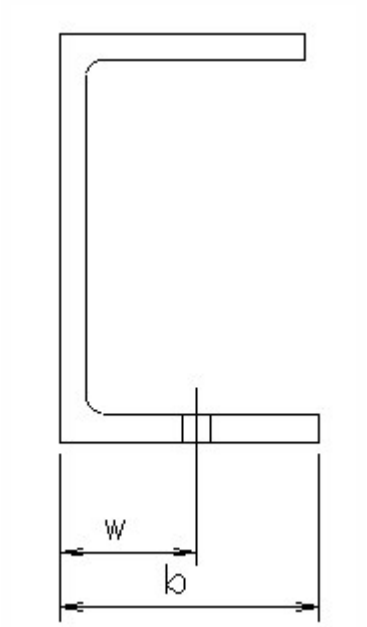
*Bolt configurations*

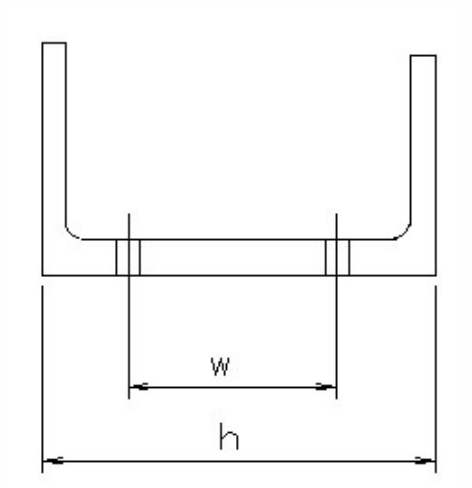
# Angle section



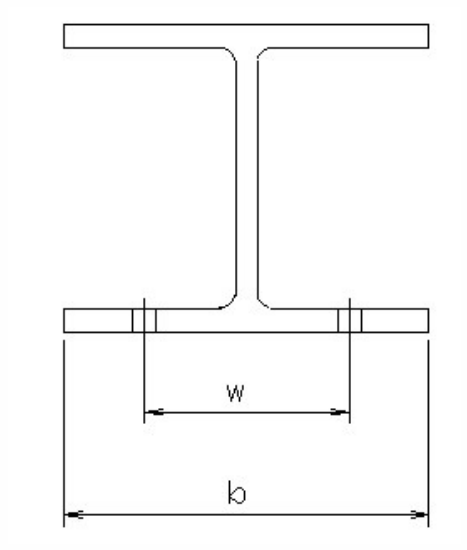




# Channel section

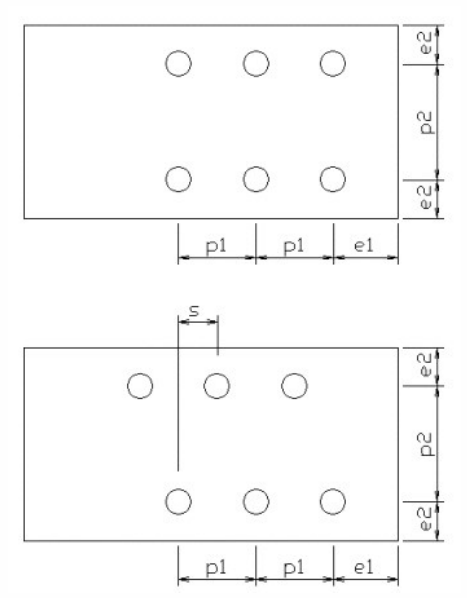


# I section

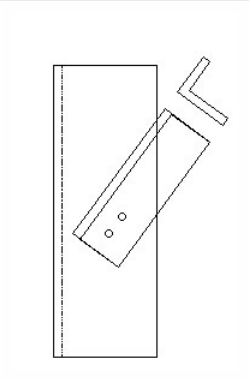
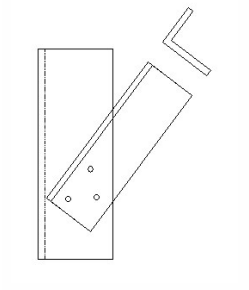
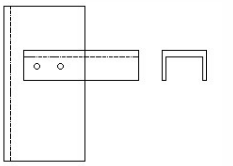
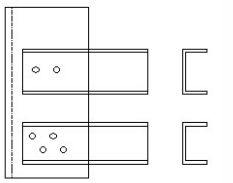



# Gusset plate




### Diagonal element bolted on a column

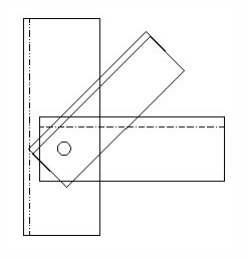
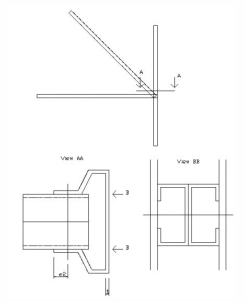
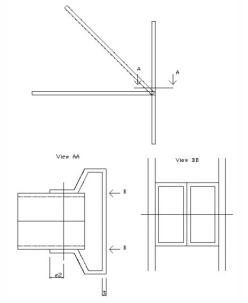
Both ends of the selected diagonal elements are connected to the column element. The column element is an angle section. The following configurations are possible:

| | |
|---|---|
| angle diagonal with single bolt row |  |
| angle diagonal with double staggered bolt row |  |
| channel diagonal bolted through flange with single bolt row |  |
| channel diagonal bolted through web with single<br>channel diagonal bolted through web with double (non) staggered bolt row |  |

Note: For bolt configurations see chapter Diagonal element bolted on a gusset plate.

## Multiple diagonal elements connected to one column

The selected diagonal elements (1 or 2 elements) are connected by means of 1 single bolt to the column element. The column element is an angle section or a cold-formed section. The following configurations are possible:

| | |
|---|---|
| angle diagonal(s) on angle column |  |
| cold formed diagonal on cold formed column |  |
| RHS diagonal on cold formed column |  |

## Properties of bolted diagonal connection

The connection properties consist of several input parts. Each part can be edited on a separate tab sheet.

| | |
|---|---|
| General | Defines the connection name and comments. |
| Connection | Sets the element and plate properties. |
| Bolts configuration | Defines the bolt configuration. |

## Gusset properties

### *2D Plate group*

| | |
|---|---|
| [Material] | Use this button to select the material property for the gusset plate. The default material is taken from the diagonal element. |
| Thickness | Defines the thickness of the gusset plate |
| Throat thickness | Defines the weld throat of the gusset plate. When the value of "0.0" is input, the weld size during the calculation is considered as a half of the gusset thickness.<br><br>The weld size is used to calculate the necessary weld length for the gusset plate. |

### *First diagonal element group*

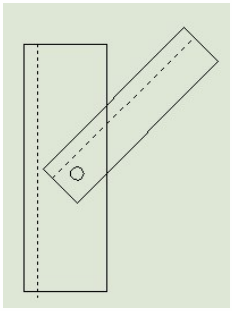
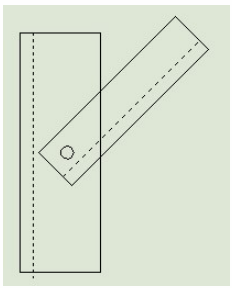
| | |
|---|---|
| N | The critical normal force is displayed. Depending on the settings, the normal forces is the critical traction force or the critical traction/compression force. |
| Short, Long<br>Web, Flange | Use this radio button to select the proper part of the section for bolting the gusset plate.<br>Angle section - Short : bolted in short leg<br>Angle section - Long : bolted in long leg<br>U section - Web : bolted through web<br>U section - Flange : bolted through flange |

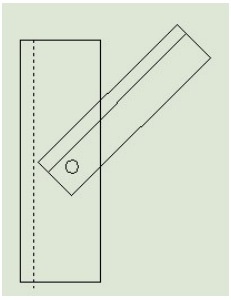
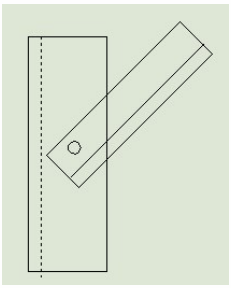
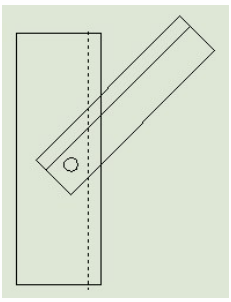
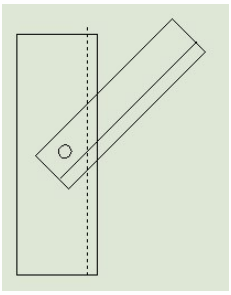
## Type Column I and Column II

For each angle diagonal, the position is defined.

### *Diagonal position group*

| | |
|---|---|
| Position 1 | Diagonal is inside the angle column. |
| Position 2 | Diagonal is outside of the angle column. |
| Position 3 | Diagonal is outside of the angle column (crossing the leg of the angle column) |
| Up | Leg of the angle diagonal is on the upper side. |
| Down | Leg of the angle diagonal is on the lower side. |

| | |
|---|---|
| Position 1<br>Flange Up |  |
| Position 1<br>Flange Down |  |

| | |
|---|---|
| Position 2<br>Flange Up |  |
| Position 2<br>Flange Down |  |
| Position 3<br>Flange Up |  |
| Position 3<br>Flange Down |  |

## Bolt configuration

These groups are valid for all types.

### *Bolts group*

| | |
|---|---|
| [Bolt] | Use this button to define the bolt characteristics. |
| Hole d | Display of the related bore hole for the bolt. The default value is taken from the bolt characteristics. |

### *Bolts position group*

| | |
|---|---|
| One row | Defines the number of rows. |

| | |
|---|---|
| Two rows | |
| No. in one row | Defines the number of bolts in one row. |
| [Optimisation] | Use this button to let the program search for the number of bolts, with respect to capacity of the connection. |
| p1 | Defines the spacing p1 between the bolts on a row.<br>The default value can be set in the basic settings for each bolt diameter. |
| p2, w | Defines the spacing p2 between the bolt rows. |
| staggered<br>non-staggered | Defines the bolt staggering. |
| s | For staggered bolt position, defines the staggered pitch, the spacing of the centres of two consecutive holes, measured parallel to the member axis. The default value is p2/2. |

### *First diagonal group*

| | |
|---|---|
| e1 | Defines the end distance in the first diagonal.<br>The default value can be set in the basic settings for each bolt diameter. |
| we2 | Defines the edge distance in the first diagonal. |

### *Second diagonal group*

This group is valid for the 'Column II' type.

| | |
|---|---|
| e1 | Defines the end distance in the second diagonal.<br>The default value can be set in the basic settings for each bolt diameter. |
| we2 | Defines the edge distance in the second diagonal. |

### *2D Plate group*

This group is valid for the 'Gusset' type.

| | |
|---|---|
| e1 | Defines the end distance in the plate.<br>The default value can be set in the basic settings for each bolt diameter. |
| e2 | Defines the edge distance in the plate.<br>The default value can be set in the basic settings for each bolt diameter. |

### *Column group*

This group is valid for the 'Column I' and 'Column II' type.

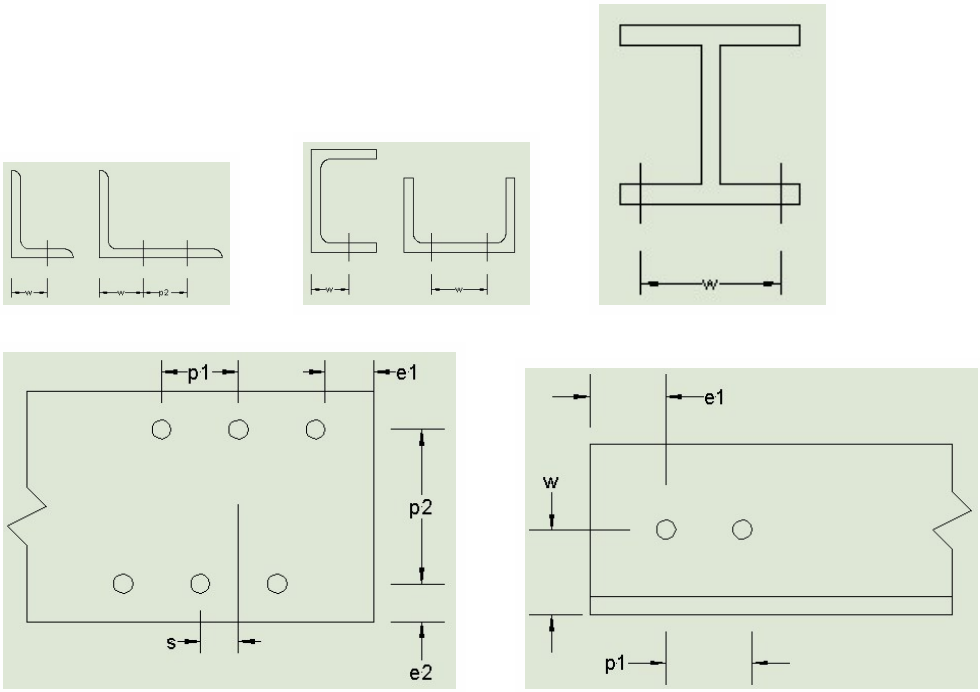
| | |
|---|---|
| we2 | Defines the edge distance in the column. |

### *View bolt limits*

Check this box to visualize the bolt limits in the diagonal and column elements.

## Geometrical configurations

### *Geometrical configurations for w, e1, e2, p1, p2, s*




## Default values

### *Default values for w, e2, p2, s*

The default values for standard sections are stored in the profile library: the properties 74, 75 and 76 describe the default bolt positions in the element.

| Property number | Description |
|---|---|
| 74 | w1 |
| 75 | w2 |
| 76 | w3 |

If the values w1, w2 and w3 are not present in the profile library, the following defaults are used :

### *Default values w and p2 for angle sections*

| | 1 bolt line<br><br>w | 2 staggered<br>bolt lines<br>w | 2 staggered<br>bolt lines<br>p2 |
|---|---|---|---|
| equal legs<br>w1<>0<br>w2=0 | w1 | b/3 | b/3 |
| equal legs<br>w1<>0 | b/2 | w1 | w2 |

| | | | |
|---|---|---|---|
| w2<>0 | | | |
| equal legs<br>w1=0<br>w2=0 | b/2 | b/3 | b/3 |
| unequal legs<br>long leg<br>w1<>0<br>w2=0 | w1 | b/3 | b/3 |
| unequal legs<br>long leg<br>w1<>0<br>w2<>0 | b/2 | w1 | w2 |
| unequal legs<br>short leg<br>w3<>0 | w3 | b/3 | b/3 |
| unequal legs<br>short leg<br>w3=0 | b/2 | b/3 | b/3 |

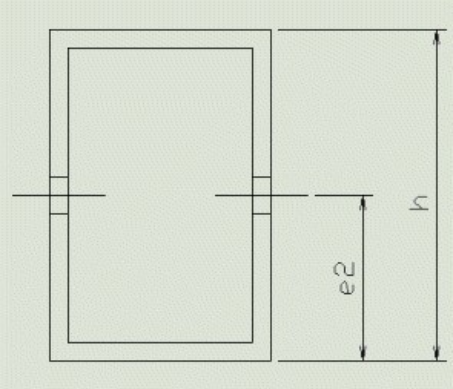
### *Default value w for U sections*

| | w |
|---|---|
| flange connection<br>w1<>0 | w1 |
| flange connection<br>w1=0 | b/2 |
| web connection | h/2 |

### *Default values w for I sections*

| | w |
|---|---|
| w1<>0 | w1 |
| w1=0 | b/2 |

### *Default values e2 for RHS sections*



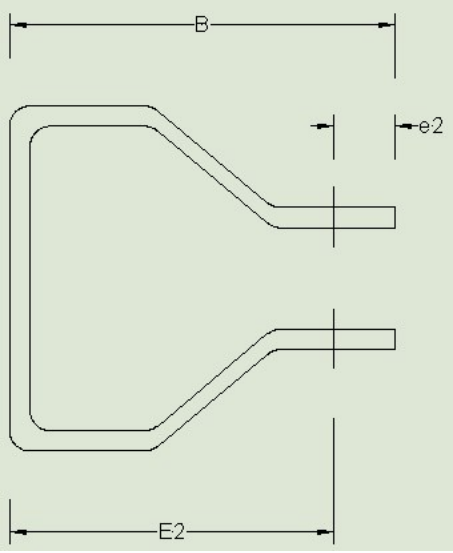

The default values for e2 is h/2.

### *Default values e2 for cold formed sections*

The default bolt configuration is stored in the profile library : the properties 67,48, 142,143 describe the element thickness and the default bolt positions in the element.

| Property number | description |
|---|---|
| 67 | s (thickness) |
| 48 | B (width) |
| 142 | sp (number of shear planes) |
| 143 | E2 |

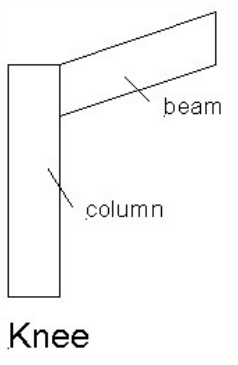
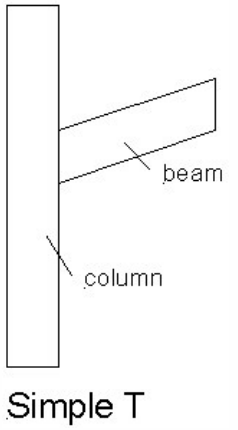
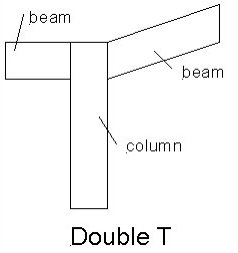
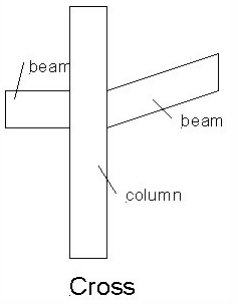
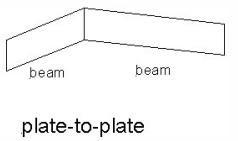
The default value e2=B-E2.

