Note: A more user-friendly version of the guide is available here: Reinforcement Guide.

It is recommended to use the other version, as it opens much faster, offers a better table of contents, allows linking to specific sections, and comes with a built-in search. The content is identical.

General information

ProgressXML format

The PXML files are used to transfer both the production schedule from ERP (LOS and Plant Control) and the reinforcement data from Tekla to profit (reinforcement production planning software) and later to the machines (BlueMesh and Pluristar).

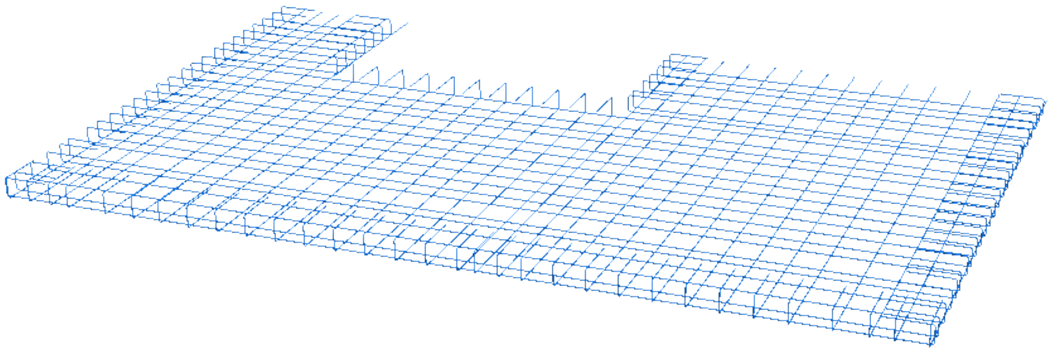
Complete specification of the PXML format can be found here: https://pxml.eu.

Steel types

A reinforcement "block" or a "steel" is a collection of reinforcing bars according to their type and the method of production. It may be a group of wires building mesh to produce in BlueMesh, or a group of loose rebars to produce in Pluristar or manually.

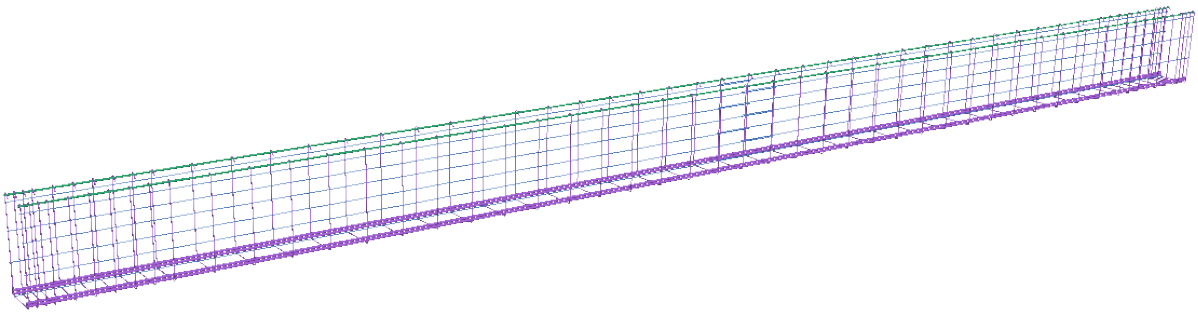
Mesh is a group of bars to be produced as a single mesh. Mesh can contain bent bars, but only such that can be made one by one, there can be no crossing bars in the bent part.

Meshes are grouped based on the reinforcement names specified in the export settings.



