STEEL Solver Documentation

General description

Steel is a 3D beam analysis program. It calculates all deformations, local forces, moments and stresses in 3D frames modelized by beams and submitted to static loads.
This program is based on the displacement method and includes axial and torsional rigidities and the center of shear can be different from the center of gravity.

This document details all calculations to help the users.

Table of Contents

1 Solver kernel

1.1 Units

Coordinates	mm
All dimensions of beam types	mm
All thicknesses of beam types	mm
Angle	°
Force	kN
Moment	kN.m
Linear load	kN/m
Pressure load	kN/m²
Body acceleration	m/s²
Imposed displacement	mm
Imposed rotation	mRad
Imposed spring stiffness deflection	kN/mm
Imposed spring stiffness rotation	kN/mRad
Areas	cm²
Inertias	cm⁴
Moduli	cm³
Stress	N/mm²
Young modulus	N/mm²
Yield stress (Re)	N/mm²
Density	kg/m³

1.2 Solver

The solver uses the displacement method. We resolve the following equation for each model tacking into account all beams, all applied loads and all defined boundary conditions:

$$F = K.U$$

With:

K : global rigidity matrix
U : global displacement matrix
F : global loading matrix.

To be able to solve this equation, the solver considers all beam fixed at both ends, being described by a rigidity matrix.

The general rigidity matrix for one beam is:

$$K_{beam} = \begin{bmatrix} \alpha & & & & & & & & & & & \\ 0 & 12b_z & & & & & & & & & & \\ 0 & 0 & 12b_y & & & & & & & & & \\ 0 & 0 & 0 & t & & & & & & & & \\ 0 & 0 & -6b_yL & 0 & (4+\Phi_z)b_yL^2 & & & & & & symmetric & \\ 0 & 6b_zL & 0 & 0 & 0 & (4+\Phi_y)b_zL^2 & & & & & & \\ -\alpha & 0 & 0 & 0 & 0 & 0 & \alpha & & & & & \\ 0 & -12b_z & 0 & 0 & 0 & -6b_zL & 0 & 12b_z & & & & \\ 0 & 0 & -12b_y & 0 & 6b_yL & 0 & 0 & 0 & 12b_y & & & \\ 0 & 0 & 0 & -t & 0 & 0 & 0 & 0 & 0 & t & & \\ 0 & 0 & -6b_yL & 0 & (2-\Phi_z)b_yL^2 & 0 & 0 & 0 & 6b_yL & 0 & (4+\Phi_z)b_yL^2 & \\ 0 & 6b_zL & 0 & 0 & 0 & (2-\Phi_y)b_zL^2 & 0 & -6b_zL & 0 & 0 & 0 & (4+\Phi_y)b_zL^2 \end{bmatrix}$$

With:

$$\alpha = \frac{EA_x}{L} ; b_z = \frac{EI_z}{(1+\Phi_y)L^3} ; b_y = \frac{EI_y}{(1+\Phi_z)L^3} ; t = \frac{GI_x}{L} ;$$

The bending stiffness parameters with the effect of shear b_z ; b_y ; Φ_y ; Φ_z (shear effect ON)

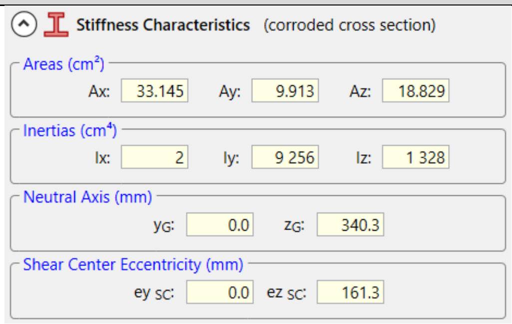
$$b_z = \frac{EI_z}{(1+\Phi_y)L^3} ; b_y = \frac{EI_y}{(1+\Phi_z)L^3}; \Phi_y = \frac{12EI_z}{GA_zL^2} ; \Phi_z = \frac{12EI_y}{GA_yL^2}$$

The bending stiffness parameters without shear effect (Shear effect OFF):

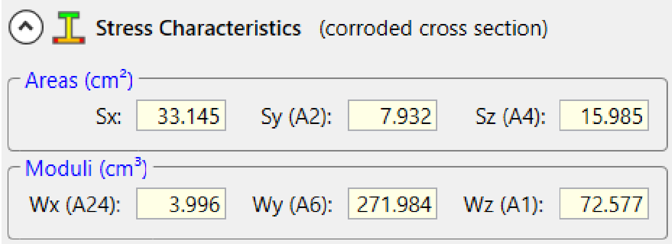
$$b_z = \frac{EI_z}{L^3} ; b_y = \frac{EI_y}{L^3} ; \Phi_y = 0 ; \Phi_z = 0$$

The total calculated deformation is the algebraic sum of the deformations generated by axial stresses (tension and bending) and the deformations generated by shear stresses which depend on the value of the shear modulus G.

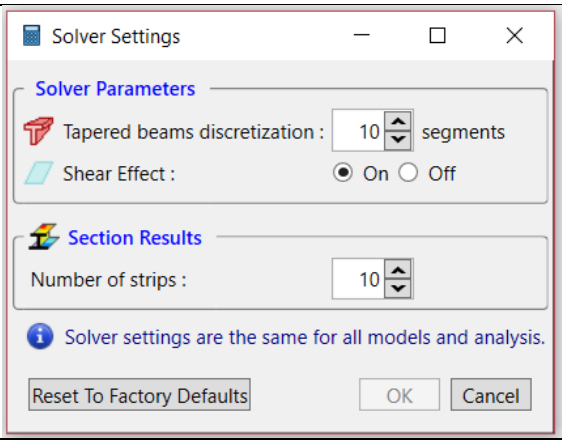
As a consequence, all stiffness characteristics have to be calculated to launch the calculation. The following data is displayed:

Stiffness characteristics	Interface: Entity info panel
Ax (cm^2): cross section area	
Ay (cm^2): shear area in y-direction	
Az (cm^2): shear area in z-direction	
Ix (cm^4): torsional inertia	
Iy (cm^4): moment of inertia about y-axis	
Iz (cm^4): moment of inertia about z-axis	
Position of the center of gravity G from the bottom (mm)	
Distance from shear center to G (mm)	

And to calculate normal and shear stresses, the following stress data is calculated and displayed:

Data for stress calculation (only the maximum value is displayed)	Interface: Entity info panel
Sx (cm²): shear stress area in x-direction	
Sy (cm²): shear stress area in y-direction	
Sz (cm²): shear stress area in z-direction	
Wx (cm³): minimum torsion modulus	
Wy (cm³): minimum strength modulus about y-axis	
Wz (cm³): minimum strength modulus about z-axis	

All solver parameters are adjustable in the “Solver settings window” in the menu Analyses:

	**Tapered Beam discretization:** - Minimum: 2 segments - Maximum 50 segments **Shear Effect:** ON/OFF **Number of strips:** define the number of section along all beams in all models. Default is 10 segments namely 11 sections results.

1.3 Beam characteristics

1.3.1 Material characteristics

Shear modulus (N/mm²) is calculated with the following formula:

$$G = \frac{E}{2(1+\nu)}$$

With:

ν : Poisson's coefficient

E : Young modulus in N/mm²

1.3.2 Stiffness calculation and stress calculation parameters

Ax and Sx: cross section area and stress area in x-direction. It is always the same value which is calculated with the cross-section area.

Ay and Sy: shear area in y-direction and shear stress area in y direction

Ay (cm²): shear area in y-direction	Sy (m.mm): shear stress area in y direction
$A_y = \dfrac{I_z^2}{\int \frac{M_z(y)^2}{e} ds}$	$S_y = \dfrac{I_z \,.\, e}{M_z(y)}$

With:

$M_z(y)$: The static moment of the part of the sectional area located below the current abscissa s with respect to z-axis.

e : The thickness corresponding to the current abscissa s.

$$M_z(y) = \int_0^{S_0} y(S).\,e(S)dS$$

Az and Sz:

Az (cm²): shear area in z-direction	Sz (m.mm): shear stress area in z-direction
$A_z = \dfrac{I_y^2}{\int \frac{M_y(z)^2}{e} ds}$	$S_z = \dfrac{I_y \,.\, e}{M_y(z)}$

With:

$M_y(z)$: The static moment of the part of the sectional area located below the current abscissa s with respect to y-axis.

e : The thickness corresponding to the current abscissa s.