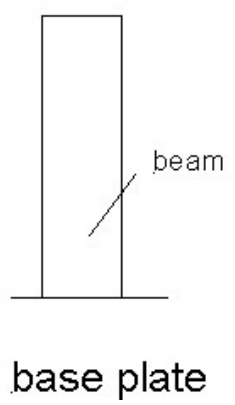



# Geometric types

## Introduction to geometric types

According to its geometry, a connection can be sorted into specific group. SCIA Engineer enables the user to deal with the following groups of connection:

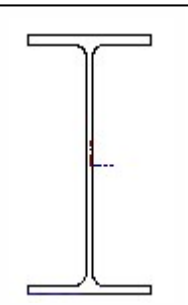
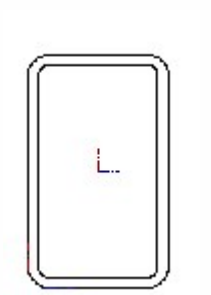
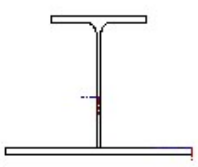
| | |
|---|---|
| knee |  |
| simple T |  |
| double T |  |
| cross |  |
| plate-to-plate |  |

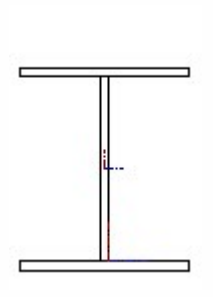
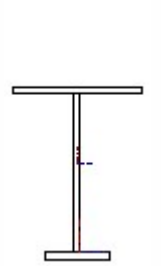
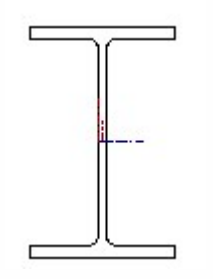
| | |
|---|---|
| base plate |  |

Note: In all types, the continuous element is called column, the element where the connection is situated, is called beam.

## Supported cross-section types

The SCIA Engineer Connections module supports a limited number of cross-section types that may appear on connected beams. SCIA Engineer can deal with the following cross-section types:

| | |
|---|---|
| rolled I beam |  |
| RHS –rolled hollow section |  |
| built up I section (made of a flat and T section) |  |

| | |
|---|---|
| symmetrical welded I section (made of three flats) |  |
| asymmetrical welded I section (made of three flats) |  |
| I section with a haunch<br>(elements with variable height) |  |

# Connection parts

## Connectors

### Introduction to connectors

Connectors, as the name suggests, connect two part together. In SCIA Engineer the user may come across two types of connectors: bolts and anchors.

Bolts are used for connection of two steel parts together. Anchors appear at base plate and connect the steel plate to a concrete base.

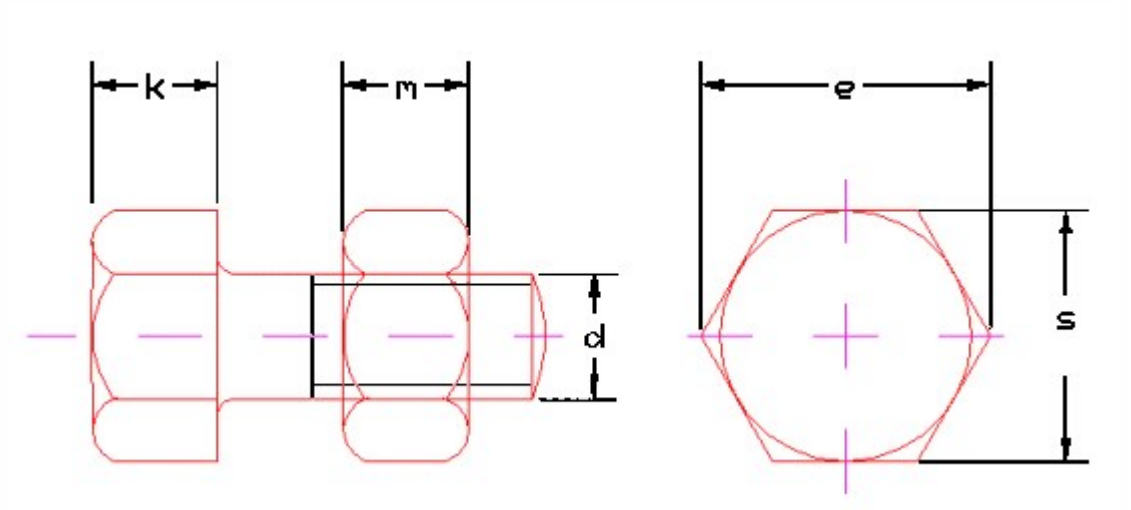
### Bolts

Bolts are used together with appropriate washers and nuts.

#### Bolt

Bolts are defined by the following parameters:

- bolt description
- bore hole
- construction diameter (= diameter required for bolt key)
- head diameter (see s in Fig. below)
- diagonal head diameter (see e in Fig. below)
- head height (see k in Fig. below)
- gross cross-section A of the bolt
- tensile stress area As of the bolt



### Nut

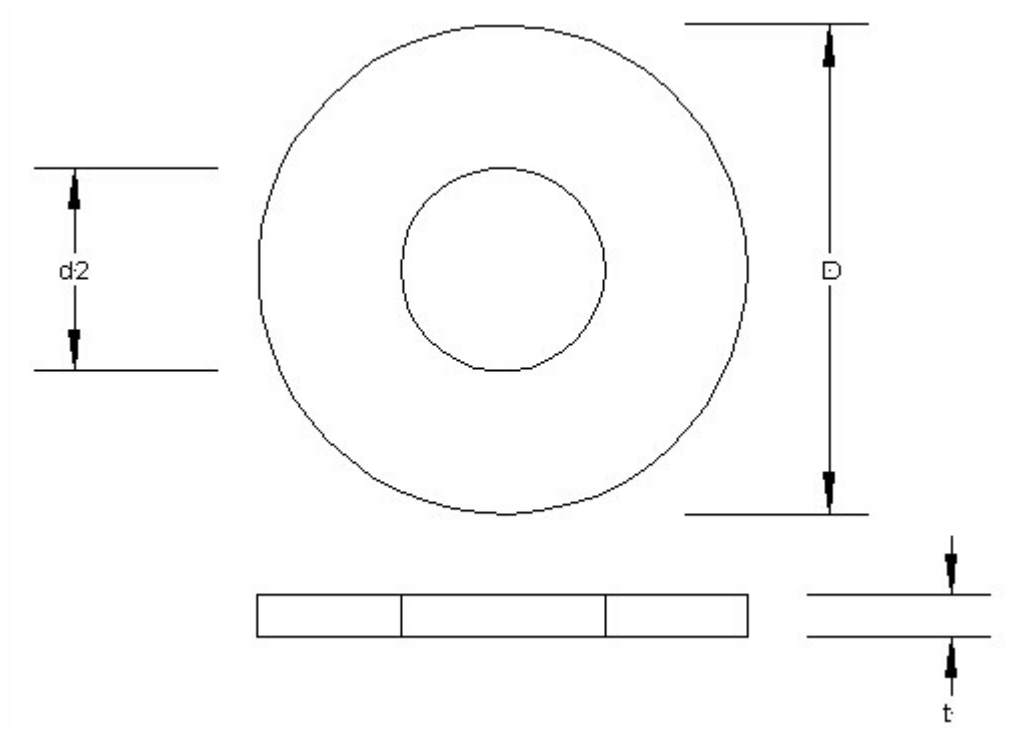
Nuts are defined by the following parameters:

- nut description
- nut diameter (see s in Fig. above)
- nut diagonal diameter (see e in Fig. above)
- nut height (see m in Fig. above)

### Washer

The following properties define a washer (see Fig. below):

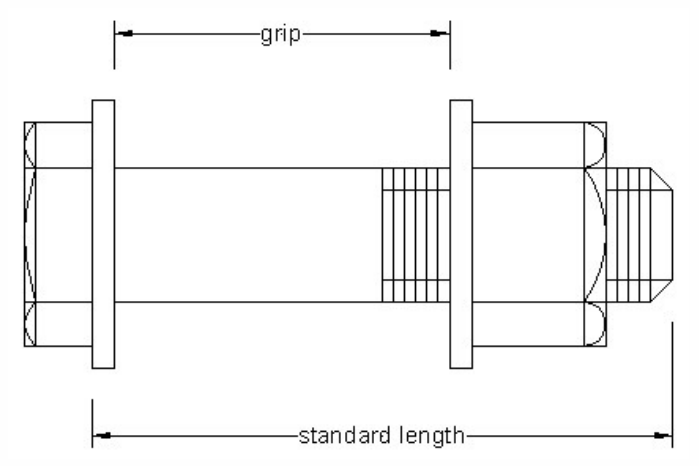
- washer description
- washer dimension D
- washer dimension t
- washer dimension d2
- washer material



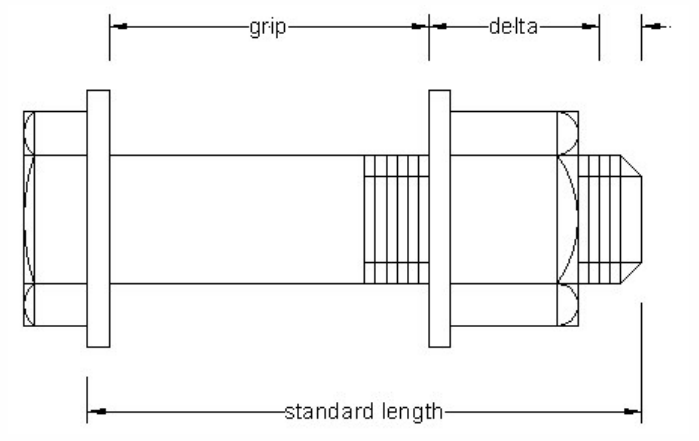

## Bolt length

A standard bolt length can be defined in 2 ways:

- by the relation standard length - grip length,

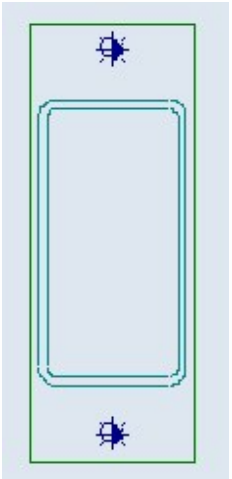


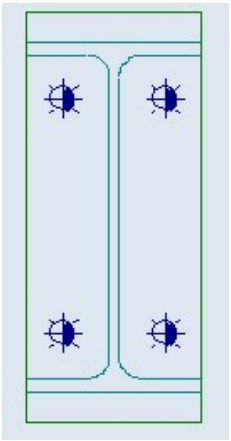
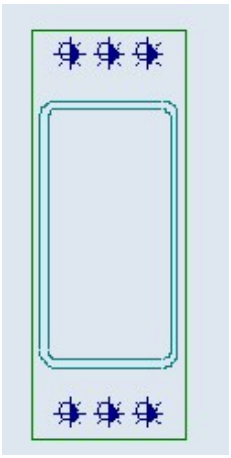
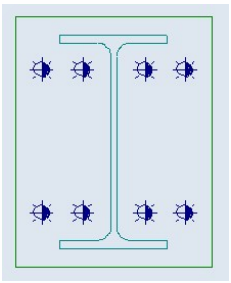

- by adding a specific value "delta" to the grip length : the standard length is given by (grip+delta+nut height+number of washers x washer thickness), rounded to following standard length,




## Bolt pattern

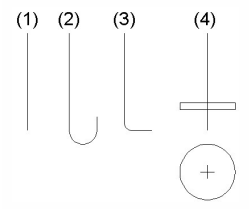
Depending on geometrical conditions and on load the connection is subject to, there can be used various bolt patterns.

| | |
|---|---|
| one bolt per row |  |

| | |
|---|---|
| two bolts per row |  |
| three bolts per row |  |
| four bolts per row |  |

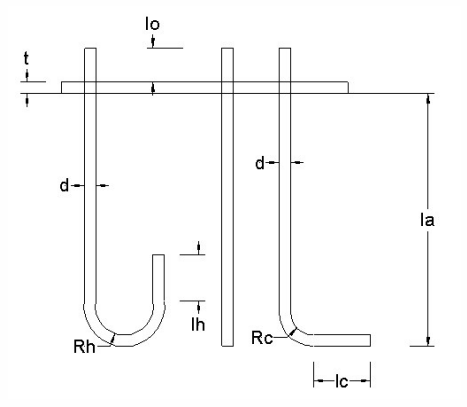
## Anchors

SCIA Engineer supports several anchor types:



1. straight anchor
2. hooked anchor
3. curved (bended) anchor
4. anchor with circular plate (washer plate)

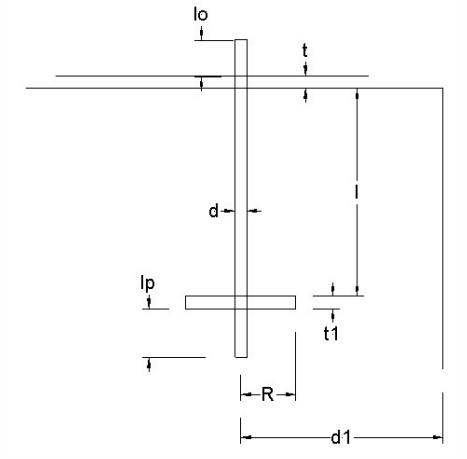
## Straight, hooked and curved anchor




| | |
|---|---|
| la | calculated anchor length |
| t | base plate thickness |
| d | anchor diameter |
| lo | overlength = f1 x d, default value f1=2 |
| lh | retourlength hooked anchor = f2 x d, default value f2=5 |
| lc | retourlength curved anchor = f3 x d, default value f3=5 |
| Rh | inner radius hooked anchor = f4 x d, default value f4=1.5 |
| Rc | inner radius curved anchor = f5 x d, default value f4=3 |

Note: Values f1, f2, f3, f4, f5 are input values and depend on anchor type.

## Anchor with a circular plate



| | |
|---|---|
| l | anchor length |
| t | base plate thickness |
| d | anchor diameter |
| lo | overlength = f1 x d, default value f1=2 |
| lp | overlength at circular plate = f6 x d, default value f6=3 |
| t1 | thickness circular plate |
| R | radius circular plate |
| d1 | distance from anchor to side of concrete block |

Note: Values f1, f6, l, d1 are input values. Values R, t1 are calculated by program.

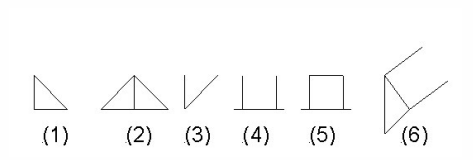
# Welds

## Introduction to welds

Weld is defined by:

- weld dimension
- weld length
- weld position
- weld type

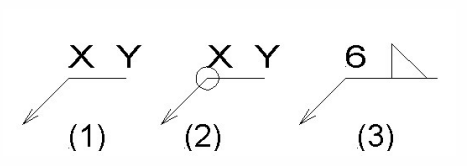
### Weld types




| | |
|---|---|
| 1 | fillet weld |
| 2 | double fillet weld |
| 3 | bevel (HV) weld |
| 4 | square weld |
| 5 | plug weld |
| 6 | weld length at haunch |

Note: Wed symbol (6) is not defined in codes. This symbol is used to represent the weld length which is calculated for haunch analysis. In the graphical representation, symbol (6) or symbol (3) can be used for depiction of weld size at haunches.

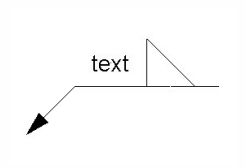
### Weld graphical symbols




Graphical representation of weld is made by the above shown welding symbol. The letter X stands for the weld size, and Y for the weld symbol. The circle symbol in (2) is the weld-all-around symbol.

The example on the right in (3) means : fillet weld of 6 mm weld size.

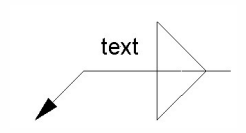
## Fillet weld




Possible applications of fillet weld are:

- short endplate / beam flange,
- closing plate,
- small web doubler,
- welded pinned plate.

## Double fillet weld




Double fillet weld can be used in numerous applications:

### Bolted connection

- endplate / beam flange,
- endplate / beam web,
- haunch web / beam flange,
- haunch web / end plate,
- stiffeners.

### Welded connection

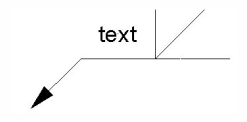
- column flange / beam flange,
- haunch web / column flange,

- stiffeners.

### Pinned connection

- endplate / column flange.

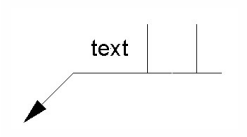
## Bevel weld




Possible applications of bevel weld are:

- haunch flange / endplate,
- haunch flange / beam flange,
- haunch flange / column flange.