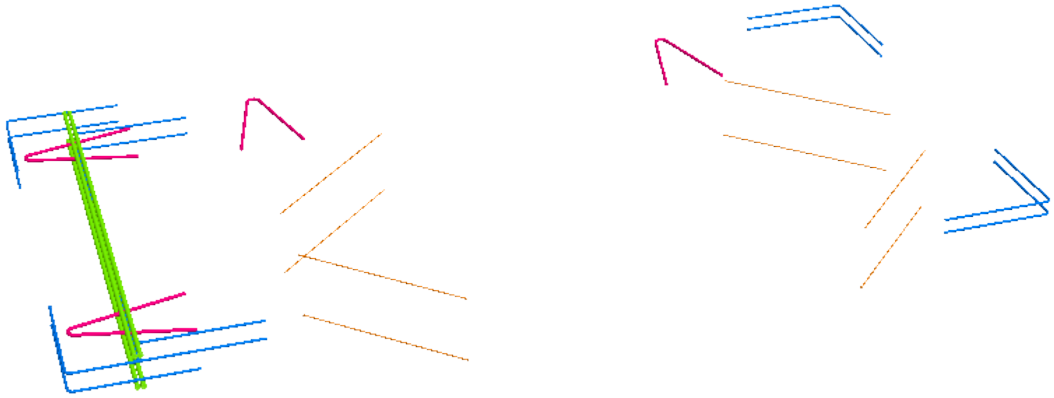
Cage is a group of reinforcement to be produced flat with the mesh welding machine and then bent into the final shape with the mesh bender. Cages are used mostly when the bendings cannot be made with the BlueMesh, for example because of crossing wires in the bent segments.

Cages are grouped based on the cage group number defined in the User-defined attributes dialog.



Loose reinforcement is a group of bars produced on Pluristar or manually. Such bars can be later manually added to a separately produced mesh or cage, or placed directly in the form.

Loose bars are grouped by reinforcement name but are always produced one by one.



Mesh types

The PXML specification defines the following mesh types. Our machine is not fitted with the beam bender and mesh type `1 = Bent mesh 2D` is automatically changed to `3 = Bent mesh 3D`.

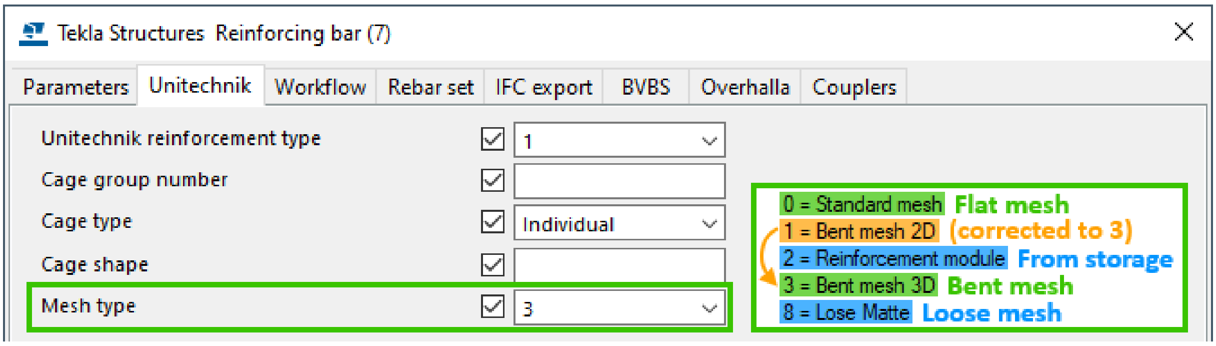
3.9.5 MeshType

For *Steel* blocks of the *mesh*/*cage* type, machine production of the entire unit is possible in principle.
A production process can then be specified via the **MeshTyp**:

- 0: Standard process. **Flat mesh or mesh to produce unrolled**
- 1: Mesh bending 2D: this mesh is to be fed to a beam bending machine. **Auto-corrected to type 3**
- 2: Manual or semi-automated production of a reinforcement module (e.g., by manual welding of automatically positioned stirrup series). **Item from storage**
- 3: Mesh bending 3D: this mesh is to be fed to a single-head bending machine. **Bent mesh**
- 4: cover mesh (such as the cover of a cage, or of a solid wall reinforcement).
- 5: cover mesh 2D; same as type '1', but will be supplied and delivered together with a type '4'.
- 6: cover mesh 3D; same as type '3', but will be supplied and delivered together with a type '4'.
- 7:
- 8: loose mesh: will not be produced by the steel machine, but will be added manually (usually, such a mesh is retrieved from the mesh warehouse). **Produce as loose**
- 9: application-specific type.

The mesh type `0 = Standard mesh` can be used to flatten a mesh modelled with bendings, for example in the crest of SDT. The mesh will be unrolled when it is sent to the machine.

The mesh type can be changed in the `Unitechnik` tab of the UDA dialog. Export may take this parameter from any rebar and thus property should be set for all rebars grouped in a mesh.