1.3.3 Inertias and modulus

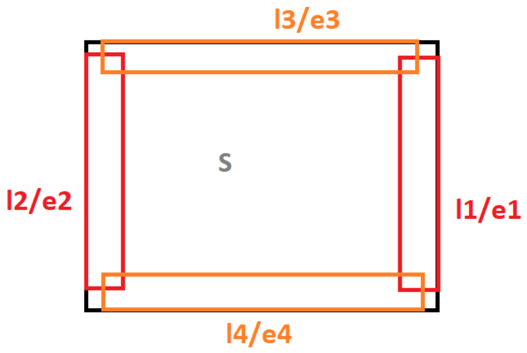
Moment of inertia about x axis (torsional inertia): Ix

Ix open section: F, Bulb, Angle, H and T	**Ix closed section:** Box, Tube	**Ix closed section:** **F + 2 plating continuous with equal width**
$\mathrm{I_x} = \sum_{i=1}^{i=n} \frac{l \, . e^3}{3}$	$\mathrm{I_x} = \frac{4. S^2}{\sum_{i=1}^{i=n} \frac{l}{e}}$	$I_{\mathrm{x}} = H^2 L \frac{e_1 . e_2}{e_1 + e_2}$
With: S : the internal area of the closed box section l : the length of each part e : thickness of each part		With: L : either width of attached plating or either with of top plating because both widths are equal. e_1 : the thickness of the top plating e_2 : the thickness of the attached plating H : web height + ½ e_1 + ½ e_2

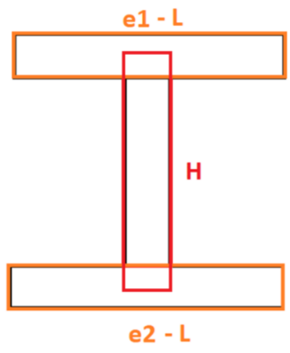
Note: size should be taken at mid-thickness.

Example for a box and an F + 2 plating shapes:

Box shape:



F + 2 plating shape:



Moment of inertia about y axis (Iy) and moment of inertia about z axis (Iz)

The stresses are obtained with:

$Sigma_y = M_y \frac{(d_z I_z - d_y I_{yz})}{I_{yz2}}$	$Sigma_z = M_z \frac{(d_z I_{yz} - d_y I_y)}{I_{yz2}}$

$I_{yz2} = I_y * I_z - {I_{yz}}^2$

The sigma unitary is :

$Sigma\_unitaire_y = \frac{(d_z I_z - d_y I_{yz})}{I_{yz2}}$	$Sigma\_unitaire_z = \frac{(d_z I_{yz} - d_y I_y)}{I_{yz2}}$

The unitary modulus is :

$W_y = \frac{1}{Sigma\_unitaire_y}$	$W_z = \frac{1}{Sigma\_unitaire_z}$

With d_z and d_y the coordinate of the calculation point from centre of gravity

1.3.4 Stresses calculation

For each calculation point, the stresses parameters are calculated:

Data for each stress calculation points
Sx (cm^2): stress area in x-direction
Sy (cm^2): shear stress area in y-direction
Sz (cm^2): shear stress area in z-direction
Wx (cm^3) : torsion modulus
Wy (cm^3) : strength modulus about y-axis
Wz (cm^3) : strength modulus about z-axis

Then, after the resolution, the program will calculate, for each beam, the following stresses at each calculation point all along the beams.

σ: **axial stress**	τ: **shear stress**
$\sigma_{Fx,i} = \frac{F_x}{S_x}$	$\tau_{Mx,i} = \frac{M_x}{w_{x,i}}$
$\sigma_{My,i} = \frac{M_y}{w_{y,i}}$	$\tau_{Fy,i} = \frac{F_y}{S_{y,i}}$
$\sigma_{Mz,i} = \frac{M_z}{w_{z,i}}$	$\tau_{Fz,i} = \frac{F_z}{S_{z,i}}$

With:

i : The ID of the stress point

F_x : Axial force (compressive or tensile)

F_y : Transverse force parallel to local y-axis

F_z : Transverse force parallel to local z-axis

M_x : Torsional moment

M_y : Bending moment about local y-axis

M_z : Bending moment about local z-axis.

Each part of beams are displayed in 2D-plane.

Example for H cross section:

Few points are detailed below:

Stress point A1:

$Sigma\ (A1) = \ \sigma_{Fx,A1} + \sigma_{My,A1} + \sigma_{Mz,A1}$

$Tau\ (A1) = \ \tau_{Mx,A1}(\tau_{Fy,A1} = 0\ and\ \tau_{Fz,A1} = 0)$

Stress point A2 (slave node):

$Sigma\ (A2) = \ \sigma_{Fx,A2} + \sigma_{My,A2}\ (\sigma_{Mz,A2} = 0)$

$Tau\ (A2) = \ \tau_{Mx,A2} + \tau_{Fy,A2}\ (\ \tau_{Fz,A2} = 0)$

Stress point A23+ (slave node):

$Sigma\ (A23 +) = \ \sigma_{Fx,A23+} + \sigma_{My,A23+} + \sigma_{Mz,A23+}$

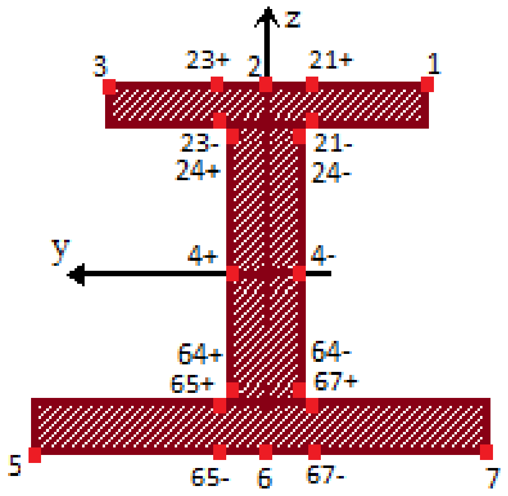
$Tau\ (A23 +) = \ \tau_{Mx,A23+} + \tau_{Fy,A23+} + \ \tau_{Fz,A23+}$

Stress point A24+ (slave node):

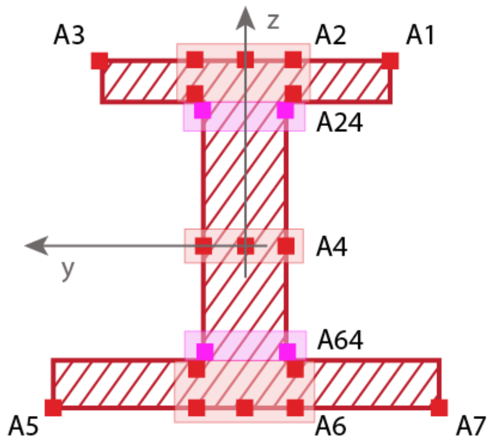
$Sigma\ (A24 +) = \ \sigma_{Fx,A24+} + \sigma_{My,A24+} + \sigma_{Mz,A24+}$

$Tau\ (A24 +) = \ \tau_{Mx,A24+} + \ \tau_{Fz,A24+}\ (\tau_{Fy,A24+} = 0)$

All calculation points :



Displayed stresses:



1.4 Beam properties specificity

1.4.1 Bulb standard profile

Bulb cross-section data are transformed into an equivalent angle profile.

The dimensions of the equivalent angle profile are calculated based on BV rule NR467, Pt B, Ch 4, Sec 3, 3.1.2.

The equivalent profile angle is obtained in mm from the following formula:

$$H_w = H'_w - \frac{H'_w}{9.2} + 2$$

$$t_w = t'_w$$

$$\mathrm{B} = \varphi[\, t'_w + \frac{H'_w}{6.7} - 2]$$

$$t_B = -\frac{H'_w}{9.2} - 2$$

With φ coefficient equal to:

$\varphi = 1.1 + \frac{(120 - H'_w)^2}{3000}$ if $H'_w \leq 120$

$\varphi = 1$ if $H'_w > 120$

With

H'_w : Height of the bulb section in mm

t'_w : thickness of the bulb section in mm

B : Breadth of the equivalent angle profile in mm

H_w : Height of the equivalent angle profile in mm

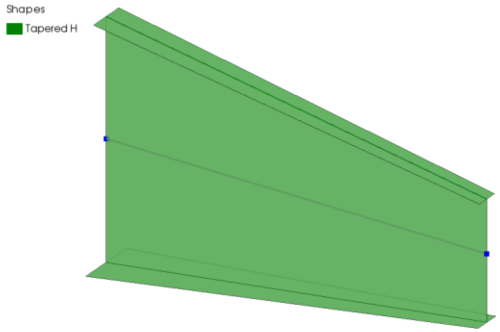
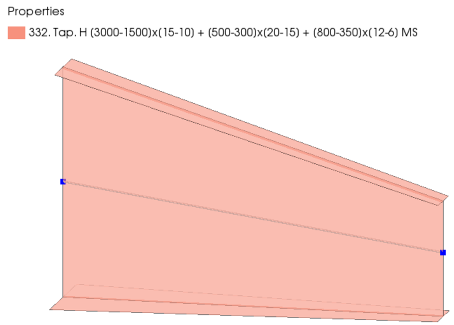
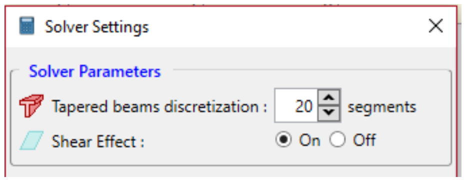
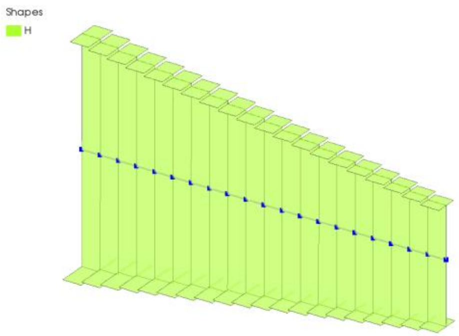
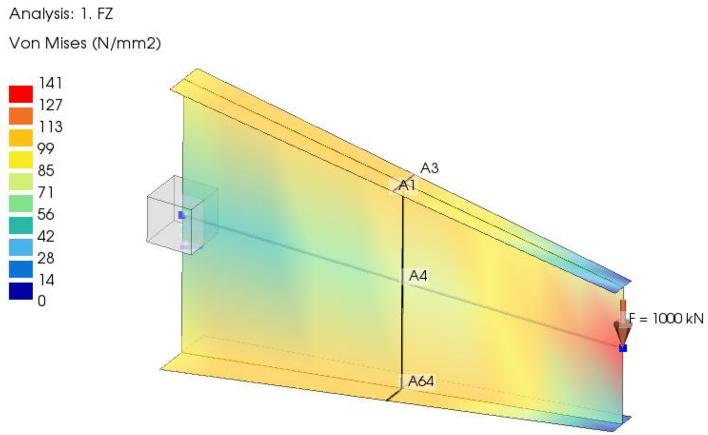
t_w : thickness of the equivalent angle profile in mm

1.4.2 Tapered calculation

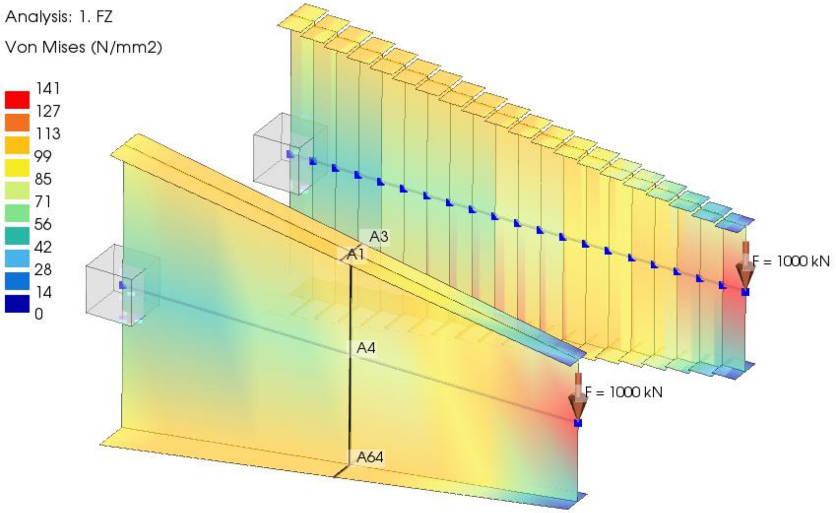
Tapered beam calculation is based on the discretization of the beam with constant beam sections in pre-processing. The constant beam section is calculated in middle of each segment.

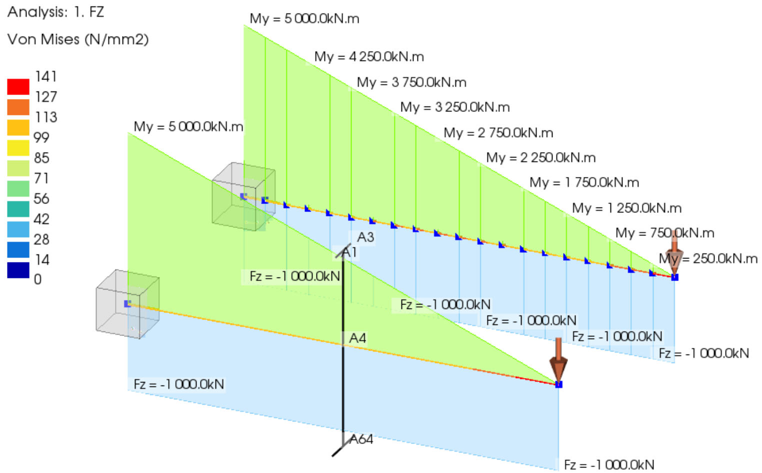
Then in post-processing, the stresses and shear are calculated with the exact size at each section along the beam.

Example:

Initial design	Solver calculation	Post-processing
Tapered beam is a property itself. The start and the end section dimension are defined in one beam property.   	Displacements and internal loads are calculated based on the equivalent constant beam section as shown in the example below:   Note that the number of constant segment is a user parameter.	Stresses and shear are calculated and displayed all along the beam at each section. The exact size at x location is used.  Note: Stiffness and stress characteristics at the current X location along the beam are displayed in entity info panel as well.

If we compare the analysis results of a beam defined by a unique tapered beam property and the same beam modeled by several strips with constant beam properties (varying along the beam), the results are equivalent:





Notes:

Holes are not allowed in tapered beam property. However, releases and rigids are allowed.

Overall stresses are evaluated in tapered beam at each section results considering the exact z location of the calculation point.

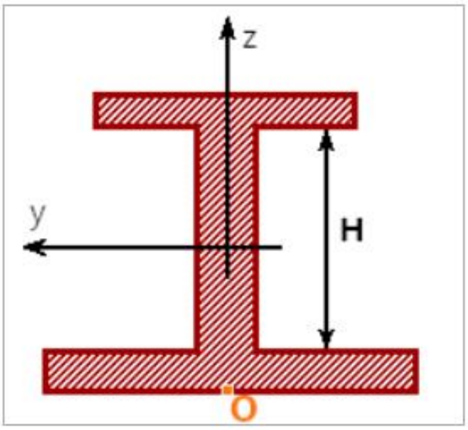
1.4.3 Assembly beam property

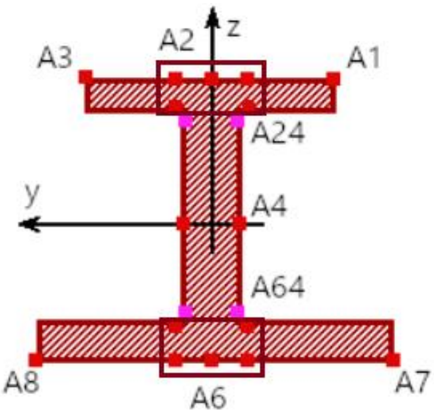
An assembly is a combination of a standard profile and one or two plating. The calculation between an assembly and the equivalent standard profile are exactly the same: same stiffness characteristics and same stress characteristics.

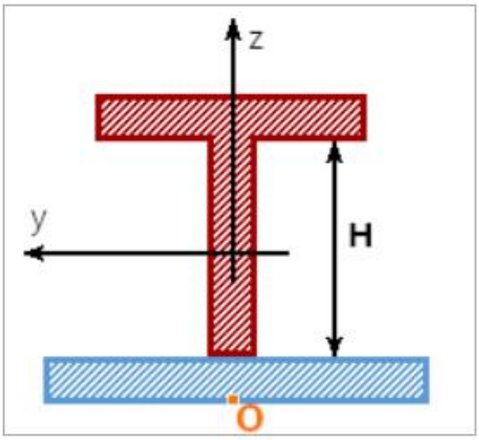
As the consequence, why and when do I have to use an assembly instead of a standard profile?

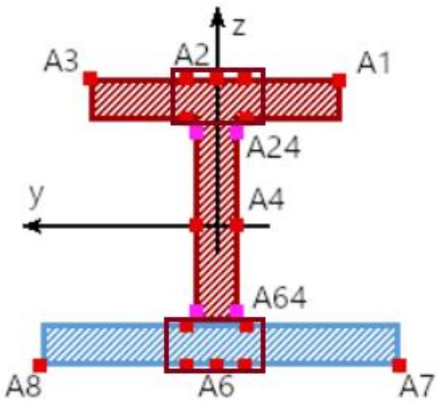
- to use continuous associated plating approach because only assembly beam property are available for selection.
- to use the color mode “properties grouped by plating “ to check your model. It is an easy way to check the double hull, the single hull and the pillars for ship structure for example.
- to use the automatic tool to define loaded width because it is applicable only for assembly beam properties and the tool is using the attached plating width.
- to check the correspondence between pressure loads and beam property. It is more relevant to apply pressure loads on a beam property with an attached plating instead of a standard profile.