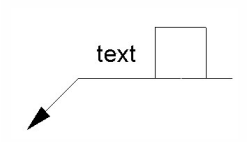
## Square weld




Possible application of square weld is:

- large web doubler.

## Plug weld




Possible application of plug weld is:

- weld size web doubler.

# Plates

## Introduction to plates

Plates in connections may have various roles:

- endplate
- stiffener

- backing plate
- web doubler
- flange wideners
- pinned plate
- short endplate

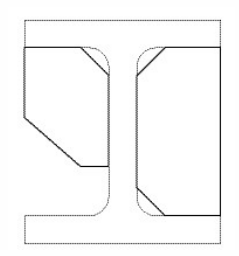
## End plate

An end plate is a rectangular plate attached to the end of a beam. One of bolt patters can be used to connect the plate with column.

## Stiffener

The geometry of stiffener is defined by the position and the shape. There are two option:

- rectangular stiffener (the left hand side of the picture below),
- triangular stiffener (the right hand side of the picture below).



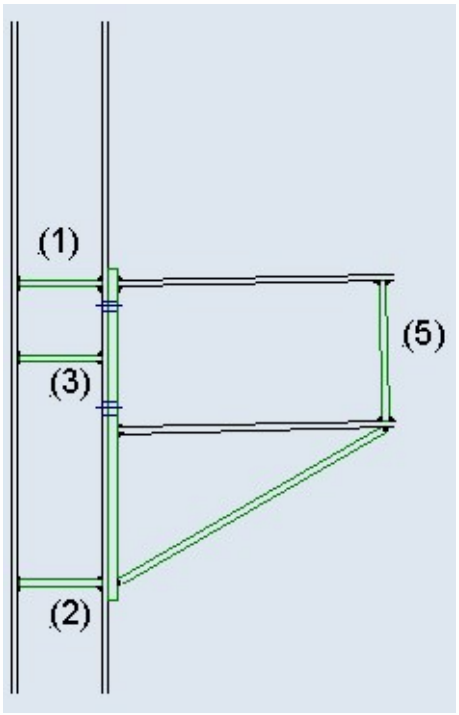
Possible positions of stiffeners are summarised in the following table and shown in the accompanying pictures.

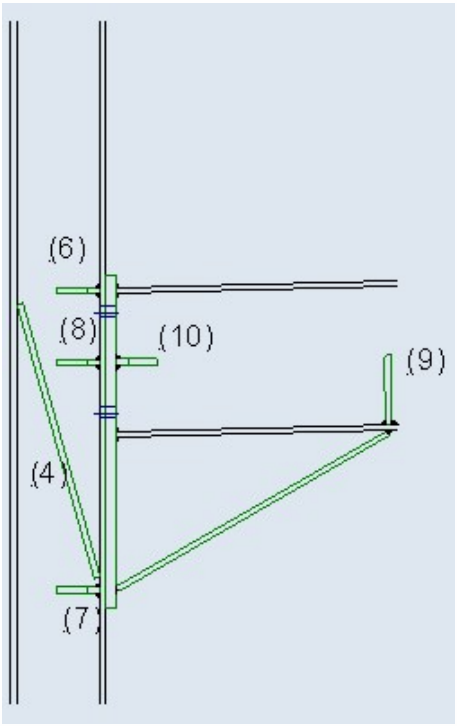
### Rectangular stiffener

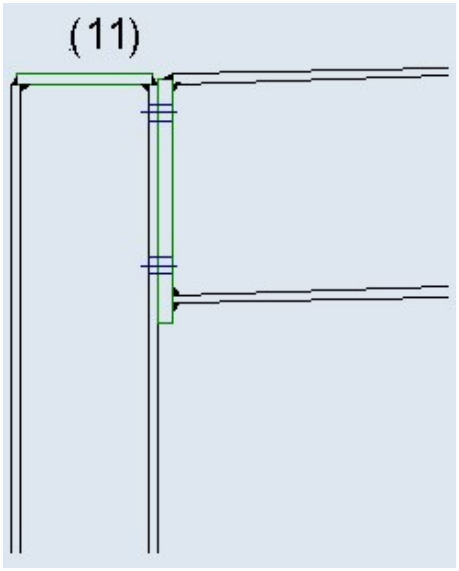
| | |
|---|---|
| 1 | column web, at top side |
| 11 | closing plate at top side |
| 2 | column web, at bottom side |
| 3 | column web, between bolts |
| 4 | column web, diagonal |
| 5 | beam web, at haunch end |

### Triangular stiffener

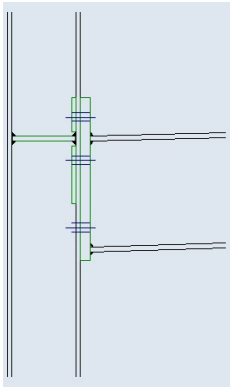
| | |
|---|---|
| 6 | column web, at top side |
| 7 | column web, at bottom side |
| 8 | column web, between bolts |
| 9 | beam web, at haunch end |


(1)
(5)
(3)
(2)


(6)
(8)
(10)
(9)
(4)
(7)


(11)

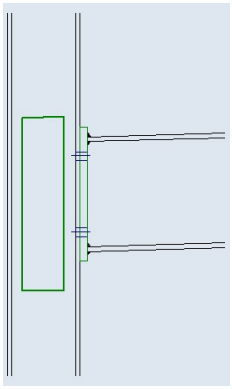
## Backing plate



The bolt pattern used for the backing plate depends on the pattern of associated end plate.

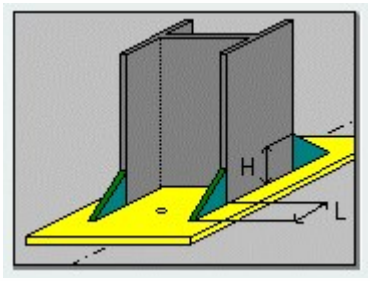
## Web doubler

A web doubler can be used to increase the strength of the column web.



## Flange widener

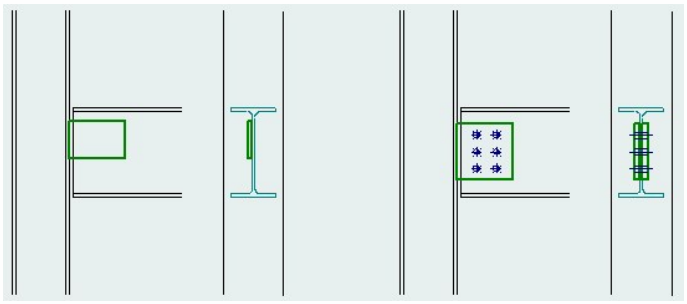
Flange widener is used to increase the width of a flange.



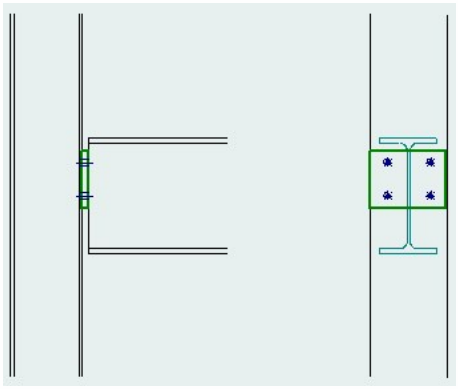

## Pinned plate

The pinned plate can be:

- bolted and/or welded,
- one sided or two sided.



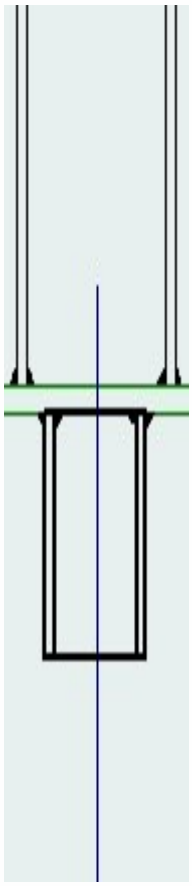
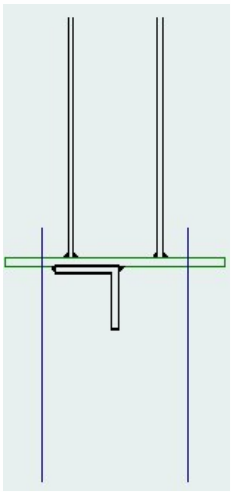
### Short endplate



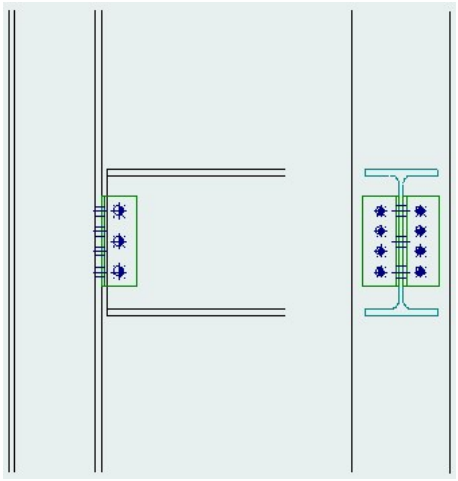
# Sections

### Shear iron

The shear iron has following section forms:

| | |
|---|---|
| rolled I |  |
| angle section |  |

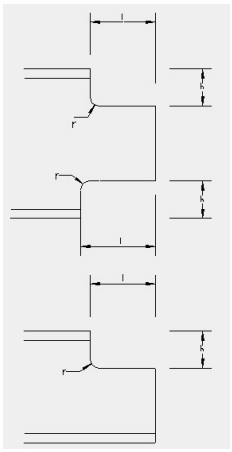
### Cleat



# Notches

### Notch

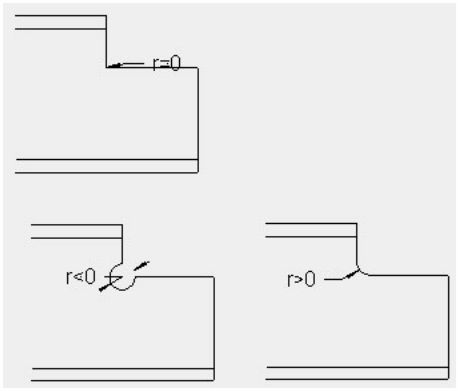
The notch is defined by depth h, length l and radius r. The notch can be situated at the top, and/or at the bottom flange.



### Rounding of a notch

The rounding r of the notch, can be

| | |
|---|---|
| 0 | no rounding |
| > 0 | r = diameter of rounding |
| < 0 | \|r\| = diameter of bore hole |




# Haunches

## Haunch

The connection may be "stiffened" by a haunch. Such a haunch is used for the purpose of the design and check of the connection (read carefully the note below).

| | |
|---|---|
| Type | Member select<br>The haunch is made from a selected type of cross-section. The list is limited to cross-sections used in the model of the structure.<br>Plate<br>The haunch is made of one flat.<br>With flange<br>The haunch is made of two flats – web and flange. |
| Input type | Height x angle<br>Height x length |
| Cross- section | If Type is set to Member select, the cross-section can be selected here. |
| Dimensions | The required dimensions of the haunch can be input here.<br>The list of dimensions differs according to the selected Type. |

Note: Adds a haunch to the connected beam. This haunch (i) affects the check of the connection, (ii) is taken into account when calculating the "substitute" stiffness of he connection (if option Update stiffness is ON), BUT (iii) is ignored in the calculation model itself (unlike the haunch defined by means of function Structure > Haunch.

# Designing a connection

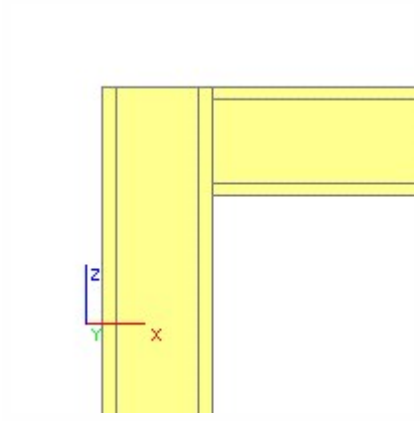
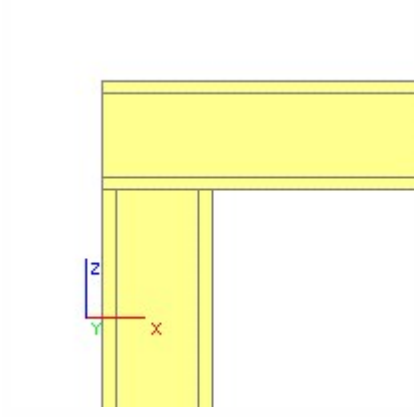
## Defining a new connection

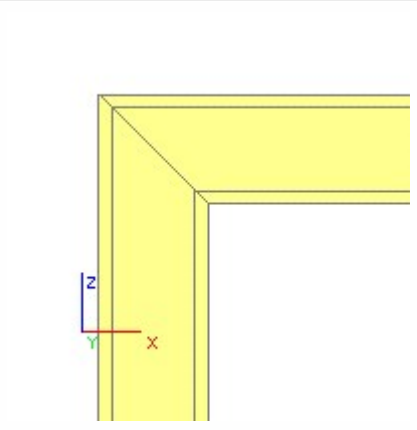
### Significance of defined beam types

When the user defines a new beam in his/her model, he/she may specify the type of the beam. The type is of no importance as far as determination of internal forces and stresses is in question. However, once the user starts with detailing and wants to design a connection of two or more beams, the parameter Type comes into effect. Each type is associated with a specific priority. The priority controls the way two beams are connected to each other.

It can be simply said that the type specifies priorities of individual beams, in other words which beam is the "master" and which one only the "slave", which determines the detailed geometry of the connection.

Let's demonstrate the said on a simple example. Let's assume a connection of two beams: vertical one and horizontal one. There are three possible configurations of such a detail.

| vertical beam | horizontal beam | connection |
|---|---|---|
| type = column<br>priority = 100 | type = beam<br>priority = 80 |  <br>knee connection |
| type = beam<br>priority = 80 | type = column<br>priority = 100 |  <br>knee connection |

| | | |
|---|---|---|
| type = beam<br>priority = 80 | type = beam<br>priority = 80 |  <br><br>plate to plate connection |

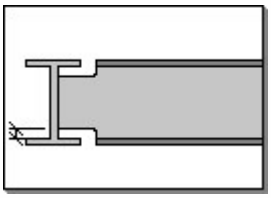
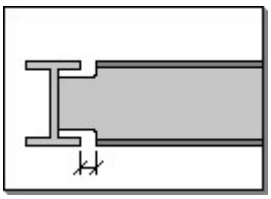
## Adjusting the default parameters

Before the design itself, the user may specify values of various connection-design-related parameters. These values will be then used as default values for individual particular connections.

### General data

This group of parameters covers the basic data such as minimal weld size, slip factor for preload bolts, minimum distances between bolts, etc.

| | |
|---|---|
| Minimum weld size | Minimum throat thickness of the welds. |
| Slip factor (preloaded bolts) | This is the slip factor m for preloaded high-strength bolts (see EC3, 6.5.8.3). |
| Factor for moment of prel. bolts | This factor k is used to determine the needed torque Mv.<br>Mv = k d Fv<br>where<br>Mv = torque<br>d = diameter of the prestressed bolt<br>Fv = design preloading force |
| Minimum distance bolt - edge plate (d) | Minimum edge distance. |
| Minimum distance bolt - top plate (d) | Minimum end distance. |
| Maximum distance bolt - edge plate (d) | Maximum edge distance. |
| Maximum distance bolt - top plate (d) | Maximum end distance. |
| Minimum distance between bolt rows (d) | Minimum spacing. |
| Maximum distance between bolt rows (d) | Maximum spacing. |
| Minimum distance between bolt on rows (d) | Minimum spacing. |
| Distance bolt – top/edge plate (pinned connection) | Default bolt position in bolted frame pinned connectors. |
| Adapt analysis stiffness | Default setting for adapting the stiffness during analysis. |

| | |
|---|---|
| overdepth for notch |  |
| overlength for notch |  |

## Frame bolted / welded

This group determines the way in which a frame connection is designed and calculated.