Reinforcement types

The PXML specification defines the following reinforcement types. For meshes and cages that are producible on our machine only the marked types are applicable. For the loose reinforcement the type does not matter and is ignored by the machine.

3.10.2 ReinforcementType (reinforcement layers)

The types of reinforcement largely follow the UNICAM definition; however, additional definitions were introduced for cage production:

- 0 = no definition.
- 1 = first rebar layer; **for cage production: stirrups**. Transversal wires*
- 2 = second rebar layer; **for cage production: longitudinal rebar**. Longitudinal wires*
- 3 = spears.
- 4 = other reinforcement (third rebar layer).
- 5 = upper reinforcement of first rebar layer.
- 6 = upper reinforcement of second rebar layer.
- 7 = upper reinforcement other rebars (upper reinforcement of third rebar layer).
- 8 = loose rebars (not welded, not connected to any other rebars). Bars added manually

*Type 2 bars follow bends of type 1 bars when unrolling

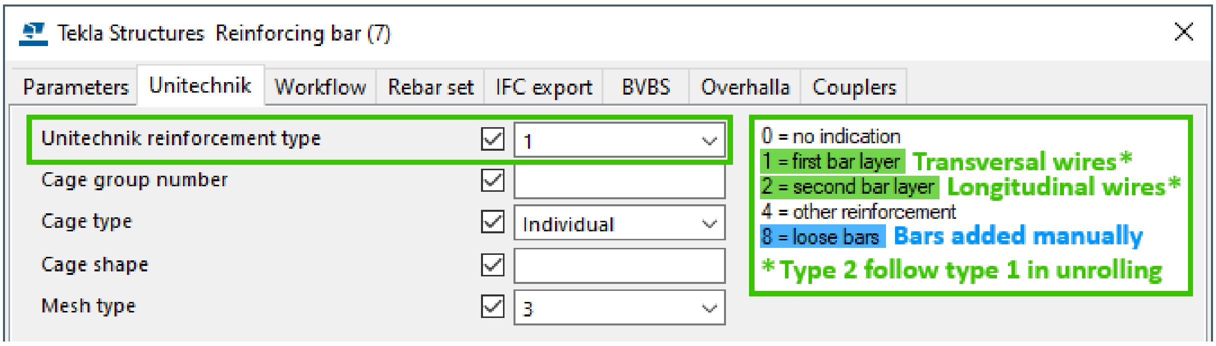
For Extirons, the ReinforcementType is equivalent to UNICAM Extiron-Type.

During production, the longitudinal wires are always placed above, and the transversal wires are welded from below. Thus, transversal wires are normally of type `1 = first bar layer`, and longitudinal wires of type `2 = second bar layer`.

Important feature of type `2` bars is that they follow bendings of type `1` bars. BlueMesh cannot produce bendings with perpendicular bars, but this can be used to correctly unroll cages (or meshes), that are produced flat. This will take precedence over the longitudinal/transversal rule.

The type `8 = loose rebars` can be used to produce some rebars as loose, while still grouping them with a mesh or cage into which they should be added.

The reinforcement type can be changed in the `Unitechnik` tab of the UDA dialog.



When the reinforcement type is left blank, Tekla will try to guess the type from the rebar placement. For many bars it chooses the type `4 = other reinforcement`, that is incorrect for our meshes.

It is recommended to always assign the reinforcement types manually for meshes and cages, and check if the correct types are set for all bars in AviCAD. The types are used when unrolling and to determine the correct order of bendings in the machine software.

Reinforcement flows

The table below presents overview of various "flows" the reinforcement can follow from the design PXML to the result of production, depending on its properties and used steel, mesh, and bar types.