Example with a H shape:

<table>
<tr><th>Standard profile</th><th>Assembly</th></tr>
<tr><td>




</td><td>




</td></tr>
</table>

1.4.4 User defined beam property

User defined beam property has been created to define beam properties which are not present in the library. Instead of entering a shape and dimensions, in case of a user defined beam property, the inputs are directly the stiffness characteristics parameters:

Areas (cm^2)		Inertias (cm^4)		Shear center eccentricity (mm)	
Ax	Cross section area	Ix	Torsional inertia	eySC	Distance from G to shear center along y-axis
Ay	Shear area in y-direction	Iy	Moment of inertia about y-axis	ezSC	Distance from G to shear center along z-axis
Az	Shear area in z-direction	Iz	Moment of inertia about z-axis		

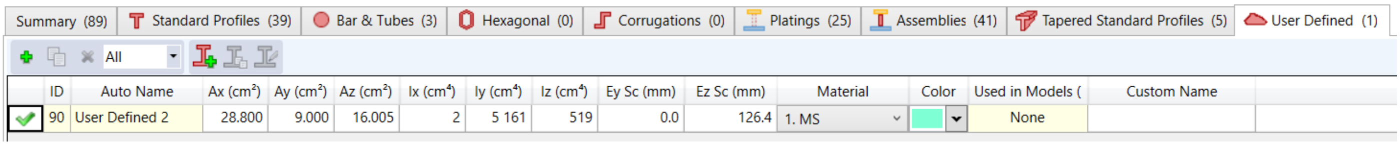
Notes:

No stresses are calculated on “user defined beam property”.

To launch an analyses, Ay and Az can be disregarded and in that case, the shear stress effect are neglected. Only a warning will be displayed.

Shear center eccentricity parameters is a vector from G to shear center. The sign has to be properly defined. All beam loads are applied at the shear center instead of G (center of gravity).

Example of user defined stiffness parameters:



	ID	Auto Name	Ax (cm^2)	Ay (cm^2)	Az (cm^2)	Ix (cm^4)	Iy (cm^4)	Iz (cm^4)	Ey Sc (mm)	Ez Sc (mm)	Material	Color	Used in Models (	Custom Name
	90	User Defined 2	28.800	9.000	16.005	2	5 161	519	0.0	126.4	1. MS		None	

1.5 Add-ons characteristics

1.5.1 Continuous members

Example for a double-wall section (F + 2 platings considered as a closed section):

If the cross section is an H with equal upper and lower flanges and the beam property is an assembly, then the program will consider that the beam belongs to a double wall structure such as a double-bottom, a cofferdam or a cellular hath-cover.

In this case, the torsional inertia I_x will be calculated according to the following classical formula:

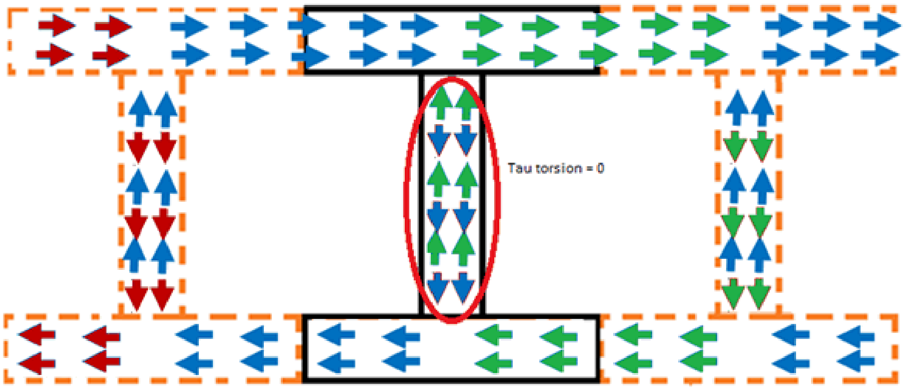
$$I_x = H^2 L \frac{e_1 . e_2}{e_1 + e_2}$$

With:

L : The width of bottom and top platings. For a double-wall, both widths are equal.
e_1 : The thickness of the top plating.
e_2 : The thickness of the bottom plating.
H : The height of the web.

And the stress flow along the web is modified: $\tau_{\text{torsion in web}} = 0$

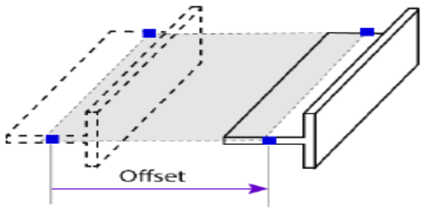
There is no shear due to torsion in the web because it is assumed that there are several beams connected to each other and the torsion flow is inverse in the web as described in the following picture.



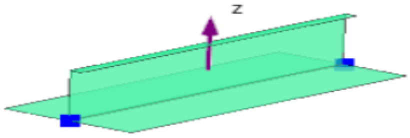
 Shear flow for each isolated beam.

1.5.2 Local z-offset

The solver uses a mathematical relationship to implement the offset: the nodes of the beam are translated along z local axis.



With **Auto**. option, the nodes are aligned with the origin point of the beam property. For an assembly, it is equivalent to align the bottom plating with the nodes:

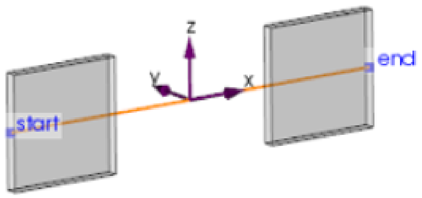


1.5.3 Rigid beams

1.5.3.1 Rigid Property

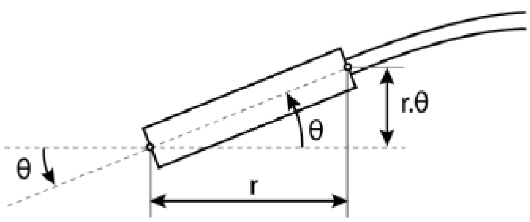
The solver creates a Rigid Property taking into account the greatest stiffness for each effect regarding the whole model.

1.5.3.2 Rigid Ends



The solver implements rigid ends using a **mathematical relationship** between the slope and deflection at the internal nodes and the slope at external nodes.

The displacement at the end of the rigid part is equal to $r.\theta$ where r is the length of the rigid part at start node and θ is the slope.



Note that the rigid end increases the stiffness concerning **rotation around y-axis and displacement along z-axis.**

1.5.4 Releases (text STEEL 3)

Normally, when several beams are connected to the same node, their origins or ends have to follow the translation or rotation displacements of this node.

An internal beam release allows to disconnect a beam origin or end from its adjacent node for what concerns one of the six translation/rotation displacements.

An internal beam release is not relative to a node but to a beam.

One beam can have several internal beam releases.

The internal beam releases are defined as follows by their freedom relative to the local axes.

1.5.5 Corrosion

There is no corrosion within the solver.

The beam properties shall be already corroded at the entrance in the solver.

1.5.6 Overall Stresses

There is no overall stress management within the solver.

Overall stress can be added later in the post-treatment but is not handled in the solver.

1.5.7 Loaded width

There is no loaded width within the solver.

Using the loaded width of the beams, pressure loads can be transformed into linear loads which will be processed by the solver.

1.5.8 Specific approach for corrugations: (count)

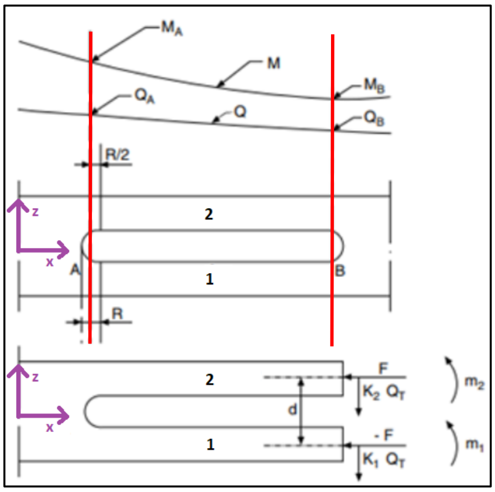
The solver will make a property equivalent to *count* times the defined corrugated property.

1.5.9 Hole calculation

The normal, shear and combined stresses are corrected around a quasi-rectangular manhole cut in the web plate of a beam submitted to bending and shear.

The cross-section of the beam must be either an H profile, either a T profile with an attached plating or either an F profile with two platings.