| | |
|---|---|
| Transformation of efforts | Specifies which internal forces are used as solicitation forces : In axis : internal forces in the node (intersection of beam and column axis). In edge : internal forces at the connection |
| Weldsize haunch | The presentation of weldsize at haunches can be either a fillet V weld, or the length as given in chapter Weld sizes for haunches in Frame Connect : Theoretical Background. |
| Allowable relative error for limit moments (%) | This is the permissible exceeding percentage of the Moment design resistance Mj,Rd, for a connection to still be classified as OK. |
| Omitting weld in beff (FcRd) | This is related to formula J.19 & J.20 of Ref. [1]. It is possible to omit the portion of the welds in the determination of the effective width beff, which is used to determine the design resistance of column web to transverse compression (Fc,wc,Rd). |
| Include stress in column flange | This is related to the use of the reduction factor kfc in the determination of the design resistance of column flange in bending. It is possible to omit kfc, so that no reduction for the longitudinal compressive stress scom,Ed in the column flange is made. See Ref. [1] J.3.5.5.2 (4) |
| Capacity beam flange | This is related to the determination of Fc,fb,Rd ,the determination of the design compression resistance of the beam flange in compression, for beams stiffened with haunches : Prof : Mc,Rd will be calculated for the beam cross- section only. Prof + haunch : Mc,Rd will be calculated for the cross-section consisting of the beam + the haunch(es).<br><br>Compr. haunch cfr. SPRINT : For the compressed haunch flange, the Fc,fb,Rd is calculated according to SPRINT regulations. See also chapter Compression resistance for haunches in Connect Frame : Theoretical Background. |
| Use alternative method for Ft.Rd.1 | This feature allows the user to use an alternative method, given in Ref. [1] J.3.2.4, for the determination of Ft,Rd for mode 1: Complete yielding of the flange.<br><br>This method leads to a higher value of the design resistance for mode 1. |
| Always adapt stiffeners | When this option is selected, the thickness of stiffeners will be always changed during the calculation of the node, depending on actual internal forces and critical limit forces. See Weld size calculation and Stiffener dimensions in Connect Frame : Theoretical Background.<br><br>When this option is not selected, the inputted thickness of stiffeners will be changed only if the inputted values are smaller than the calculated minimum thickness. |
| Always adapt | When this option is selected, the weld sizes are calculated during the calculation of the node, depend- |

| | |
|---|---|
| weldsizes | ing on the actual internal forces and the critical limit forces.<br>When this option is not selected, the default values are taken.<br>See Weld size calculation and Stiffener dimensions in Connect Frame : Theoretical Background. |
| Apply stiffness classification check | When this option is selected, the stiffness classification and the check of the required stiffness are applied. |
| Use internal forces for weld size calculation | Check this box for using the internal forces when dimensioning the weld sizes and the stiffeners. If not, the limit capacities are used. |
| Use stiffeners in column web panel resistance | Check this box if you want to use the rectangular stiffeners (in tension and compression zone) in the capacity of the column web panel in shear. |
| Use last bolt only for shear capacity | Check this box to have at least 1 bolt row which will not be considered in tension. This bolt row is used for full shear capacity. The bolt is situated the closest to the compression point. |

## Base plate

This group determines the way in which a base plate is designed and calculated.

| | |
|---|---|
| Apply stiffness classification check | When this check box is active, the stiffness classification and the check of the required stiffness are applied. |
| Concentration factor kj | Conservatively kj can be taken as 1.0. For the correct value, consult Ref. [5]. |
| Joint coefficient betaj | bj may be taken as 2/3 provided that the characteristic strength of the grout is not less than 0.2 times the characteristic strength of the concrete foundation and the thickness of the grout is not greater than 0.2 times the smallest width of the steel base plate. |
| Friction coefficient | Fill out the appropriate friction coefficient between mortar and steel. |
| Fck of concrete block | Characteristic compressive cylinder strength of the concrete at 28 days. |
| Good bond condition | Check this box for good bond condition. See EC2 Ref.[6], clause 5.2.2 for more information. |
| Bar type | High-bond bars: bars with a ribbed surface<br>Plain bars: bars with a plain surface |
| Friction included | This option allows you to take account for the friction resistance when determining the shear resistance VRd of the joint. Check this box to take account for the friction resistance. |
| Anchorage type | The Anchorage type describes the way the holding down bolts are anchored into the foundation :<br>straight: fixing is only based on bond<br>hooked : The anchors are provided with a hook<br>curved: see Note 1 below the table<br>circular plate: A washer plate is provided as the load distributing member |
| Overlength (d) | See definition of anchors in Connection Parts |

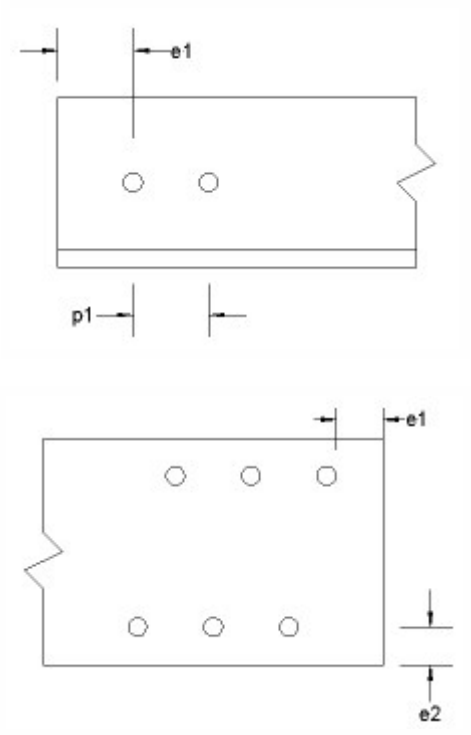
| | |
|---|---|
| Retourlength hooked anchor (d) | See definition of anchors in Connection Parts |
| Retourlength curved anchor (d) | See definition of anchors in Connection Parts |
| Inner radius hooked anchor (d) | See definition of anchors in Connection Parts |
| Inner radius curved anchor (d) | See definition of anchors in Connection Parts |
| Overlength at circular plate (d) | See definition of anchors in Connection Parts |
| Thickness circular plate (d) | See definition of anchors in Connection Parts |
| Radius circular plate (d) | See definition of anchors in Connection Parts |
| Distance from anchor to side of concrete block | See definition of anchors in Connection Parts |
| Anchor length | See definition of anchors in Connection Parts |
| Use support reactions | When this option is selected, the support reactions are used to design the base plate connection.<br>If not, the member forces of the column are used to check the connection. |
| Use internal forces for anchor lengths | Check this box in order to use internal forces when dimensioning the anchor length. Otherwise, the limit capacities are used. |

Note 1: Hooked and curved are basically the same types, they only differ in geometrical properties so they will result in the same anchorage length (curved is provided for future purposes)

Note 2: Hooked and curved bolt types should not be used for bolts with a specified yield strength higher than 300 N/mm$^2$ (According to Ref. [1]).

## Bolted diagonal

For common bolt diameters (M12 to M36) , the default values for e1 and p1 ( in the diagonal element) and the default values e1 and e2 (in the plate element) are set for staggered and non-staggered bolt positions.




Normal force type

The critical normal force for the connection design is searched using the selected criterion:

| | |
|---|---|
| Tension only | only tensile forces are considered |
| Tension and compression | both compression and tensile forces are considered |

## Partial safety factors

Here, partial safety factors can be specified.

| | Symbol | Remarks | Default value |
|---|---|---|---|
| cross- section and plates | $\gamma_{M0}$ | Resistance of Class 1,2 or 3 sections | 1.1 |
| members to buck-ling | $\gamma_{M1}$ | Resistance of members to buckling + Class 4 sections | 1.1 |
| bolted con-nection | $\gamma_{Mb}$ | | 1.25 |
| welded con-nection | $\gamma_{Mw}$ | | 1.25 |
| prestressed bolts | $\gamma_{Ms,ult}$ | Slip resistance for the ultimate limit state | 1.25 |
| concrete | $\gamma_{c}$ | Only used for a base plate connection | 1.5 |
| friction plate/-concrete | $\gamma$ | Safety factor on friction between steel and concrete. Only used for a base plate connection. | 2 |

| | | | |
|---|---|---|---|
| Partial Safety fact. Truss | $\gamma_{M1}$ | Safety factor on design compression/tension resistance. | 1.1 |

## Expert System

The following Relative errors can be set:

| | |
|---|---|
| Section | Sets the admissible relative dimension error for retrieving the connection beam and connection column. This is only valid when the check on dimensions option is set, or when a VARH section is used, or when search by name was without success. |
| Material | Sets the admissible relative material error for retrieving the connection materials (i.e. the material properties for beam, column, endplate, bolt, etc.) |
| Configuration | The data in the expert database are stored for a certain geometrical configuration. This is measured by the angle between the beam and the column or the base plate or the endplate. The tolerance for this angle is defined in the edit box. |
| Position | This relative error takes into account the difference between the design moment resistance at the top side and the bottom side. If the relative error between the design moment resistance at the top side and the bottom side is larger than the value which is set here, an inverted position of the connection will be considered. This value has no sense for pinned connections. |

## Order option group

This is the default order option for sorting the relevant connections.

- Unity check: sorting by unity check
- Priority sorting by priority number

## Filter options group

The number of entries in the retrieved connection list, can be customized and filtered by means of following options:

| | |
|---|---|
| uc min<br>uc max | Sets limits for the unity check. Only the unity checks, which are inside the boundaries, will be present in the connection list.<br>TIP: a good maximum value is 0.90. |
| priority min<br>priority max | Sets limits for the priority number. |
| capacity filter | The unity check can be based:<br>on the capacity table values,<br>on the calculated capacity values or<br>on a combination of both (minimum or maximum). |
| bolt exclude list | For a bolted connection, you can eliminate bolt grades. The connection list will not contain connections using the selected bolt grades. |

| | |
|---|---|
| source filter | Selects the desired sources for retrieving the connections. |

### Geometry checks group

This group specifies the criterion for selecting the connection beam and connection column. This can be performed using the section name (check on names), or by means of section dimensions (check on dimensions). The check on names option is the faster one. The check on dimensions option is always used for VARH sections or when the retrieving by names is without success.

The check on column properties check box should be activated if the user wants to include the column properties for finding the matching connections. If this is not activated, only the beam properties are consulted. It is obvious that with this last option, the retrieved unity check can differ from the calculated unity check. The number of retrieved connections will be larger.

***The procedure for adjustment of default parameters***

1. Open service Steel:
   1. using menu function Tree > Steel,
   2. or using tree menu function Steel ( ).
2. Open function Connections > Setup ( ).
3. Adjust the individual parameters in the table as required.
4. Confirm with [OK].

## Making a new connection

The procedure for the definition of a new connection is similar for all analysis types. This chapter presents the general procedure. The following chapters then deal with some specifics of individual connection types.

> Note: The connection may also be inserted into an intersection of two or more beams where linked node has been generated.

***The general procedure for the definition of a new connection***

1. In the graphical window of SCIA Engineer display the part of the structure where the new connection is supposed to be located. Make especially sure that the appropriate node and beams forming the connection can be clearly seen and easily selected.
2. Start appropriate function for the required type of connection either using menu function Tree > Steel > Connections > xxx or using tree menu function Steel > Connections > xxx, where xxx can be one of the following:
   1. Frame bolted / welded - strong axis,
   2. Frame bolted / welded - weak axis,
   3. Grid pinned,
   4. Bolted diagonal.
3. Select the node where the connection is supposed to be designed.
4. The program automatically selects all the beams that come to or pass through the selected node. If required, deselect any of the beams.
5. Press [Esc] to close the function.

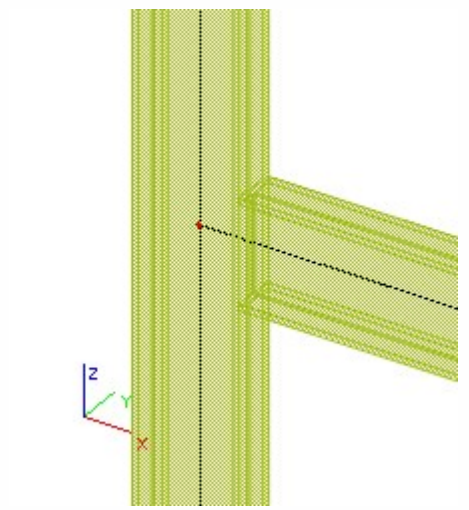
6. The program automatically creates the connection in the selected node. The type of the connection depends on geometrical conditions and adjusted priorities.
7. The property window displays all the parameters that are relevant to the specific type of connection.
8. Adjust the parameters of the generated connection in the property window.
9. Clear the selection to finish the design of the particular connection.

## Defining a new frame connection in strong axis

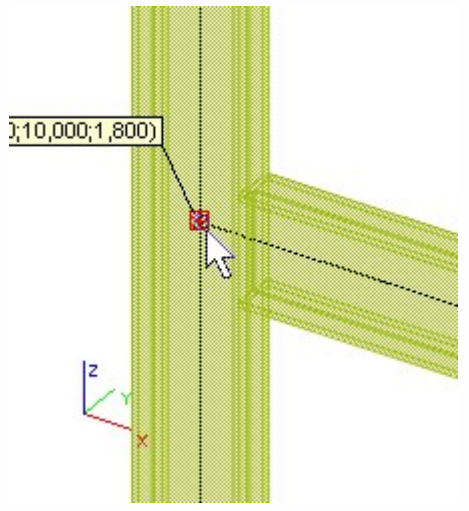
The procedure for the definition of a new frame connection in strong axis will be shown on an example of a horizontal beam attached to a vertical column in the middle of the column height. However, other configurations are possible as well (e.g. frame corner, etc.).

### *The procedure to define a new frame connection in strong axis*

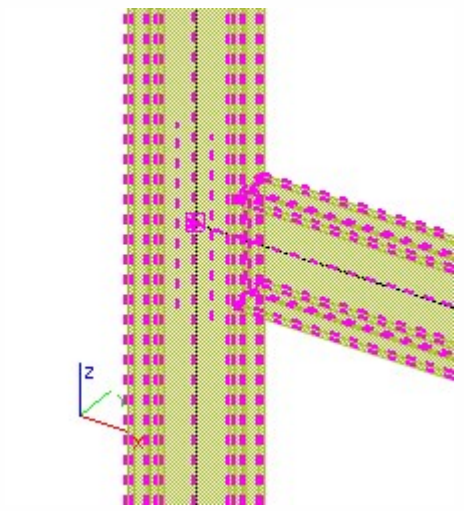
Let's have a column with a beam attached to it in the middle of the column height.



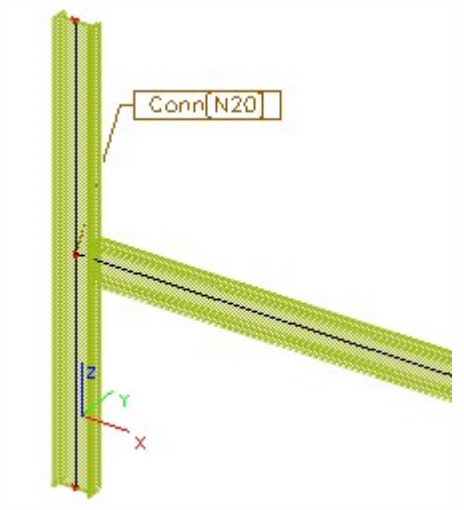

Open service Steel. Start function Connections > Frame bolted/welded – strong axis. Follow the instructions on the command line and select the point of connection.




All the beams that pass the selected node are selected. If required, unselect unnecessary beams.



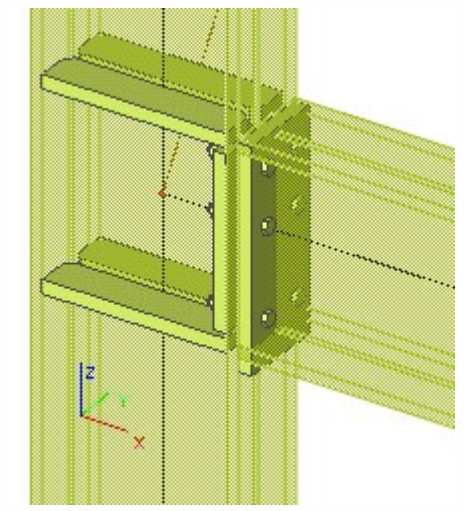

Press [Esc] to complete the action. The connection is generated in the selected node and a connection symbol is drawn on the screen.




Use the property window to define all required parts of the new connection.

| Side ->[B16] | |
|---|---|
| Connection type | Frame bolted |
| End plate | ☒ ... |
| Backing plate | ☒ ... |
| Bolts | ☒ ... |
| Top stiffener | ☒ ... |
| Bottom stiffener | ☒ ... |
| Diagonal stiffener | ☐ |
| Web doubler | ☐ |
| Update stiffness | ☐ |
| Calculation type | for loadcase/cc |
| Output | Normal |
| Length for stiffn... | 4,000 |

The connection in the graphical window is redrawn to reflect your settings.



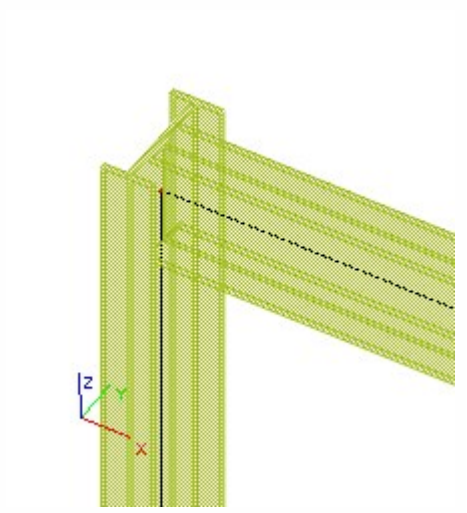

Note: Remember that a linked node must exist in the selected node where connection is to be defined.

## Defining a new frame connection in weak axis

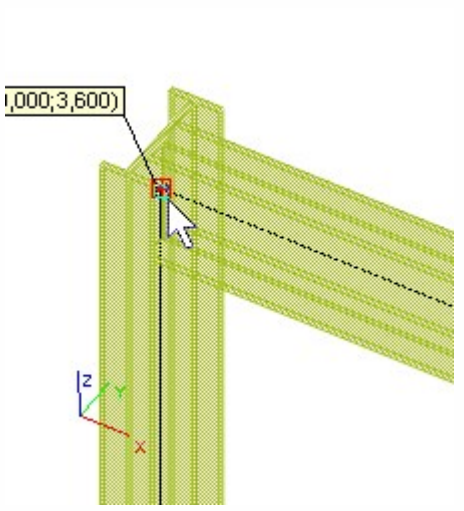
The procedure for the definition of a new frame connection in weak axis will be shown on an example of a horizontal beam attached to the head of a vertical column. However, other configurations are possible as well.

### *The procedure to define a new frame connection in weak axis*

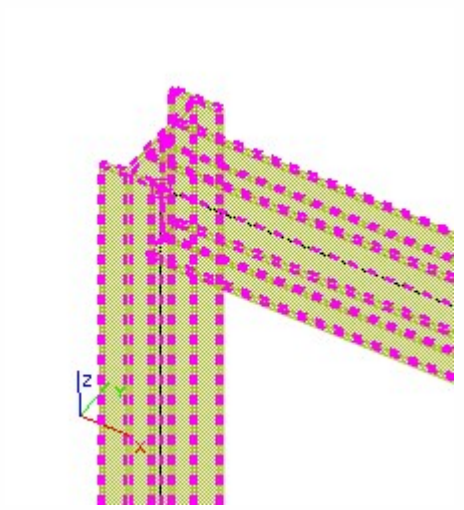
Let's have a column with a beam attached to its top end.



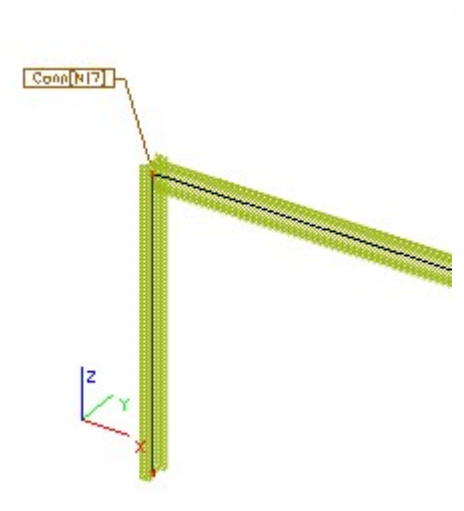

Open service Steel. Start function Connectuions > Frame bolted/welded – weak axis. Follow the instructions on the command line and select the point of connection.




All the beams that pass the selected node are selected. If required, unselect unnecessary beams.



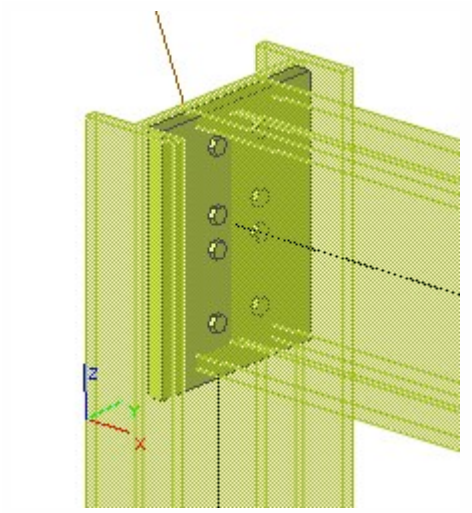

Press [Esc] to complete the action. The connection is generated in the selected node and a connection symbol is drawn on the screen.




Use the property window to define all required parts of the new connection.

| Side ->[B17] | |
|---|---|
| Connection type | Frame bolted |
| End plate | ☒ ... |
| Bolts | ☒ ... |
| Update stiffness | ☐ |
| Calculation type | for loadcase/cc |
| Output | Normal |
| Length for stiffn... | 4,000 |
| Weld | ... |

The connection in the graphical window is redrawn to reflect your settings.



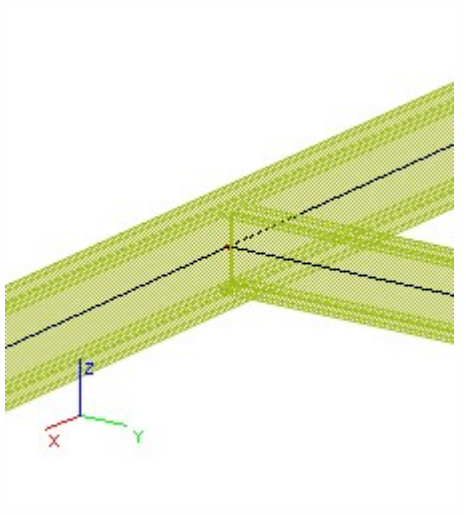
Note: Remember that a linked node must exist in the selected node where connection is to be defined.

## Defining a new grid pinned connection

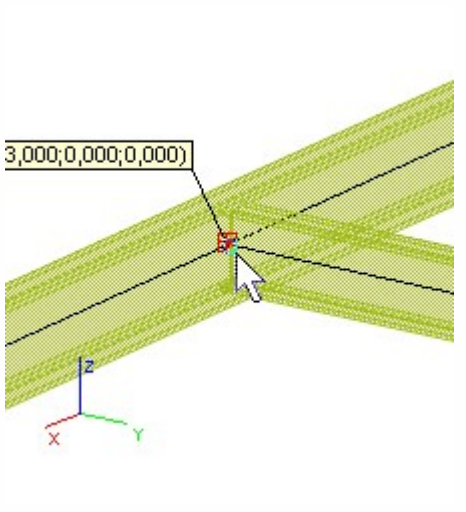
The procedure for the definition of a new grid pinned connection will be shown on an example of a horizontal beam attached to the second horizontal beam in the middle of its length. However, other configurations are possible as well.

### *The procedure to define a new grid pinned connection*

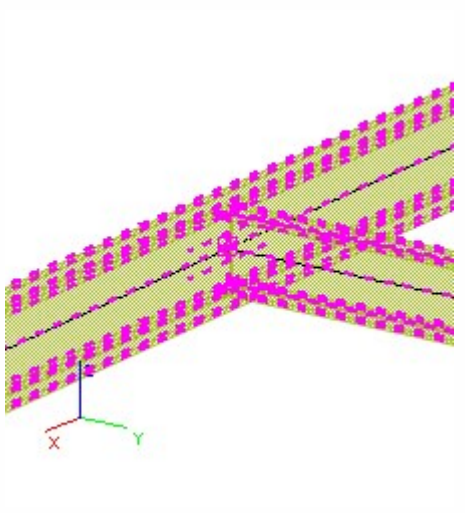
Let's have two intersecting (joining) horizontal beams.



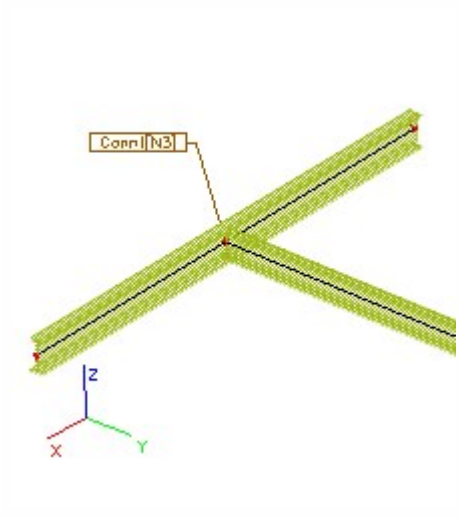

Open service Steel. Start function Connections > Grid pinned. Follow the instructions on the command line and select the point of connection.




All the beams that pass the selected node are selected. If required, unselect unnecessary beams.



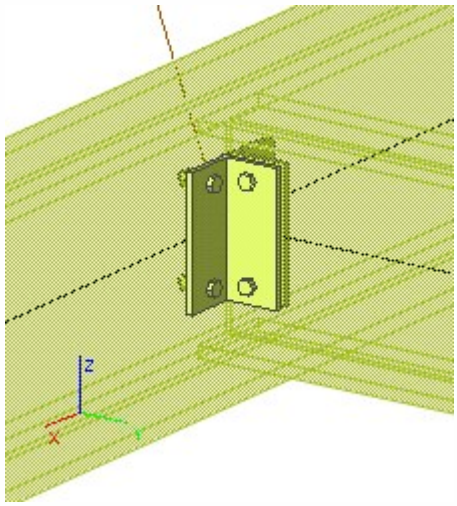

Press [Esc] to complete the action. The connection is generated in the selected node and a connection symbol is drawn on the screen.




Use the property window to define all required parts of the new connection.

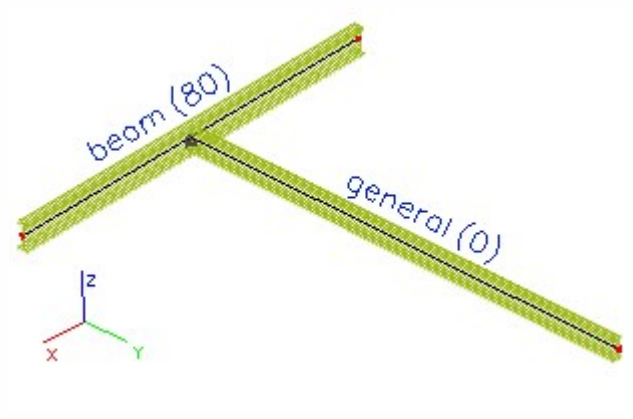
| Side ->[B2] | |
|---|---|
| Connection type | Grid pinned |
| Element type | Cleat |
| Cleat | ☒ ... |
| Bolts | ☒ ... |
| Calculation type | for loadcase/cc |
| Output | Normal |
| Beam notch | ... |
| Weld | ... |

The connection in the graphical window is redrawn to reflect your settings.

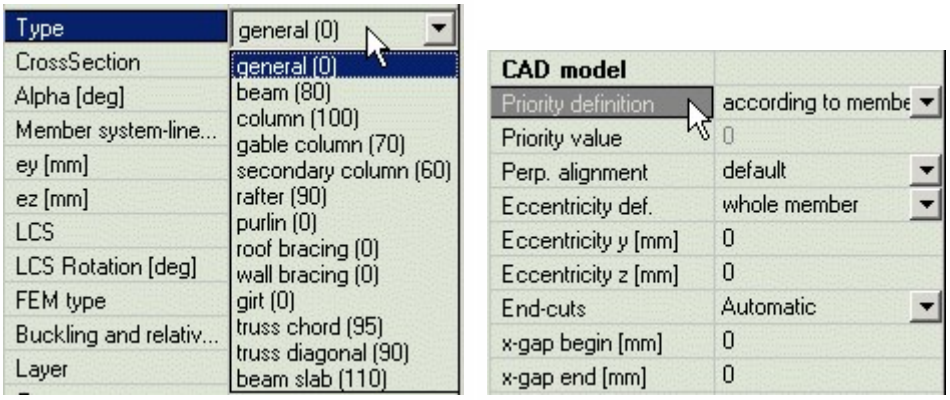



Note: Remember that a linked node must exist in the selected node where connection is to be defined.

Note: The beam that passes the connection must have HIGHER priority than the beam that ends in the connection. E.g. like below:




The priority can be adjusted in the property dialogue of each beam in the field Type or in group CAD model:

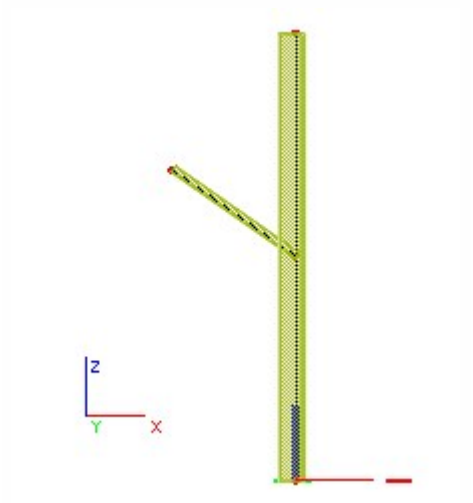



## Defining a new bolted diagonal connection

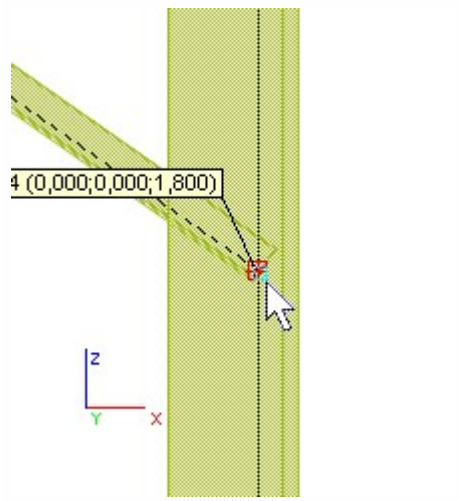
The procedure for the definition of a new bolted diagonal will be shown on an model example of a vertical column with a diagonal attached to it. However, other – more realistic – configurations are possible as well.

### *The procedure to define a new bolted diagonal connection*

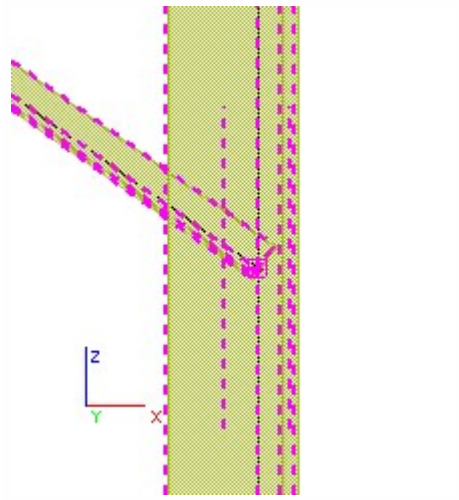
Let's have the model situation.



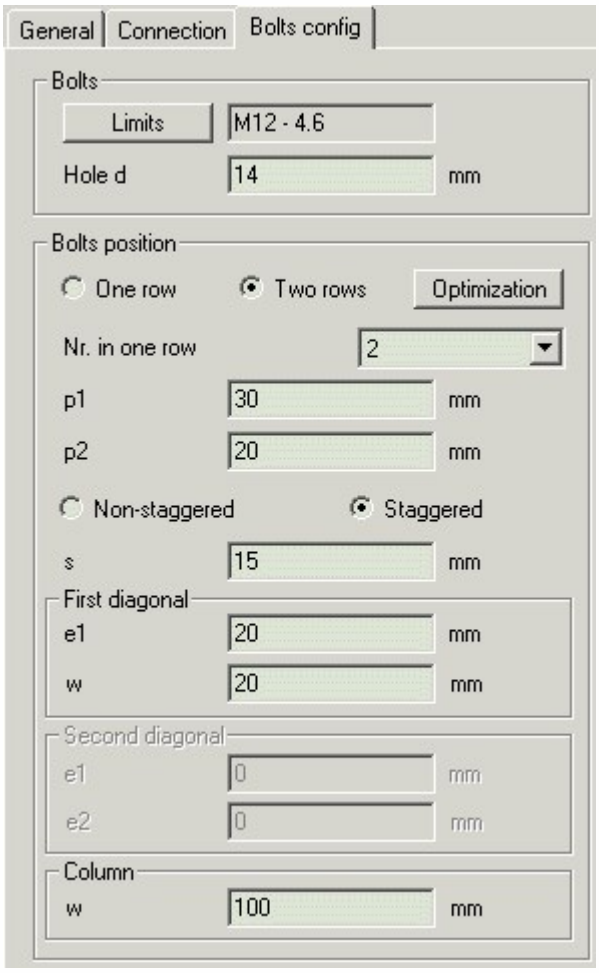
Open service Steel. Start function Connectuions > Bolted diagonal. Follow the instructions on the command line and select the point of connection.

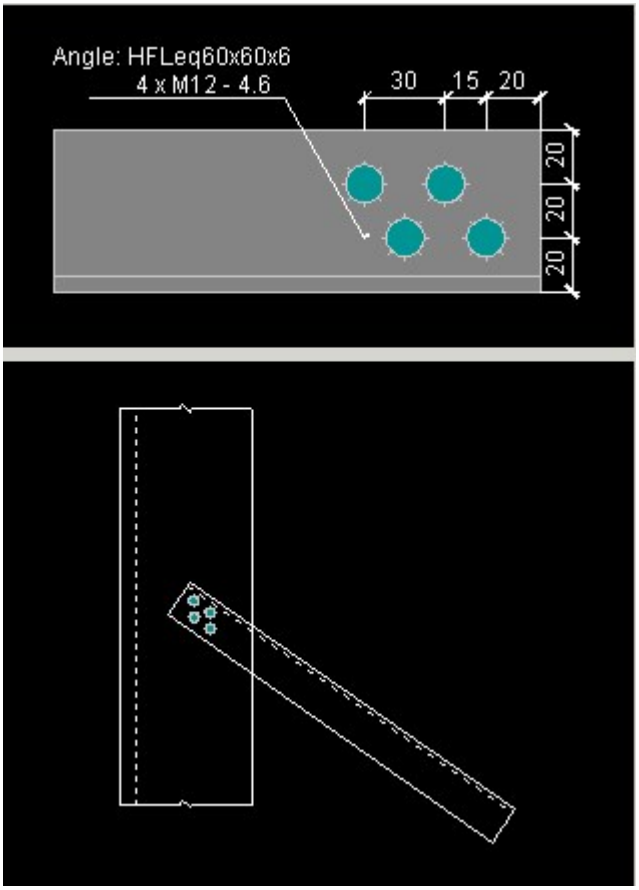



All the beams that pass the selected node are selected. If required, unselect unnecessary beams.