Shear and bending stresses are calculated all along the hole based on BV rules except for the following improvements:



1. the deflection due to shear stress is taken into account when calculating the stiffness of the lower and upper beams :

$K_{1-My/Fz} = \dfrac{\frac{I_{y1}}{\alpha_1}}{\frac{I_{y1}}{\alpha_1} + \frac{I_{y2}}{\alpha_2}}$	$\alpha_1 = 1 + \dfrac{12EI_{y1}}{GS_{z1}L^2}$	$\alpha_2 = 1 + \dfrac{12EI_{y2}}{GS_{z2}L^2}$

With:

L : Width of the hole

S_z : Shear area in the z-direction

Id = 1: The lower beam

Id = 2: The upper beam

2. Horizontal bending moment and transverse shear force are dispatched to the two reduced beams (ID = 1 and ID = 2) proportionally with their inertia.

$K_{1-Mz/Fy} = \dfrac{I_{z1}}{I_{z1} + I_{z2}}$	$K_{2-Mz/Fy} = \dfrac{I_{z2}}{I_{z1} + I_{z2}}$

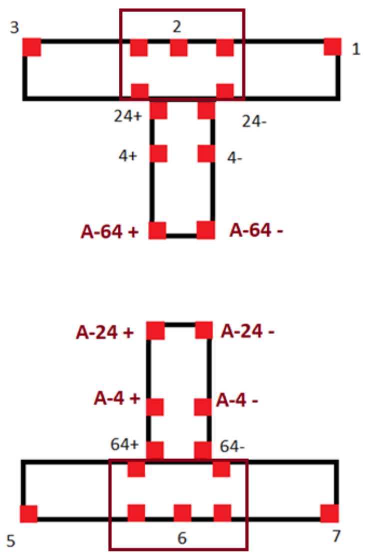
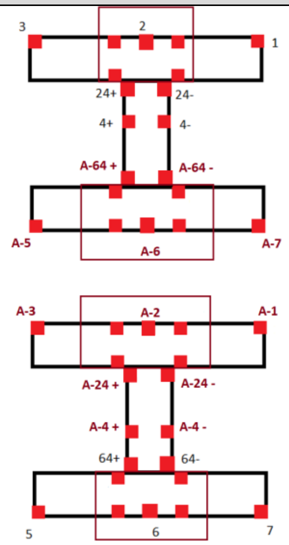
3. Compressive or tensile force is dispatched to the two reduced beams (ID = 1 and ID = 2) proportionally with their inertia.

$K_{1-Fx} = \dfrac{A_{x1}}{A_{x1} + A_{x2}}$	$K_{2-Fx} = \dfrac{A_{x2}}{A_{x1} + A_{x2}}$

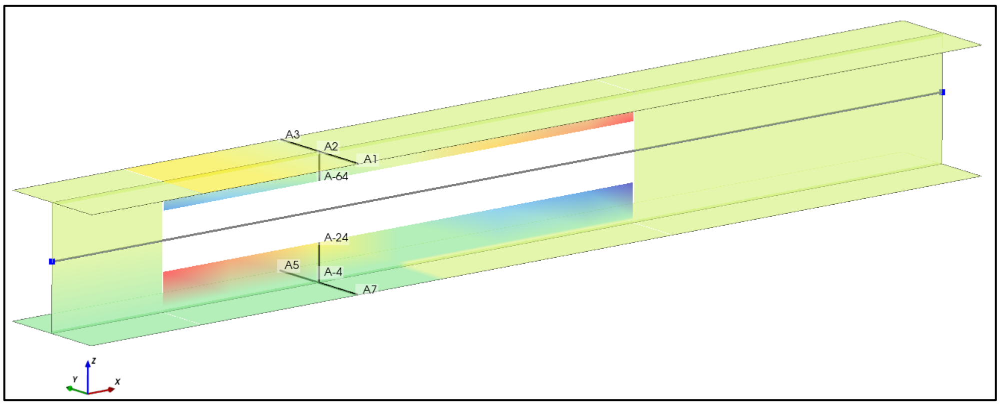
Compressive or tensile stress is algebraically added with compressive and tensile stress coming from the secondary bending moment.

4. Torsion is considered equal to zero in holes: $M_x = 0$ in upper and lower reduced beams along the hole.
5. The vertical shear force considered at each cross section along the hole is not the maximum of Ta and Tb calculated respectively at the left and at the right part of the hole, but the exact one corresponding to each section along the hole.

Calculation stress points along the hole: H cross section example:

Without flange	With flange
	

3D-View of beam cross-section:



Note:

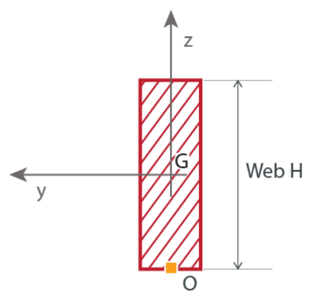
It is allowed to create and solve Hole calculation with Hole height higher than Hole length which do not follow the BV rule hypothesis of large openings. In that case, it is recommended to do additional calculation.

2 Appendix

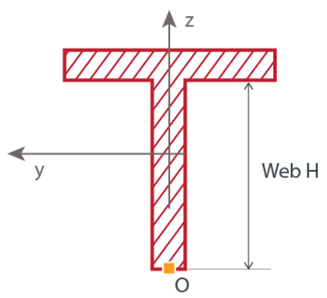
2.1 List of Beam properties

2.1.1 Standard profiles

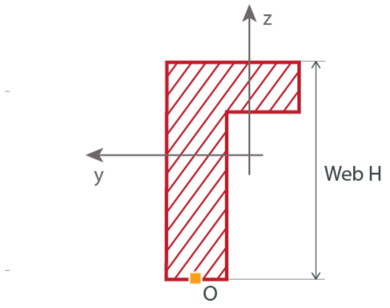
Flat bar (F)



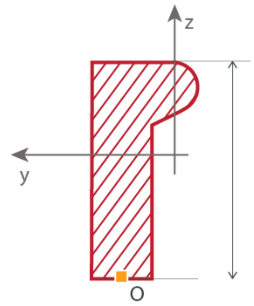
T cross-section (T)



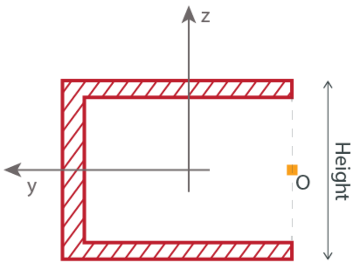
Angle cross-section (A)



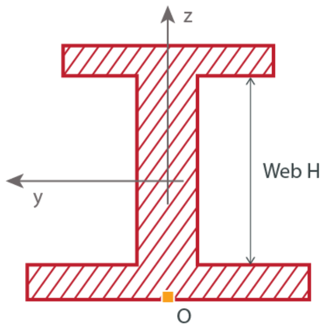
Bulb (B)



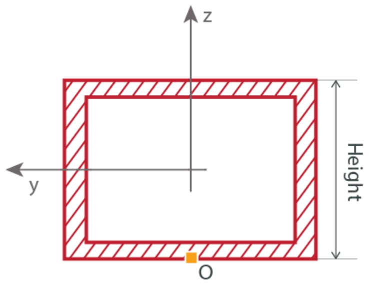
U cross-section



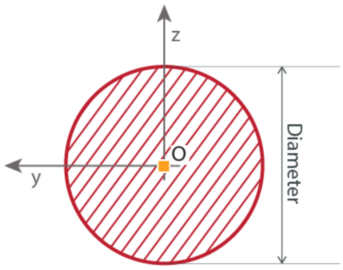
H cross-section



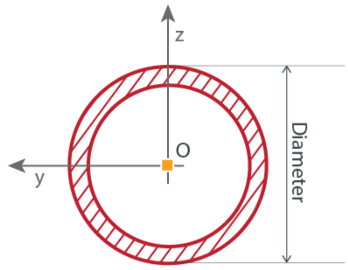
Box cross-section



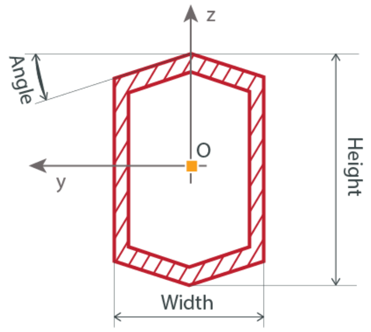
Bar cross-section



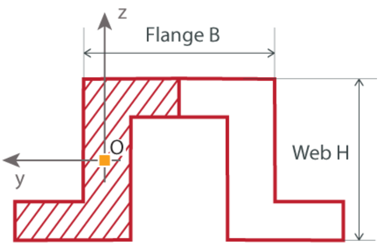
Tube cross-section



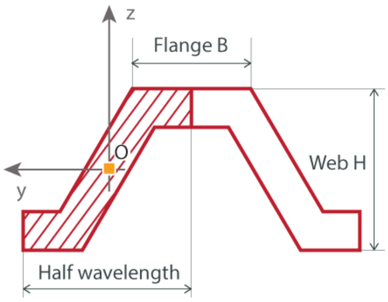
Hexagonal cross-section (Hexa)



U-shape corrugated cross-section (CorrU)