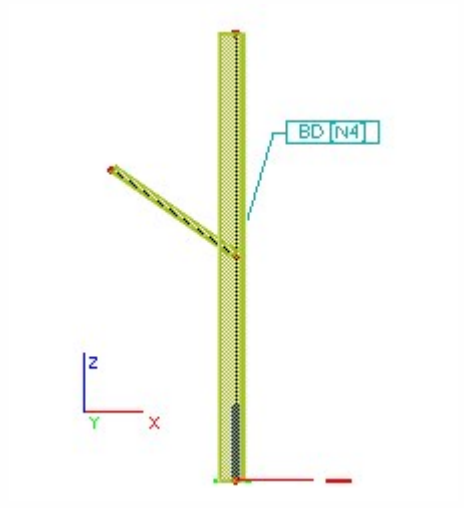



Press [Esc] to complete the action. The Bolted connection dialogue is opened on the screen. Make necessary adjustments.

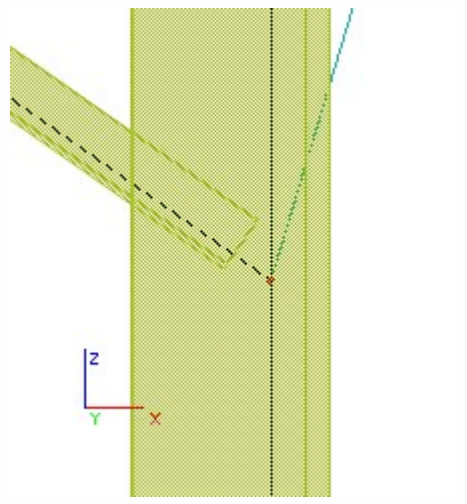

General
Connection
Bolts config
Bolts
Limits
M12 - 4.6
Hole d
14
mm
Bolts position
One row
Two rows
Optimization
Nr. in one row
2
p1
30
mm
p2
20
mm
Non-staggered
Staggered
s
15
mm
First diagonal
e1
20
mm
w
20
mm
Second diagonal
e1
0
mm
e2
0
mm
Column
w
100
mm




The connection is generated in the selected node and a connection symbol is drawn on the screen.




The connection in the graphical window is redrawn to reflect your settings.




Note: Remember that a linked node must exist in the selected node where connection is to be defined.

## Specifying the parameters of connection

Once a connection is created, its parameters may be displayed in the property window, adjusted and modified in order to satisfy all the requirements laid on the particular connection.

The parameters, of course, depend on the connection type that is being edited. Knee connection will have different parameters in the table than e.g. plate-to-plate connection. If more that one connection is edited at the same time (i.e. more than one connection is selected), the intersection of the parameters common to all selected connections is displayed in the property window.

The property table also hides all the "illogical" parameters. For example, unless the user specifies that there are bolts in the connection, it is not possible to select their length, diameter and other parameters.

In order to make the design of connection a straightforward action, the parameters are sorted in separate groups. The user may either open or close the group in the property window. It may be a good practice to close those groups whose design has been finished. Thus the list of parameters in the property window becomes shorted and clearer.

### Overview of connection parameter groups

The following overview presents the groups of parameters that may appear in the property window. Some groups of parameters or some particular parameters in groups may be available only for specific connection type or types.

### Basic (ungrouped) parameters

| | |
|---|---|
| Name | Specifies the name that appears in outputs. |
| Type of loads | Specifies the "type of load" that is used for calculations (checking) of the connection. E.g. load cases or load case combinations may be selected. |
| Frame type | Specifies the type of the frame. The frame may be with bracing or without it. |
| Connection geometry | Informs the user about the geometrical type of the connection. |

### Side => [Beam name]

Several beams may come to the connection. Each of the beams has got an individual Side group in the table.

The parameters in the group define which particular parts form the particular connection. The contents of the group may vary according to which parameters have been already specified in the group.

Possible parameters in the Side group are:

| | |
|---|---|
| Connection type | Either frame bolted or frame welded connection can be designed. |
| End plate | Specifies that an end plate is used. |
| Backing plate | Specifies that a backing plate is used. |
| Bolts | Specifies that bolts are used. |
| Top stiffener | Specifies that a top stiffener is used. |
| Bottom stiffener | Specifies that a bottom stiffener is used. |
| Diagonal stiffener | Specifies that a diagonal stiffener is used. |
| Web doubler | Specifies that the web of the column is stiffened by a web doubler. |
| Update calculation | If this option is ON, the program calculates the resistance and stiffness of the connection and displays them in the table at the end of Side group. |
| Calculation type | Specifies the load conditions in the connection. If the type is set to "for load cases and combinations", the program performs also the checking of the connection. And again the results are shown in the table at the end of Side group. |
| Output | Each connection may be "exported" into the document. Option Output then specifies the extent of the output report generated in the document. |
| Length for stiffness classification | Specifies the length for stiffness classification. |
| Update stiffness | Automatically calculates the stiffness of connection. |
| Element type (for Frame Pinned) | Selects type of plate for frame pinned connection. |
| Pinned plate (for Frame Pinned) | Specifies that a pinned plate is used. |
| Cleat (for Frame Pinned) | Specifies that a cleat is used. |
| Save to expert database | Saves the connection to expert database. |
| Load from expert database | Loads a connection from expert database. |
| Beam notch (for Grid Pinned) | The shape and size of the cleat. |
| Node (for Bolted Diagonal) | Informs about the node where the connection is defined. |
| Gusset material (for Bolted Diagonal) | Defines the gusset material. |
| Edit Bolted Diagonal (for Bolted Diagonal) | Provides for editing of this type of connection. |
| Haunch | Adds a haunch to the connected beam. This haunch:<br>• affects the check of the connection, |

| | |
|---|---|
| | • is taken into account when calculating the substitute" stiffness of he connection (if option Update stiffness is ON),<br>• is ignored in the calculation model itself (unlike the haunch defined by means of function Structure > Haunch. |

## End plate

This group of parameters specifies properties and dimensions of the end plate welded to the connected beam.

## Backing plate

This group of parameters specifies properties and dimensions of the backing plate.

## Bottom stiffener

This group of parameters specifies properties, shape and dimensions of the bottom stiffener.

## Top stiffener

This group of parameters specifies properties and dimensions of the top stiffener.

## Diagonal stiffener

This group of parameters specifies properties and dimensions of the diagonal stiffener.

## Web doubler

This group of parameters specifies properties, type and dimensions of the web doubler. It also automatically calculates the size of the element.

## Pinned plate

This group of parameters specifies properties, type and dimensions of the pinned plate.

## Cleat

This group of parameters specifies properties, type and dimensions of cleat.

## Bolts

Here, the bolt assembly, pattern, distance and other parameters concerning bolts are specified.

## Stiffener between bolt rows

This group of parameters specifies properties of the stiffener between bolt rows.

## Weld

This group specifies parameters of weld used in the connection.

## Concrete data

This group summarises properties of concrete block to which a steel beam is fixed by means of anchors.

## Anchor data

The Anchor data group provides for input of anchor type, shape and properties.

Example:

Let's assume a standard knee connection of a column and a beam. There will be only one Side group in the table. It will contain parameters:

- end plate,
- top stiffener,
- bottom stiffener,
- diagonal stiffener,
- web doubler.

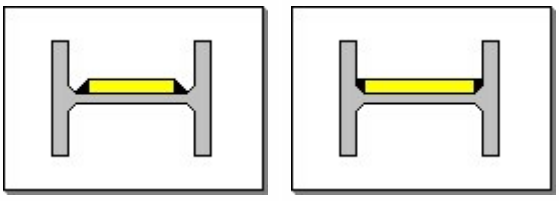
As soon as the user inserts an end plate to the connection, two more parameters are added to the group:

- backing plate,
- bolts.

Further let's assume that the user specifies the bolts in the connection. At that moment the connection becomes "realistic" and new items are added to the group:

- update calculation,
- calculation type,
- output.

Bubble help: The editing and adjusting of individual connection parameters is quite easy thanks to a sophisticated feature implemented in the property window. Whenever the user places the mouse cursor over a value cell in the table, the program shows a "bubble" help that brings a simple drawing explaining the meaning of the particular parameter. For example, the pictures below explain the difference between a small and large web doubler.



# Expert System

For Frame connections (bolted, welded and pinned), a connection database is available. The contents of this database consists of some pre-defined connections and can hold user defined connections. Besides the geometrical data, the capacity properties and the stiffness properties for a given configuration are saved in this database. The capacity and stiffness values are based on the ultimate limit state of the joint.

During the input of the connection data, this data can be searched in the expert database.

By means of various options, the user can customise and filter the search algorithm in the database.

## Contents

## The predefined connections

The predefined connections are based on the following tables:

| |
|---|
| Bemessungshilfen für profilorientiertes Konstruieren<br><br>Auflage 1997<br><br>Stahlbau-Verlagsgesellschaft mbH Köln |
| Stahlbau Kalender 1999<br><br>Bemessungshilfen für nachgiebige Stahlknoten mit Stirnplattenanschlüssen<br><br>Ernst & Sohn, DSTV, 1999, Berlin |

The capacity data are introduced as table values.

A supplementary set for predefined connections is generated by SCIA Engineer. The capacity data are introduced as calculated values.

The predefined connections are locked, and cannot be modified.

To reduce the number of data in the database, the predefined connections are based on beam properties. This implies that the proposed unity check during the search algorithm can be different from the calculated unity check.

## User defined connections

A second part of the database consists of user-defined data, which he can input and delete. For this data, both beam and column data are considered. The capacity data are introduced as calculated values.

## Load from expert database

For each relevant side the best-fitted connection data is represented by its connection name and the graphical representation. The best fitting result is the connection with the largest unity check but lower than 1.

[OK] button accepts the proposed connections; [Cancel] button ignores the proposed connections.

A complete list of the matching connections is given.

This connection list is based on the following search criteria:

- the joint type (welded, bolted, pinned) is matching,
- the geometrical type (beam-column, beam-beam, base plate) is matching,
- the angle between column and beam is between given limits (see chapter Adjusting the default parameters),

- the beam properties (geometry and material) are between given limits (see chapter Adjusting the default parameters),
- the column properties (geometry and material) are between given limits (see chapter Adjusting the default parameters),
- check for local constraints ( for example : haunch at the top is not always possible for a knee connection, check for admissible pitch, …).

If these conditions are met, the connection data are retrieved and sorted in unity check order or in priority order (see chapter Adjusting the default parameters).

For each entry, the connection name, the unity check, the position (normal or inverse), the bolt grade, the source name and the priority are present.

Example

IH3E/IPE270/16/20 0.99 + DSTV 10.9 1

| | Example |
|---|---|
| Connection name | IH3E/IPE270/16/20 |
| Unity check | 0.99 |
| Position | + |
| Source | DSTV |
| Bolt grade | 10.9 |
| Priority | 1 |

For welded and bolted beam-column and beam-beam connections, the unity check is based on the moment capacity. For bolted base plate connections, the unity check is based on moment capacity and normal force capacity. For pinned connections, the unity check is based on shear force capacity.

The capacity of the connection can be based on the calculated value, or on the table value or on both values (see chapter Adjusting the default parameters).

A selected connection can be modified by selecting the proper entry in the list.

## Saving the connection into the expert database

Each side can be saved to the connection database. The user has to specify a unique name for the connection. The default name of the connection is composed as follows:

AA/B/CCCCCC/DDDDDD/EE/FFFFFF/G

| | | |
|---|---|---|
| AA | Geometrical configuration | BC : beam-column<br>PP : plate-to-plate<br>BP : base plate |
| B | Joint type | B : bolted<br>W : welded<br>P : pinned |
| CCCCCC | Beam section | |
| DDDDDD | Column section | |
| EE | Endplate thickness (if bolted)<br>Pinned type (if pinned) | |
| FFFFFF | Bolt diameter and bolt grade (if bolted) | |
| G | Number | |

Examples:

BC/B/IPE270/HEA260/20/M16-10.9/1

Bolted beam-to-column connection between an IPE270 beam and a HEA260 column with endplate thickness = 20 mm and bolt M16 (10.9).

PP/B/IPE270/30/M20-10.9/1

Bolted plate-to-plate connection on an IPE270 beam with endplate thickness = 30 mm and bolt M20 (10.9).

BP/B/HEA260/15/M16-10.9/1

Bolted base plate connection on a HEA260 column with base plate thickness = 15 mm and anchor M16 (10.9).

BC/W/IPE270/HEA260/1

Welded beam-to-column connection between an IPE270 beam and a HEA260 column.

PP/W/IPE270/1

Welded plate-to-plate connection on an IPE270 beam.

BC/P/IPE270/HEA260/T/1

Pinned beam-to-column connection between an IPE270 beam and a HEA260 column using pin type 1.

## Defining the bolt assembly

By default, the user selects from an excessive catalogue of prepared bolt assemblies. However, if required, he/she may define his/her own bolt assembly or edit the existing one. The bolt assembly specifies which bolts are used with which nuts and washers and defines properties of them.

### Bolts

| | |
|---|---|
| Selected bolt | Selects the particular bolt. |
| Type | Chooses either normal or pre-stressed bolt. |
| Bolt grade | Specifies the grade. |
| Ultimate tensile strength | Informs the user about the ultimate tensile strength of the selected bolt. |
| Bolt length | Defines the way the bolt length is defined:<br>• standard length<br>• grip + delta (common in the U.S.) |
| Length | Specifies the length of the bolt. |
| Grip limit - lower | Specifies the lower grip limit. |
| Grip limit – upper | Specifies the upper grip limit. |

### Nuts

| | |
|---|---|
| Selected nut | Selects the particular nut. |
| Nut grade | Specifies the grade. |

### Washers

| | |
|---|---|
| Washer at head | Says whether the washer is present at the head of the bolt. |
| Washer at nut | Says whether the washer is present at the nut. |
| Selected washer | Selects the particular washer. |

#### *The procedure for the definition of bolt assembly*

1. Have the connection properties displayed in the property window:
   1. this happens during the definition of a new connection,
   2. or when an existing connection has been selected for editing.
2. Make sure that item Bolts is selected in the connection property table.
3. In the required table group Bolts (there may be several Bolts groups in the table, one group per each beam in the connection) press the button at the right hand side of item Selected bolt assembly ( 

 ).
4. The Bolt assemblymanager opens on the screen (The manager is one of many SCIA Engineer managers and its operation is unified throughout the whole SCIA Engineer environment.).
5. Define a new bolt assembly or edit the existing one as required.
6. Close the Bolt assembly manager.
7. Use the new bolt assembly where required.

## Selecting the bolts

SCIA Engineer offers a long list of common bolts. The library contains all necessary information which is crucial for the proper design of connections.

If necessary, the user may edit the data related to bolts in the Bolts manager. The manager is one of many SCIA Engineer managers and its operation is unified throughout the whole SCIA Engineer environment.

#### *The procedure for definition of bolt*

1. Invoke the Bolt assembly manager and have it displayed on the screen.
2. Press the button at the utmost right end of item Selected bolt.
3. This button opens the Bolts manager on the screen.
4. Define a new bolt or edit any of pre-defined ones.
5. Close the Bolts manager.

### Selecting the nuts

SCIA Engineer offers a long list of commonly used nuts. The library contains all necessary information which is crucial for the proper design of connections.

If necessary, the user may edit the data related to nuts in the Nuts manager. The manager is one of many SCIA Engineer managers and its operation is unified throughout the whole SCIA Engineer environment.

*The procedure for definition of nut*

1. Invoke the Bolt assembly manager and have it displayed on the screen.
2. Press the button at the utmost right end of item Selected nut.
3. This button opens the Nuts manager on the screen.
4. Define a new nut or edit any of pre-defined ones.
5. Close the Nuts manager.

### Selecting the washers

SCIA Engineer offers a long list of commonly used washers. The library contains all necessary information which is crucial for the proper design of connections.

If necessary, the user may edit the data related to washers in the Washers manager. The manager is one of many SCIA Engineer managers and its operation is unified throughout the whole SCIA Engineer environment.

*The procedure for definition of washer*

1. Invoke the Bolt assembly manager and have it displayed on the screen.
2. Press the button at the utmost right end of item Selected washer.
3. This button opens the Washers manager on the screen.
4. Define a new washer or edit any of pre-defined ones.
5. Close the Washers manager.

## Editing the existing connection

### Modifying the parameters of connection

Generally speaking, connection defined in SCIA Engineer is nothing more than an extra information saved with the structure model. From this point of view, it can be compared to e.g. load, support, mass, etc. And similarly to the named entities the connections belong to the Additional data.

Consequently, any connection can be easily modified using the same way as additional data are edited.

*The procedure for the modification of parameters of connection*

1. Select the connection or connections that should be modified.
2. The intersection of properties for the selected connections is displayed in the Property window.
3. Change the parameters as required.

4. The change is automatically applied.
5. Clear the selection.

If only a single connection should be modified and the user would prefer to see a regular modal dialogue on the screen, an alternative approach may be used.

***The alternative procedure for editing of connection***

1. Position the mouse cursor over the connection that should be modified.
2. Click the right mouse button.
3. The graphical window pop-up menu appear on the screen.
4. Select function Edit properties.
5. The property dialogue for the selected connection is opened in a separate modal dialogue.
6. Change any parameters you need to modify.
7. Confirm the changes with button [OK].
8. The operation is completed.

## Copying the connection

Generally speaking, connection defined in SCIA Engineer is nothing else than an extra information saved with the structure model. From this point of view, it can be compared to e.g. load, support, mass, etc. And similarly to the named entities the connections belong to the Additional data.

Consequently, any connection can be copied to another node the same way as additional data are copied.

***The procedure for copying of connections***

1. Select the connection that is to be copied.
2. Icon Copy add data (  ) becomes accessible on toolbar Geometrical manipulations.
3. Click the icon.
4. Define the target position for the copied connection.
5. The selected connection is copied into the new location.
6. If required, select another target positions.
7. Press [Esc] to and the function.

The function for copying of additional data is also accessible via the window pop-up menu. Instead of clicking the button on the Geometrical manipulations toolbar, it is possible to use the alternative approach.

***The alternative procedure for copying of a connection***

1. Select the connection that is to be copied.
2. Position the mouse cursor outside any entity on the screen.
3. Click the right mouse button to invoke the pop-up menu.
4. Select function Copy add data ((

 )).

5. Follow the final steps of the procedure described above.

There is also an alternative to the above mentioned procedure.

***The second alternative procedure for copying of a connection***

1. Position the mouse cursor over the connection you want to copy.
2. Click the right mouse button.
3. The pop-up menu appears on the screen.
4. Select function Copy add data ((  )).
5. The function will treat the single entity – the one over which the mouse cursor was positioned when the mouse button was clicked.
6. Define the target position for the copied connection.
7. The selected connection is copied into the new location.
8. If required, select another target positions.
9. Press [Esc] to and the function.

## Deleting the connection

Generally speaking, connection defined in SCIA Engineer is nothing more than an extra information saved with the structure model. From this point of view, it can be compared to e.g. load, support, mass, etc. And similarly to the named entities the connections belong to the Additional data.

Consequently, any connection can be deleted the same way as additional data are deleted.

***The procedure for deletion of connections***

1. Select connections that should be removed.
2. Start function Delete:
   1. either use menu function Modify > Delete,
   2. or invoke the window pop-up menu and here select function Delete.
3. A dialogue asking for your confirmation appears on the screen.
4. Confirm it.
5. The connections are deleted from the project.

# Checking the connection

## Fast checking

Fast checking of a selected connection can be made in the property table.

Once a connection is selected and its properties are shown in the property window (see chapter Modifying the parameters of connection), it is possible to display brief information about the bearing capacity of the connection.

Group Side (remember that each beam coming to the connection has got its own group Side) contains item Update calculation. When this item is ticked (selected), the program calculates automatically the design resistance of the connection and some other characteristics.

| | |
|---|---|
| Design moment resistance | Shows the overall moment resistance of the connection. |
| Design shear resistance | Shows the overall shear resistance of the connection. |
| Design normal resistance | Shows the overall normal resistance of the connection. |
| Result of stiffness, Sj | Displays the rotational stiffness of the connection, related to the actual moment Mj,Sd. |
| Result of stiffness, Sj,ini | Displays the rotational stiffness, related to Mj,Rd (without the influence of normal force). |
| Limit part | Tells which part is the limiting part of the connection. In other words, which part is the most "weak" part. |
| Limit part compression | Tells which part is the limiting part of the connection in compression. |

The above mentioned quantities are shown if Calculation type for the particular "side" of the connection is set to tension on top side or tension on bottom side. These two options mean that the program does not assume existence of results calculated in static analysis of the structure.

However, if option Calculation type is set to from load case / combination, the program takes the available results of static calculation and determines other result values related to the connection.

| | |
|---|---|
| Unity check M / MRd | Displays the result of unity check for moment. |
| Unity check V / VRd | Displays the result of unity check for shear force. |
| Unity check N / NRd | Displays the result of unity check for axial force. |
| Moment | Displays the actual moment acting in the connection. |
| Shear force | Displays the actual shear force acting in the connection. |
| Check – Moment | Displays the result of moment check. |
| Check – Shear force | Displays the result of shear force check. |

Note 1: In order to enable the program to show the basic checking results, i.e. the resistance and stiffness values, the connection must be already defined reasonably. That means that it must contain at least the most essential parts. For example, the knee connection must contain an end plate and bolts.

Note 2: In order to enable the program to show the results of complete check (i.e. the values from the second table), the condition from Note 1 must be fulfilled and the structure

model must have been already calculated and at least one result of that analysis must be available.

# Detailed checking

Once a connection is designed, the program is able to perform the checking of the connection and show the results. The results may be presented in a brief form directly in the property table of the connection, or they may be sorted out in a readable tabular style in the document.

| | |
|---|---|
| brief | The output brings just the principal data related to the connection. |
| normal | The output contains the all the essential information about the connection and its checking. |
| detailed | The report consists of all available data related to the connection. |

Regardless of the output type, the result report is generated into the document or preview window of SCIA Engineer.

### *The procedure for creation of output in Preview window*

1. Select the connection or connections that should be checked.
2. The intersection of properties for the selected connections is displayed in the Property window.
3. Set item Output to the required value (brief / normal / detailed).
4. Call function Print / Preview table:
    1. either using function Print data > Print / Preview table located on Project toolbar,
    2. or using menu function File > Print data > Print / Preview table.
5. The program generates tables for selected connection (or connections) in the Preview window.

### *The procedure for creation of output in the Document*

1. Select the connection or connections that should be checked.
2. The intersection of properties for the selected connections is displayed in the Property window.
3. Set item Output to the required value (brief / normal / detailed).
4. Call function Table to document:
    1. either using function Print data > Table to document located on Project toolbar,
    2. or using menu function File > Print data > Table to document.
5. The program generates tables for selected connection (or connections) in the Document.

# Checking summary

Checking of a connection may be carried out once two basic conditions are met:

- the connection has been designed,
- the project has been successfully calculated and results are available.

The checking results in a brief table containing the essential data about the connection, its resistance and the load it is subject to.

### *The procedure for checking the connection*

1. Open service Steel.
2. Open branch Connections.
3. Select function Check.
4. In the Property window, adjust the required checking parameters.
5. Use function Print > Print / Preview table to display the results in the Preview window.

## Checking parameters

| | |
|---|---|
| Selection | The selection may set to:<br>All – all beams are checked<br>User – the user must select beams for the checking |
| Type of load | The checking may be carried out for load cases or for combinations. |
| Load case | Here the specific load case or combination may be selected. |
| Filter | Filter may be se to:<br>No – there is no filtering<br>Wildcard – the selection is given by the typed "wildcard expression", e.g. B*, BEAM1?, etc. |
| Values | Here the required quantity may be selected. |
| Extreme | This item tells which connections should be presented in the Preview Window (see below). |

## Extreme

| | |
|---|---|
| No / Node | All selected connections are checked and printed. |
| Global | All selected connections are checked, but only the one with extreme value is printed. |

The table with the results may look like:

| Steel | |
|---|---|
| **Node** | N2 |
| **Connection geometry** | Knee |
| **Connection type** | Frame bolted |
| **Calculation type** | for loadcase/combinations |
| **Design moment resistance Mrd [kNm]** | 12,22 |
| **Design shear resistance Vrd [kN]** | 18,13 |
| **Unity check M/MRd [-]** | 0,37 |
| **Unity check V/VRd [-]** | 0,00 |
| **Check M** | Connection satisfied |
| **Check V** | Connection satisfied |

Note: Once the Preview Window is opened and the results of connection checking are displayed in it, any change of the checking parameters leads to the regeneration of the Preview Window. There is no need to use function Print > Print / Preview table over and over again.

# Output of connections

## Screen visualisation

### Introduction to screen visualisation

A connection can be drawn on the screen both during the phase of design and when finished to the latest detail.

There are several view styles available in SCIA Engineer. Each style has its advantages and is useful for different purposes:

- simplified view,
- rendered view,
- view with dimension lines,
- user-specific view.

### Simplified view

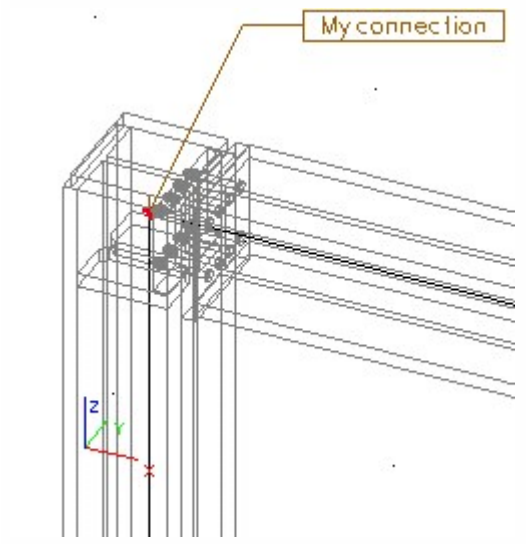
Simplified view (called also wired view) displays all the parts of the connection in the form of "wires" or separate lines. This style is advantageous for its speed. Very often it is also clear enough and can be recommended as the basic display style.

#### Simplified (wired) view of calculation model

***The procedure for the adjustment of simplified view***

1. Call menu function View > Set view parameters > Model of structure (this sets the view of the calculation model of the structure).
2. If the structure is displayed in rendered mode, click button [Render geometry] at the bottom side of the graphical window.
3. In the graphical window click the right mouse button to invoke the pop-menu.
4. Select function Set view parameters.
5. Select tab Entities.
6. In group of view parameters called Steel connections set the following parameters:
    1. tick the parameter Display,
    2. parameter Rendering set to value Wired.
7. Confirm with [OK] (this sets the simplified view of the connections).

Example

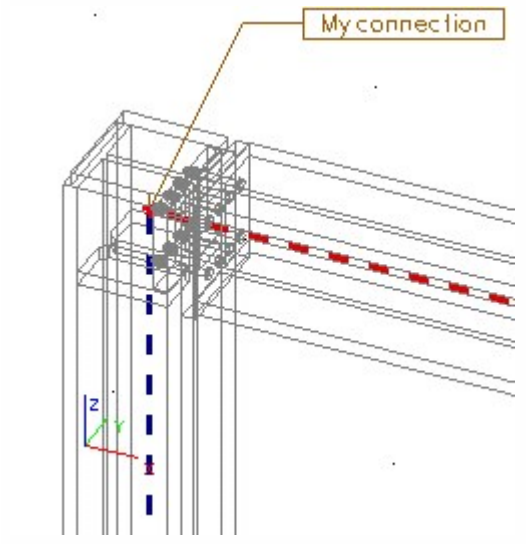



## Simplified (wired) view of CAD model

***The procedure for the adjustment of simplified view***

1. Call menu function View > Set view parameters > CAD model (this sets the view of the CAD model).
2. If the structure is displayed in rendered mode, click button [Render geometry] at the bottom side of the graphical window.
3. In the graphical window click the right mouse button to invoke the pop-menu.
4. Select function Set view parameters.
5. Select tab Entities.
6. In group of view parameters called Steel connections set the following parameters:
    1. tick the parameter Display,
    2. parameter Rendering set to value Wired.
7. Confirm with [OK] (this sets the simplified view of the connections in CAD model).

Example



# Rendered view

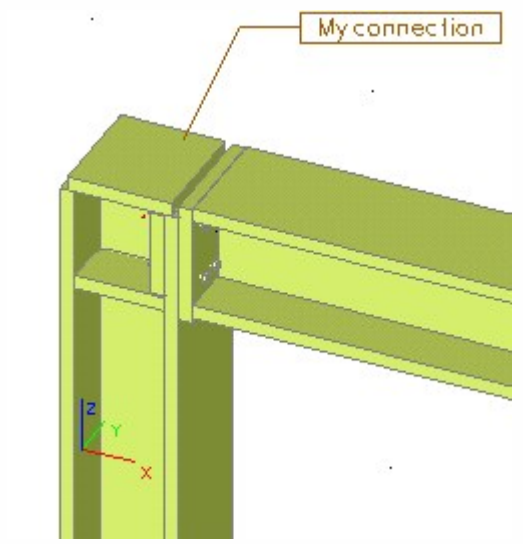
***Rendered view displays all the parts of the connection with full "visibility" of individual parts. In other words, those parts that are located behind others are invisible. The view most resembles of normal view of the structure in reality. This style is advantageous for its punctuality. It may be useful for design of complex details and for good visualisation of designed connections. It is also almost invaluable during presentations.***

## Rendered view of calculation model

***The procedure for the adjustment of rendered view***

1. Call menu function View > Set view parameters > Model of structure (this sets the view of the calculation model of the structure).
2. If the structure is displayed in wired mode, click button [Render geometry] at the bottom side of the graphical window.
3. In the graphical window click the right mouse button to invoke the pop-menu.
4. Select function Set view parameters.
5. Select tab Entities.
6. In group of view parameters called Steel connections set the following parameters:
   1. put the tick into the item Display,
   2. parameter Rendering set to value Rendered with edges.
7. Confirm with [OK] (this sets the rendered view of connections).

Example



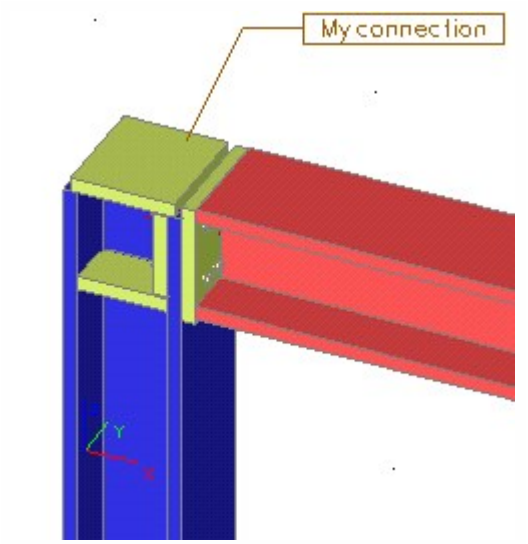

## Rendered view of CAD model

***The procedure for the adjustment of simplified view***

1. Call menu function View > Set view parameters > CAD model (this sets the view of the CAD model).
2. If the structure is displayed in wired mode, click button [Render geometry] at the bottom side of the graphical window.

3. In the graphical window click the right mouse button to invoke the pop-menu.
4. Select function Set view parameters.
5. Select tab Entities.
6. In group of view parameters called Steel connections set the following parameters:
   1. tick the parameter Display,
   2. parameter Rendering set to value Rendered with edges.
7. Confirm with [OK] (this sets the rendered view of the connections in CAD model).

Example

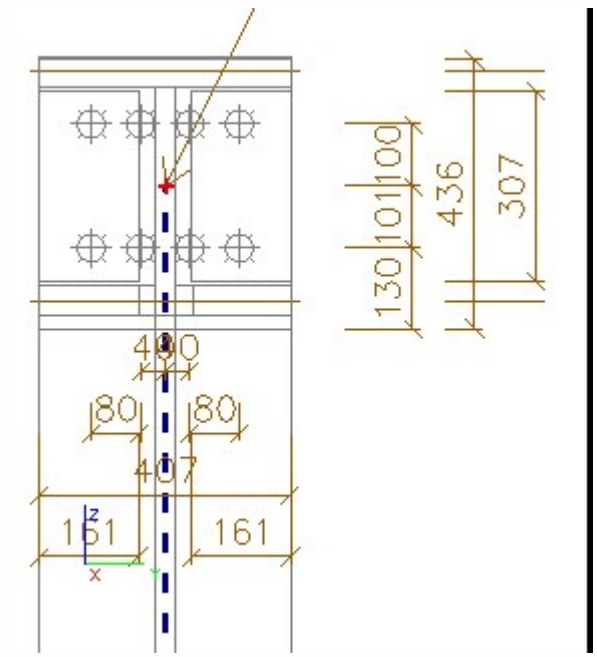



## Dimension lines

SCIA Engineer enables the user to display also dimension lines for individual connection parts. The dimension lines can be switched on in Set view parameters dialogue.

The style of dimension lines can be set in Setup > Dimension lines dialogue.

Example



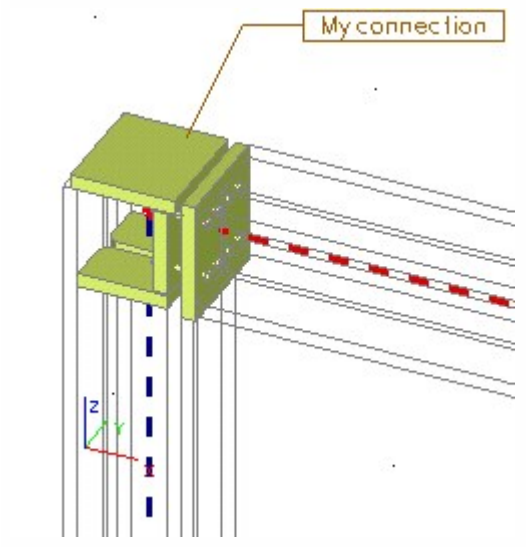

## View parameters

Connections, similarly to other SCIA Engineer entities have parameters that control the display style. These parameters are called view parameters. The user may adjust the view parameters almost arbitrarily in order to find such a display style that most reflects his/her needs, wishes, demands, or habits.

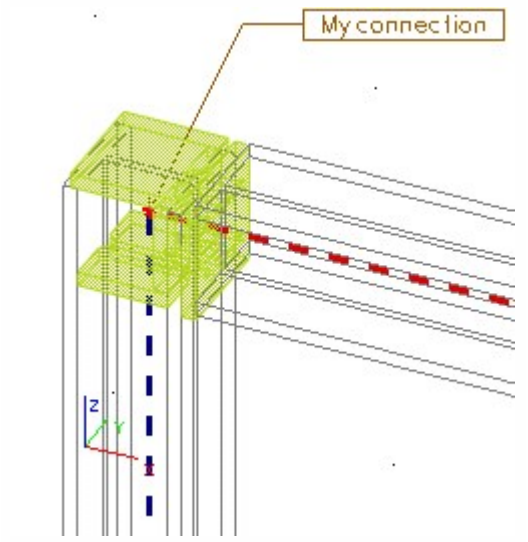
***The procedure for adjustment of view parameters for connections***

1. In the graphical window click the right mouse button to invoke the pop-menu.
2. Select function Set view parameters.
3. Select tab Entities.
4. Adjust the required parameters in group Steel connections.
5. Confirm with [OK].