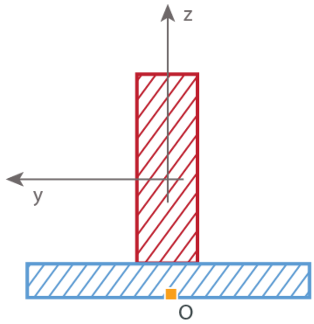


V-shape corrugated cross-section (CorrV)

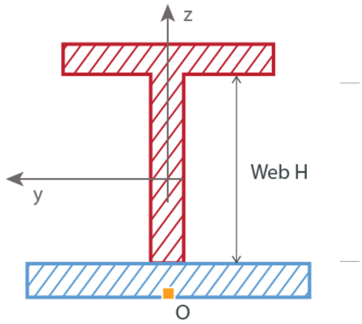


2.1.2 Assembly beam properties

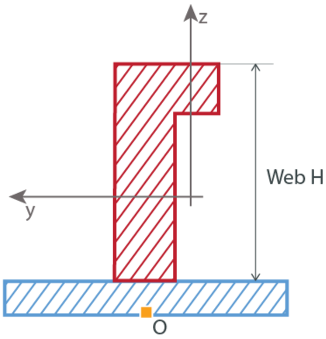
Flat bar + attached plating



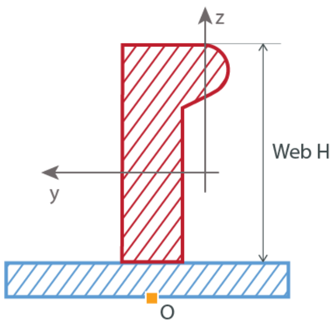
T + attached plating



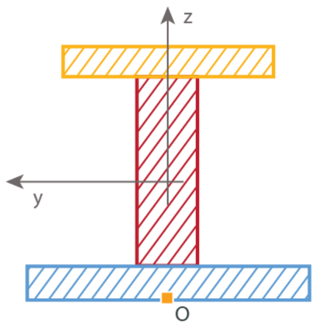
Angle + attached plating



Bulb + attached plating



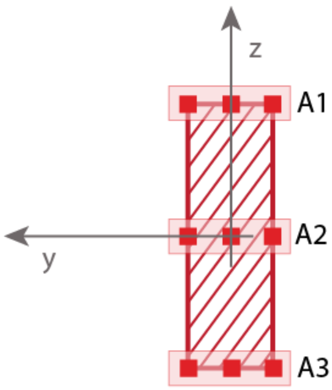
Flat bar + 2 platings



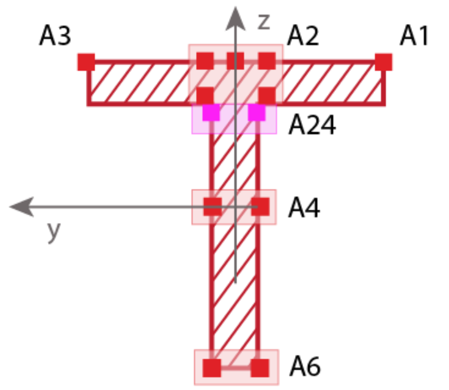
2.2 List of stress points for each beam property

2.2.1 Stress points for standard profiles

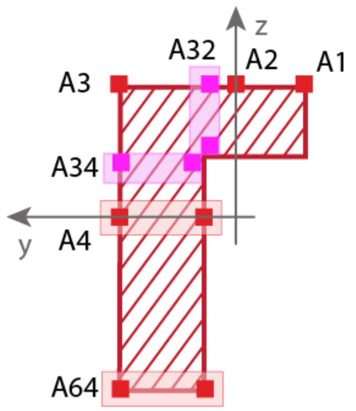
Flat cross-section (F):



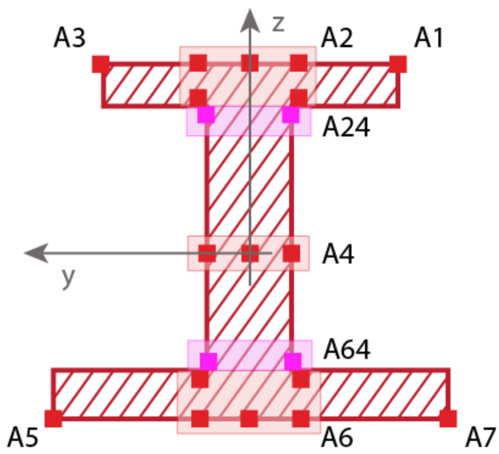
T cross -section (T):



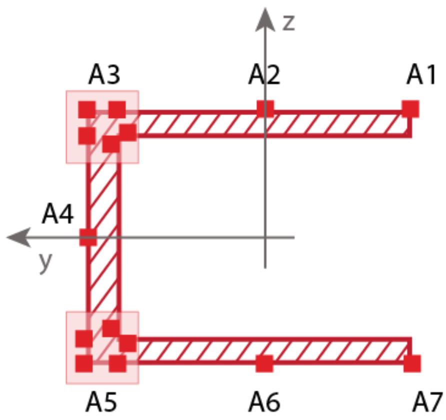
Angle (A) and Bulb (B) cross-sections:



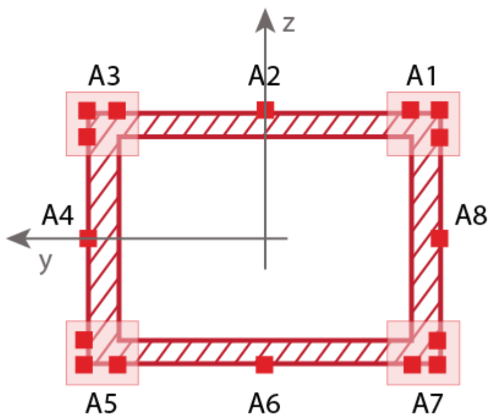
H cross-section (H) :



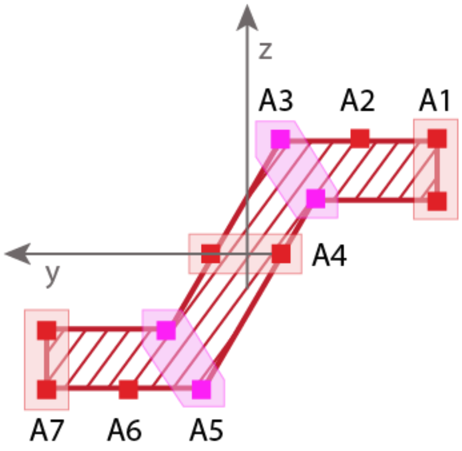
U cross-section (U) :



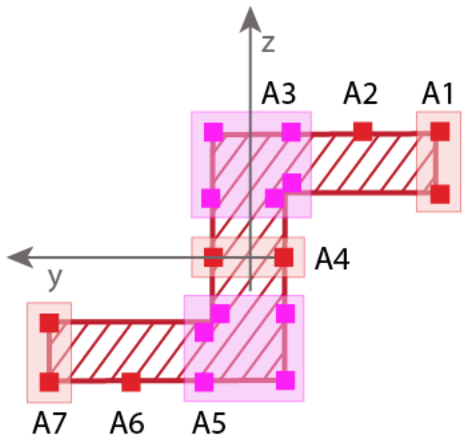
Box cross-section (Box) :



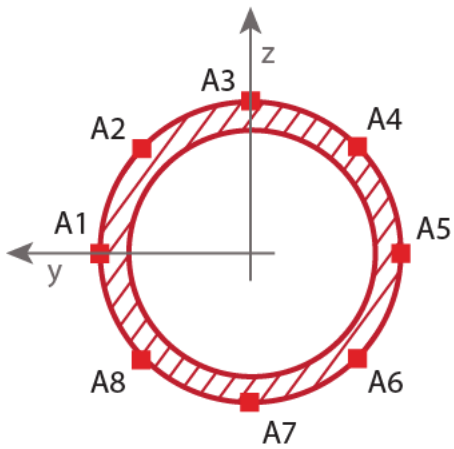
V-shape corrugated cross-section (CorrV):



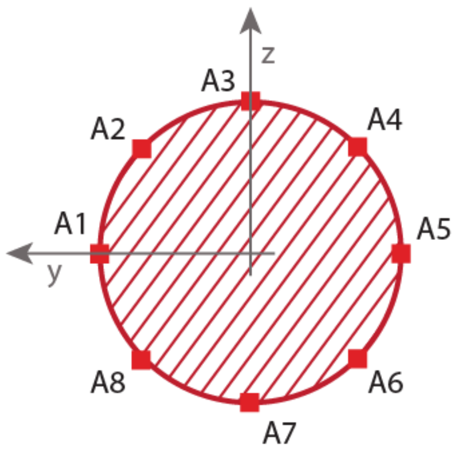
U-shape corrugated cross-section (CorrU):



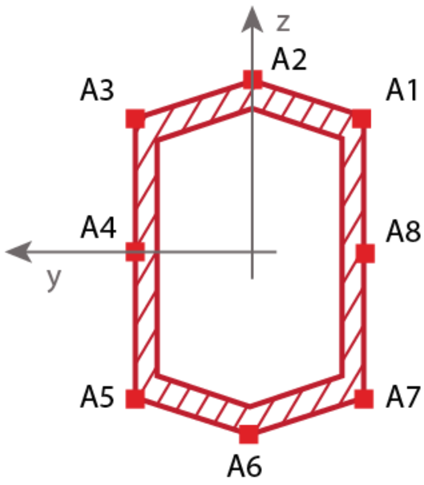
Tube cross-section:



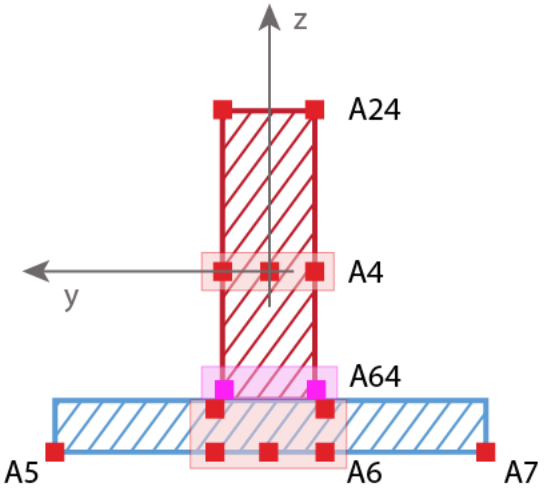
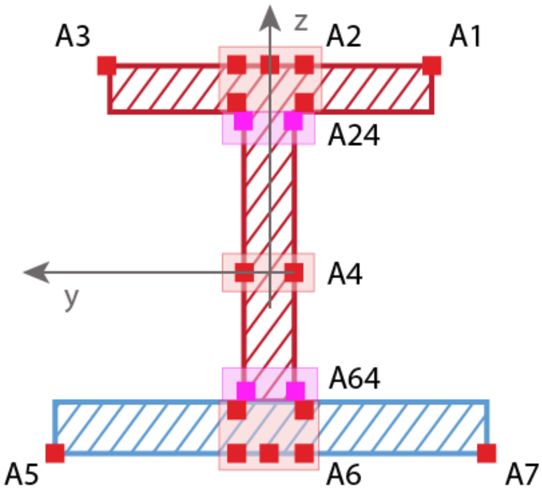
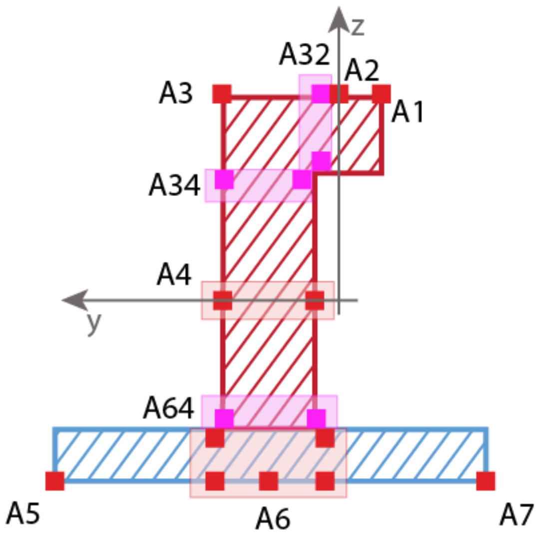
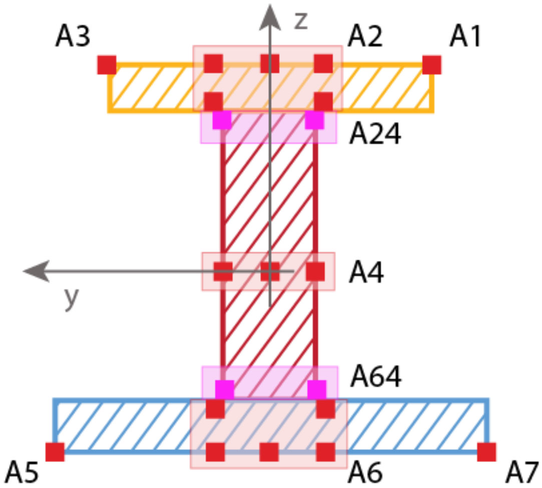
Bar cross-section:



Hexagonal cross-section (Hexa):

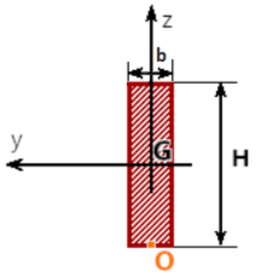


2.2.2 Stress points for each type of assembly

Flat + attached plating: 	T + attached plating : 
A or B + attached plating: 	Flat + 2 platings : 

2.3 Example: Flat calculation at A4 stress point

Flat : H (web height) and b (width=thickness)



Inertias:

$$I_x = \frac{H.b^3}{3}\,;\, I_y = \frac{b.H^3}{12}\,;\, I_z = \frac{H.b^3}{12}$$

Areas :

$$A_x = b.H$$

Distribution of static moment along y axis ($e = b = constant$):

$$M_y(z) = \int_0^{S_0} z(z)e(y)dydz = b.\int_{-\frac{H}{2}}^{z} z.dz$$

$$= -\frac{b}{2}\left(\frac{H^2}{4} - z^2\right)$$

$$\boxed{M_y(z) = -\frac{b}{2}\left(\frac{H^2}{4} - z^2\right)}$$

Static moment in A4 stress point (z=0):

$$M_y(z=0) = -\frac{b.H^2}{8}$$

Shear stress area in z-direction:

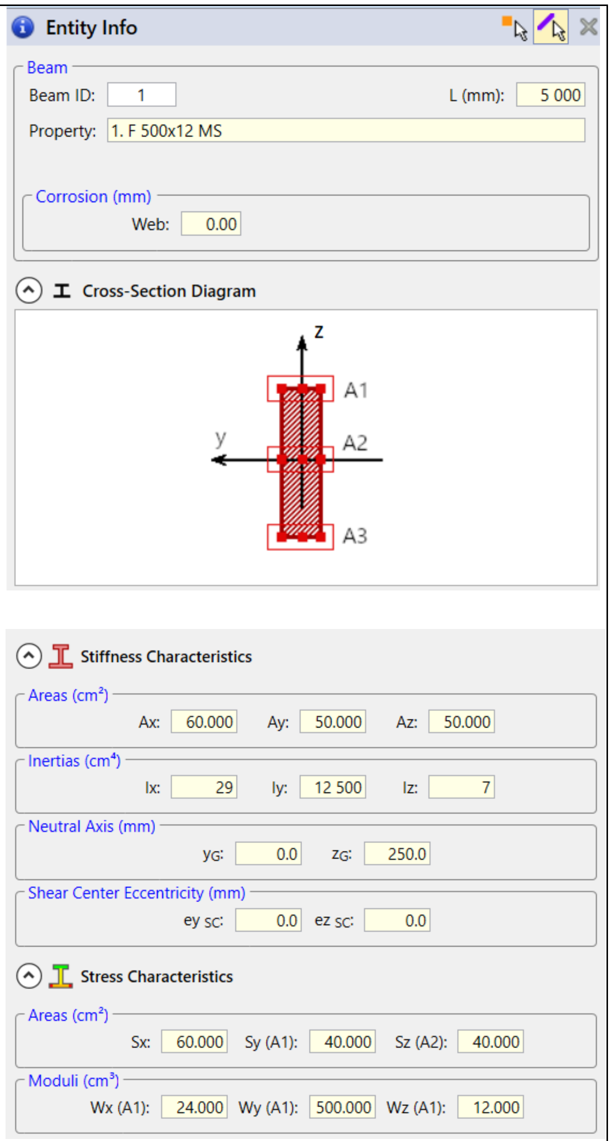
$$S_z(z) = \frac{I_y\,.b}{M_y(z)}$$

$$S_z(z=0) = \frac{\frac{b.H^3}{12}b}{\frac{b.H^2}{8}} = \frac{8}{12}(b.H) = \frac{2}{3}A_x$$