Example of view where the connection is displayed in rendered mode and the structure in wired mode




Example of view where the connection is displayed in "transparent rendering" mode and the structure in wired mode




# Drawings

## Inserting a selected drawing into the Document

Any drawing of any connection that is just displayed on the screen may be inserted as a separate drawing into the Document.

***The procedure for the insertion of a connection drawing into the Document***

1. Adjust the drawing on the screen as required.
2. Call function Picture to document:
   1. using menu function File > Print picture > Picture to document,
   2. using function Print picture > Picture to document ( > ) on toolbar Project

3. Adjust the parameters of the picture.
4. Confirm with [OK].

## Inserting a selected drawing into the Picture gallery

Any drawing of any connection that is just displayed on the screen may be inserted as a separate drawing into the Picture gallery.

***The procedure for the insertion of a connection drawing into the Picture gallery***

1. Adjust the drawing on the screen as required.
2. Call function Picture to gallery:
   1. using menu function File > Print picture > Picture to gallery,
   2. using function Print picture > Picture to gallery (  >  ) on toolbar Project,
   3. using button [Pictureto gallery] (  ) located at the bottom scroll bar of the graphical window.
3. Type the name of the picture.
4. Confirm with [OK].

## Printing a selected drawing of connection

Any drawing of any connection that is just displayed on the screen may be printed as a separate drawing.

***The procedure for the printing of a connection drawing***

1. Adjust the drawing on the screen as required.
2. Call function Print picture:
   1. using menu function File > Print picture > Print picture,
   2. using function Print picture > Print picture (  >  ) on toolbar Project
3. ***Adjust the parameters of the picture.***
4. Confirm with [OK].

## Saving a selected drawing into external file

Any drawing of any connection that is just displayed on the screen may be saved as a separate drawing into an external graphical file.

***The procedure for the exporting of a connection drawing***

1. Adjust the drawing on the screen as required.
2. Call function Save picture to file:
3. using menu function File > Print picture > Save picture to file,
4. using function Print picture > Save pictureto file (

 >  ) on toolbar Project

5. Adjust the parameters of the picture.
6. Confirm with [OK].

## Generating the drawings by Picture wizard

It is usual practice that a whole set of drawings is necessary to describe fully all parts of the connection:

- various views of the connection,
- drawings of individual parts such as backing plate, stiffener, etc.

SCIA Engineer enables the user to call a sophisticated tool called Picture wizard that does all the work for the user.

***The wizard asks the user to decide which drawings are required and what their style should be, and then generates a set of specified pictures.***

### The parameters of the Picture wizard

### Name Prefix

| | |
|---|---|
| Prefix of name | Specifies the base of picture name. |
| Scale | Determines the scale in which the picture is made. |

### Picture parameters

| | |
|---|---|
| Picture width | Specifies the width of the picture. |
| Picture height | Specifies the height of the picture. |
| Display mode | Specifies the mode of the drawings:<br>normal<br>rendered<br>wired<br>hidden lines off<br>hidden lines dashed<br>wired OpenGL |
| Text scale factor | This factor can be used to multiply the text size in order to adjust easily the drawing for various output devices. |
| Character set of texts | Specifies the character set (e.g. Western Europe, Eastern Europe, Greek, Russian, etc.) |

### Dimension line

| | |
|---|---|
| End-mark style | Specifies the style of end marks of dimension lines. |
| End-mark size | Specifies the size of end marks of dimension lines. |
| Text size | Specifies the size of text of dimension lines. |

## Other picture parameters

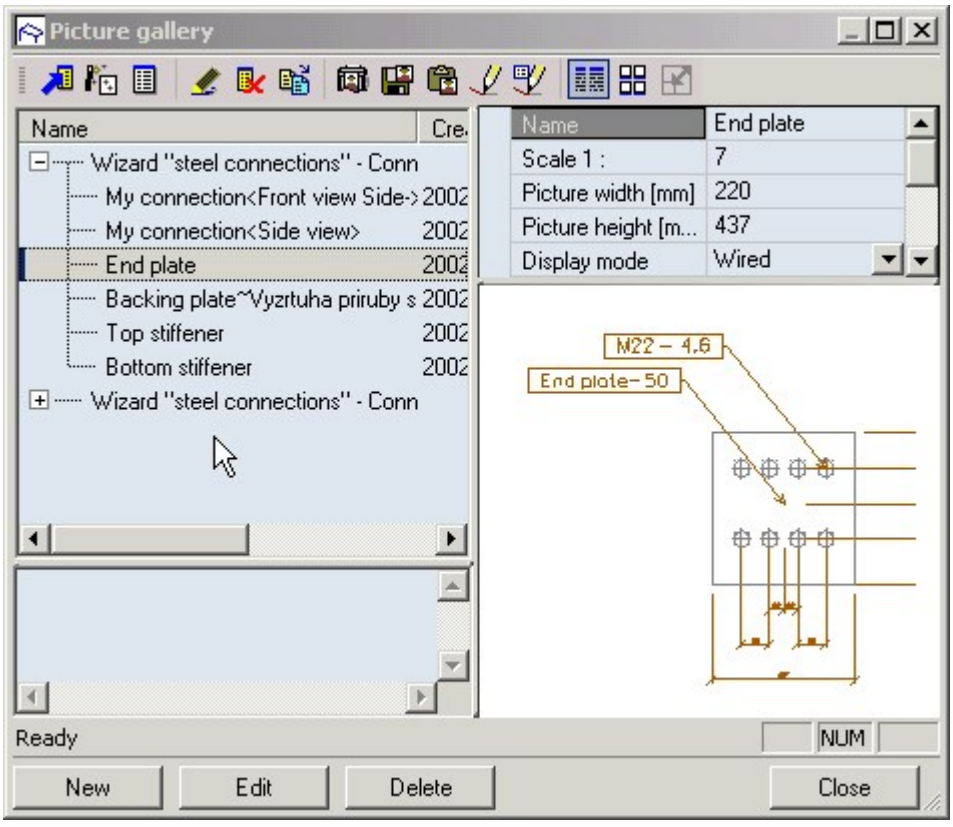
| | |
|---|---|
| Placement of name | Defines the position of drawing name on the sheet. |
| Name font size | Defines the font size for the drawing name. |

## Wizard parameters

| | |
|---|---|
| Draw connections | The user may specify which defined connections should be considered by the wizard:<br>• drawings are made for all connections defined in the project,<br>• drawings are made for selected drawings only. |
| View parameters | The drawings can be made using:<br>• view parameters of the active graphical window,<br>• view parameters assigned to CAD types. |
| Whole connection – Front view | If ON, front view of the connection is drawn. |
| Whole connection – Side view | If ON, side view of the connection is drawn. |
| Parts of connection | If ON, drawings of connection parts are generated as well. |

Example

The picture below shows the Picture gallery dialogue. The top left window lists the generated drawings for a particular connection. The bottom right window then shows the preview of selected picture.



# Document

## Inserting a connection drawing into the Document

Any drawing of any connection that is just displayed on the screen may be inserted as a separate drawing into the Document. Later the document may be edited in a way so that the final report looks as professional as possible.

***The procedure for the insertion of a connection drawing into the Document***

1. Adjust the drawing on the screen as required.
2. Call function Picture to document:
   1. using menu function File > Print picture > Picture to document,
   2. using function Print picture > Picture to document (  >  ) on toolbar Project
3. Adjust the parameters of the picture.
4. Confirm with [OK].

## Inserting a table with connection data into the Document

Data related to any connection that has been defined in the project may be inserted into the Document in the form of readable tables. Later the document may be edited in a way so that the final report looks as professional as possible.

***The procedure for the insertion of a table with connection data into the Document***

1. Adjust the drawing on the screen as required.
2. Call function Table to document:
   1. using menu function File > Print data > Table to document,
   2. using function Print data > Table to document on toolbar Project

# Relation to other modules

## Geometry and internal forces

Module Connections reads from the basic module of SCIA Engineer all the information about:

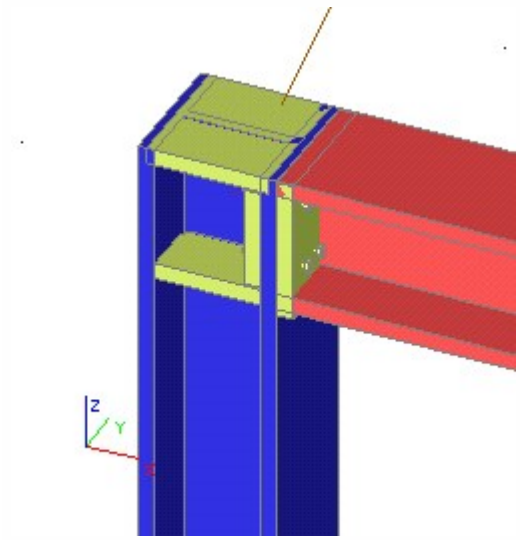
- geometry including cross-sections, materials, dimensions, etc.
- calculated internal forces (on condition that the calculation has been already carried out).

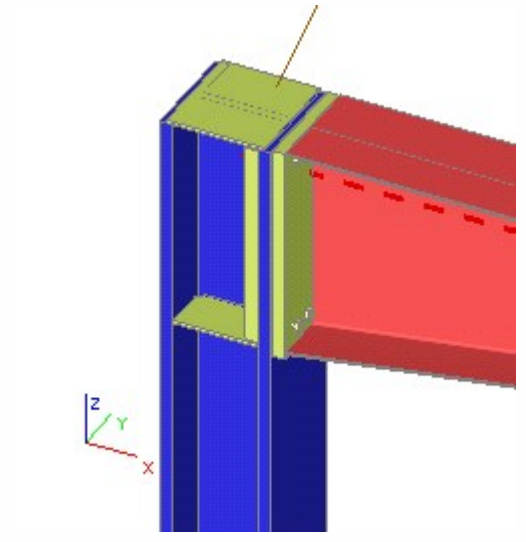
These data are used as the basis for the design and check of individual connections. The user does not have to take care about the proper definition of load that the particular connection is subject to. The program does it automatically.

## Recognition of haunches

If a haunch comes into the connection, the program is capable of recognising such a case. The design of the connection is then modified accordingly. There is no need to specify explicitly that a particular connection is a connection of beams with haunches.

The pictures below show the designed connection of a "normal" beam and of a beam with a haunch.

| | |
|---|---|
| Connection of "normal" beam |  |

| | |
|---|---|
| Connection of haunch beam |  |

# Check of stiffness

The connection is classified as rigid, pinned or semi-rigid according to its stiffness by using the initial rotational stiffness Sj,ini and comparing this with classification boundaries given in Ref.[1] Figure J.8.

The program calculates the real stiffness of the designed connection and displays it together with other results for the connection.

# Updating the stiffness of calculation model

SCIA Engineer module for design and checking of connections is capable of automatic determination of the connection stiffness. What's more, the program can also use this stiffness as an input data for the calculation of the model.

Thus, the following procedure of accurate analysis of a structure is possible:

1. creation of the structure model with default, rigid connections of individual members
2. calculation of the model and determination of "first-round" results
3. thorough design on connections (i.e. definition of end plates, stiffeners, bolts, welds, etc.)
4. automatic calculation of stiffness for individual connections
5. re-calculation of the whole model, this time with proper stiffness of individual connections
6. final review, checking and generation of printed documentation

## Switching the automatic update ON

In order to obtain automatically the stiffness of individual connections in the model, the user must ONLY check item Update stiffness located in group Side in the property table of the particular connection.

Item Update stiffness must be checked separately for each connection where the stiffness is required to be used in the calculation.

## Obtaining the results that reflect the connection stiffness

Once the Update stiffness item has been selected for required connections, the calculation of the model MUST be carried out once more.

This repeated calculation performs two tasks:

1. it generates hinges in appropriate joints and defines their stiffness according to the designed connection,
2. it performs calculation.

Note 1: The hinges defined automatically in selected connections, remain a permanent part of the calculation model. Even if the Update stiffness option is cancelled, the already defined hinges remain in the structure. If the user wants to remove them, he/she must do so manually.

Note 2: If the design of a particular connection has been changed after the re-calculation of the model, and item Update stiffness for that modified connection is ON, it is necessary to repeat the calculation once more in order to modify also the stiffness of the automatically generated hinge.

# References

## List of references

[1]

Eurocode 3 : Part 1.1.

Revised annex J : Joints in building frames

ENV 1993-1-1/pr A2

[2]

Eurocode 3

Design of steel structures

Part 1 - 1 : General rules and rules for buildings

ENV 1993-1-1:1992, 1992

[3]

P. Zoetemeijer

Bolted beam to column knee connections with haunched beams

Tests and computations

Report 6-81-23

Delft University of Technology, Stevin Laboratory, December 1981

[4]

P. Zoetemeijer

Een rekenmethode voor het ontwerpen van geboute hoekverbindingen met een kolomschot in de trekzone van de verbinding en een niet boven de ligger uitstekende kopplaat.

Rapport 6-81-4

Staalcentrum Nederland, Staalbouwkundig Genootschap, Juni 1982

[5]

Eurocode 3 : Part 1.1.

Annex L: Design of column bases

ENV 1993-1-1:1992

[6]

Eurocode 2

Design of concrete structures

Part 1: General rules and rules for buildings

ENV 1992-1-1:1991

[7]

Y. Lescouarc'h

Les pieds de poteaux articulés en acier

CTICM, 1982

[8]

Manual of Steel Construction

Load & Resistance Factor Design

Volume II : Connections

Part 8 : Bolts, Welds, and Connected Elements

AISC 1995

[9]

U. Portmann

Symbole und Sinnbilder in Bauzeichnungen nach Normen, Richtlinien und Regeln

Wiesbaden, Berlin : Bauverlag, 1979

[10]

Sprint Contract RA351

Steel Moment Connections according to Eurocode 3

Simple Design aids for rigid and semi-rigid joints

1992-1996

[11]

DIN18800 Teil 1

Stahlbauten : Bemessung und Konstruktion

November 1990

[12]

J. Rudnitzky

Typisierte Verbingungen im Stahlhochbau. 2. Auflage

Stahlbau-Verlags-GmbH-Köln 1979

[13]

H. Buchenau A.Tiele

Stahlhochbau 1

B.G. Teubner

Stuttgart 1981

[14]

F. Mortelmans

Berekening van konstructies Deel 2

Staal Acco

Leuven, 1980

[15]

Frame Design Including Joint Behaviour

Volume 1

ECSC Contracts n° 7210-SA/212 and 7210-SA/320

January 1997

[16]

F. Wald, A.M. Gresnigt, K. Weynand, J.P. Jaspart

Application of the component method to column bases

Proceedings of the COST C1 Conference

Ličge, Sept.17-19, 1998

[17]

F.C.T. Gomes, U. Kuhlmann, G. De Matteis, A. Mandara

Recent developments on classification of joints

Proceedings of the COST C1 Conference

Ličge, Sept.17-19, 1998

[18]

M. Steenhuis, N. Gresnigt, K. Weynand

Pre-design of semi-rigid joints in steel frames

COST C1 Workshop

Prague, October 1994

[19]

M. Steenhuis, N. Gresnigt, K. Weynand

Flexibele verbindingen in raamwerken

Bouwen met Staal 126

September/Oktober 1995

[20]

O. Oberegge, H-P Hockelman, L. Dorsch

Bemessungshilfen für profielorientiertes Konstruieren

3. Auflage 1997

Stahlbau-Verlagsgesellschaft mbH Köln

[21]

J. Wardenier

Hollow section joint

Delf university press 1982

[22]

Staalconstructies TGB 1990

Verbindingen

NEN6772, december 1991

[23]

Richtlijnen voor de berekening van buisconstructies-RB'82

Staalcentrum Nederland

Staalbouwkundig Genootschap, 1982

[24]

J. Mouty L. Petit

Manuel assemblages 1ičre partie, Vérification des assemblages soudés méthode simplifiée

Chambre syndicale des frabicants de tubes d'acier

Notice 1086, february 1983

[25]

M. Tournay

Manuel assemblages 2ičre partie, Dispositions constructives

Chambre syndicale des frabicants de tubes d'acier

Notice 1087, november 1980

[26]

Eurocode 3

Calcul de structures en acier

Anneexes KK et commantaires

Document de travail GT CCCA-EC3 "Assemblages"

[27]

Design of SHS welded joints

British steel corporation Tubes division

Publication TD297 / june 1985

[28]

J. Wardenier, Y.Kurobane, J.A. Packer, D. Dutta, N. Yeomans

Design Guide For circular hollow section (CHS) joints under predominantly static loading

Cidect Verlag TUV Rheinland, 1991

[29]

J.A. Packer, J. Wardenier, Y. Kurobane, D. Dutta, N. Yeomans

Design Guide for rectangular hollow sections (RHS) joints under predominantly static loading

CIDECT

Köln, 1992, Verlag TUV Rheinland