$$\boxed{S_z(z=0) = \frac{2}{3}A_x}$$

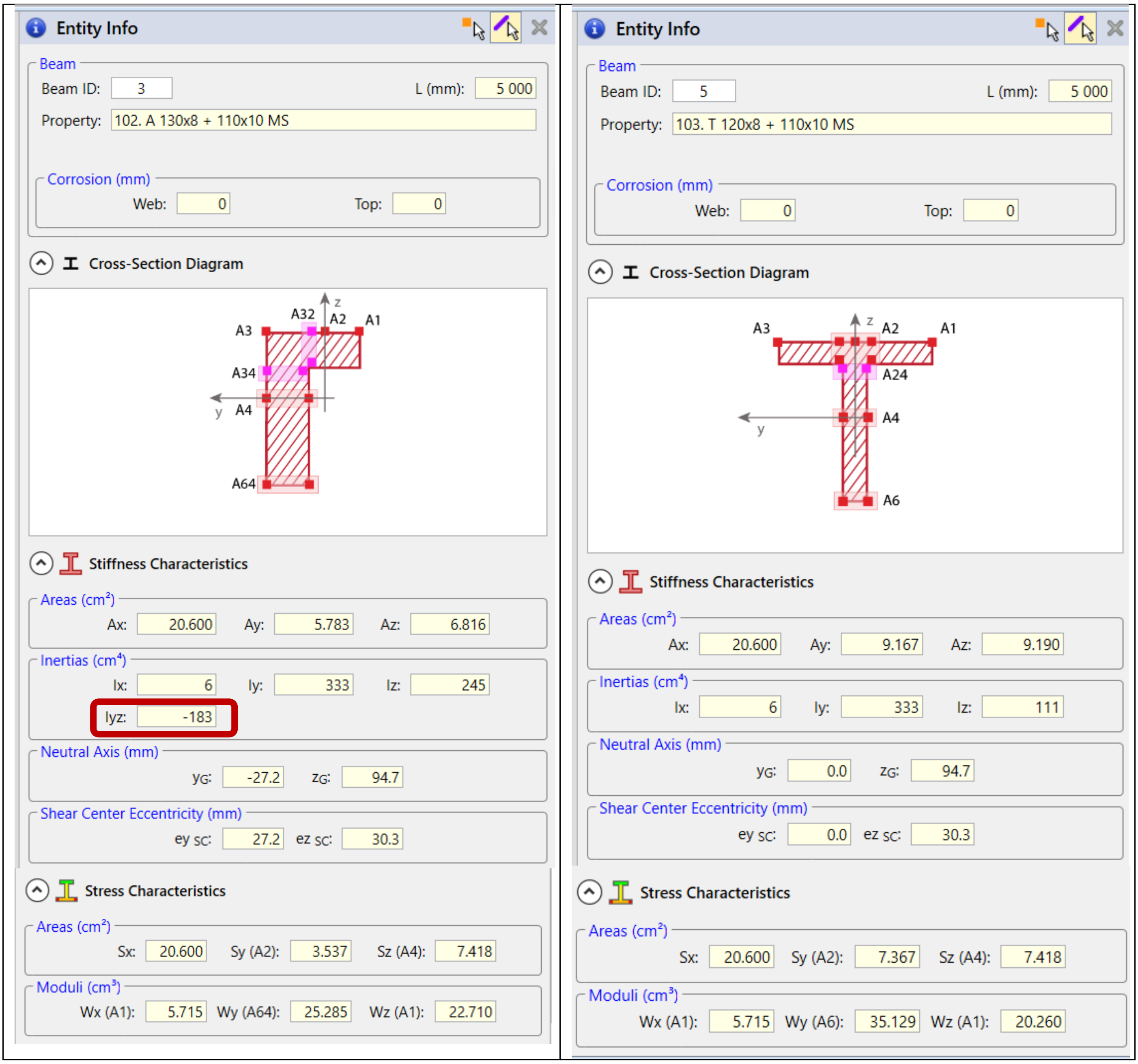
Shear area in z-direction:

$$A_z = \frac{I_y^2}{\int_{-\frac{H}{2}}^{\frac{H}{2}} \frac{M_y(z)^2}{b}\,dz}$$



$$\int_{-\frac{H}{2}}^{\frac{H}{2}} \frac{M_y(z)^2}{b}\, dz = \frac{1}{b}\int_{-\frac{H}{2}}^{\frac{H}{2}} \frac{b^2}{4}\left(\frac{H^4}{16} - \frac{1}{2}H^2z^2 + z^4\right)\, dz = \frac{b.H^5}{120}$$ $$A_z = \frac{I_y^2}{\int_{-\frac{H}{2}}^{\frac{H}{2}} \frac{M_y(z)^2}{b}\, dz} = \frac{\frac{b^2.H^6}{144}}{\frac{b.H^5}{120}} = \frac{120}{144}(b.H) = \frac{5}{6}A_x$$ $$\boxed{A_z = \frac{5}{6}A_x}$$	

2.4 Example: Difference between a T cross-section and an Angle Cross-section

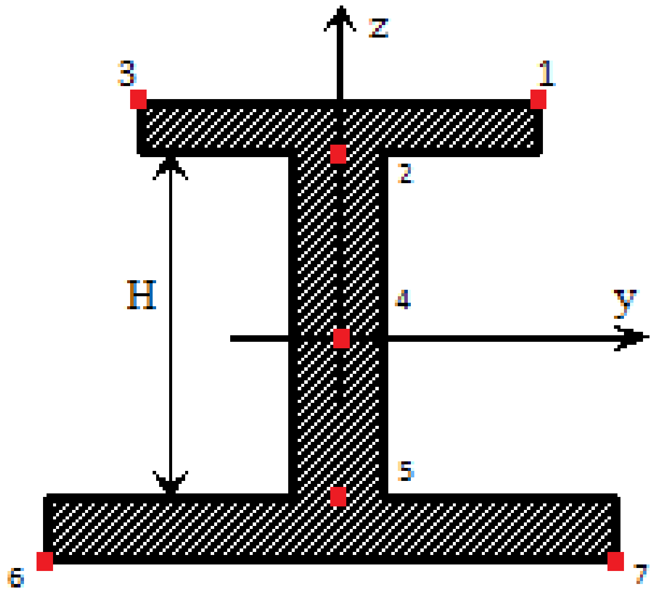
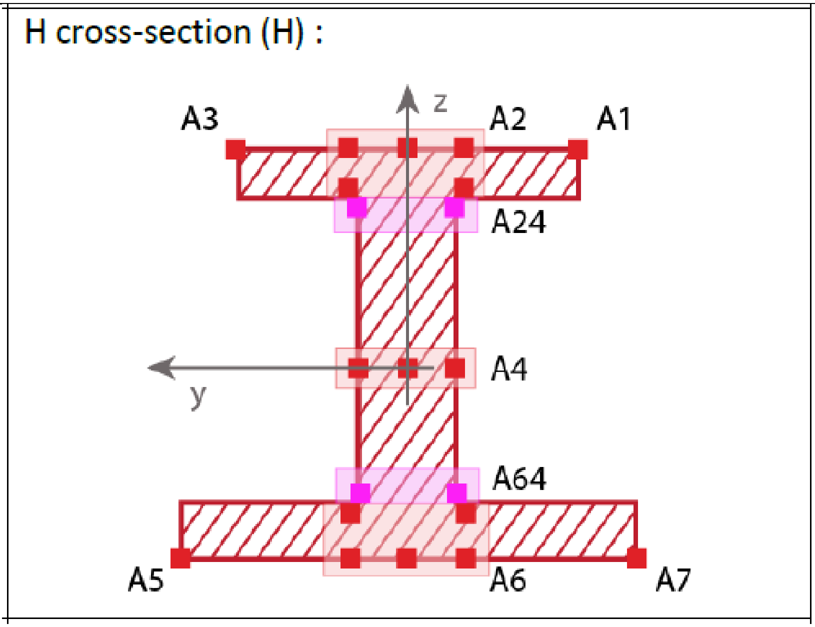


An angle in STEEL 4 is not anymore considered as a T cross-section for the calculation point of view even if it was the case in STEEL 3.

Now, in STEEL 4, we take into account the asymetry of the cross-section and as you can see, the cross inertia appears (red square in the left picture) for an angle.

If you want to get the same results in STEEL 4 as the one you get in STEEL 3, you should replace the Angle cross-section with the equivalent T cross section as shown above: T 120x8+110x10.

2.5 Example: Difference to calculate stresses for H cross-section between STEEL 3 and STEEL 4

STEEL 3	STEEL 4
 Sigma (point 2) = SigmaFx + SigmaMy (SigmaMz = 0 because point 2 is located in z axis) Tau (point2) = TauMx + TauFz (TauFy = 0 because point 2 is located in the web)	 Sigma (point 24) = SigmaFx + SigmaMy + SigmaMz (SigmaMz ≠ 0 because point 24 is not inline with z axis) Tau (point24) = TauMx + TauFz (TauFy = 0 because point 24 is located in the web) (point 24 : TauMx is calculated based on web thickness whereas in STEEL 3, in point 2, it was the maximum between TauMx (web) andTauMx (top flange).

To obtain the same results in STEEL4 than you obtained in STEEL 3, you should disregard internal loads Mz and Mx as shown below:

STEEL 3	STEEL 4