PXML			
Case	SteelType	MeshType	BarType
Flat mesh Bent mesh to produce flat	mesh	0 = Standard mesh	0 – 7
			8 = Loose bars
Bent mesh without cross bars in bent segments		1 = Bent mesh 2D 3 = Bent mesh 3D	0 – 7
			8 = Loose bars
Any mesh from storage, for example preproduced		2 = Reinf. module	(any)
Mesh to be welded manually		8 = Loose mesh	(any)
Bent mesh with cross bars in bent segments, or not directly producible	cage	(none)	0 – 7
			8 = Loose bars
Loose reinforcement	loose	(none)	(any)

Profit		
Import	Machine	Export
RST_25 on all meshes	Flat	Unrolled
	Bend	(unchanged)
Adjust bending device Snap or shift corner bars RST_25 if no corner bars	Cage	Unrolled
(nothing)	Warehouse	Do not export
Merge 2x90 bends to 180 Pluristar ø ≤ 16 & L ≤ 12m, otherwise Manual	Pluristar	(unchanged)
	Manual	(unchanged)
(when unspecified)	Unknown	Do not export Error in PTS

Production	
Machine	Result
BlueMesh	Flat mesh produced
	Bent mesh produced
	Flat mesh produced with angle variation
Pluristar	Loose bars produced
Manual	Produced manually

Design with focus on production

The introduction of mesh welding machine shifts responsibilities from the factory to the office. Previously, the reinforcement was modelled mostly to be presented on drawings, allowing simpler design, and leaving room for factory workers to handle the details. With the mesh welding machine, the complete reinforcement must be modelled with understanding and precision, because there is only a minimal room for intervention in the factory.

The designer must know the capabilities and limitations of the machines and manual production methods to plan an effective production process. The first step of modelling must always be planning of the production. The reinforcement should be divided into meshes produced as single units and loose rebars that will be supplemented manually. A crucial part is thinking through the assembly steps to ensure that each element can be reinforced correctly and effectively.

The designer should always focus on minimizing the overall cost of production, both in terms of the materials and required work. This will often mean that the mesh welding machine should be used to the biggest possible degree. The manual production may sometimes still be beneficial or necessary, particularly for small or complicated parts, not suitable for machine production, or requiring much overhead in design and factory work for little gain.

Reinforcement diameters and spacings should be optimized in both directions according to the statical or minimum requirements. The mesh does not need to be "square" – the wire spacings and diameters can differ in both directions and change along the element. Rebars should be adjusted to ensure that all parts of reinforcement fit together and embeds can be placed correctly.

Naming reinforcement

The name from Tekla is used for grouping reinforcement in the exported PXML and is printed out in the mesh labels. The name should describe well the purpose and placement of the reinforcement to help the factory workers place all parts in the right position and order.

The guidelines for specific types of reinforcement:

- **Meshes:** All names should start with `Nett`, as this is used to recognize meshes in drawings. When a big mesh is split into parts, describe the positions, or order in which they should be put together. Examples: `Nett bunn`, `Nett topp`, `Nett UK`, `Nett OK`, `Nett yttersjikt`, `Nett innersjikt`, `Nett YS`, `Nett IS`, `Nett plate midt`, `Nett ribbe høyre 2`.
- **Cages:** No prefix is required, as cages are grouped based on their numbering, but the names should describe where to position the cage in the form. Examples: `Ribbe topp`, `Konsoll`, `Ribbe bunn`, `Ribbe venstre`, `Ribbe høyre opp`, `Hovedarmering`.
- **Loose bars:** The bars to be added to a mesh or cage should be in principle named just like it. Otherwise naming should be by the reinforcement function. Examples: `Ribbe topp`, `Løft`, `Konsoll`, `Randarmering`, `Utsparing`, `Hylse`, `Endebøyler`, `Ribbebøyler`, `Vinkler`.

Production footage

A good understanding of the production helps to design the reinforcement correctly and to solve the PTS problems. The recordings below present various parts of the process.

Production of flat mesh in BlueMesh – first stage of production, before bending.

Bending of bent mesh in BlueMesh – production speed was reduced when filming.

Bending of cage with MEP – the old mesh bender, shows issues with varied spacings.

Bending of cage with Hambi – bending of cages with corner bars is also possible.

Course materials

During autumn 2022 a course was held to prepare designers to model reinforcement for mesh welding machine. This was before we had hands-on experience with the machine and much of the information is now outdated – this guide should always be the first source of knowledge.

Nevertheless, the general workflow and modelling techniques presented during the course can still be useful, particularly as an introduction: Design for mesh welding machine course.

Best practice in design

The best practice in design for the mesh welding machine change continuously, as we discover more optimal methods for production and figure out the details. The general principle is to use the machines effectively, limit the manual work and reduce the amount of reinforcement – by smart calculations and trimming bars in the areas where they are not necessary.

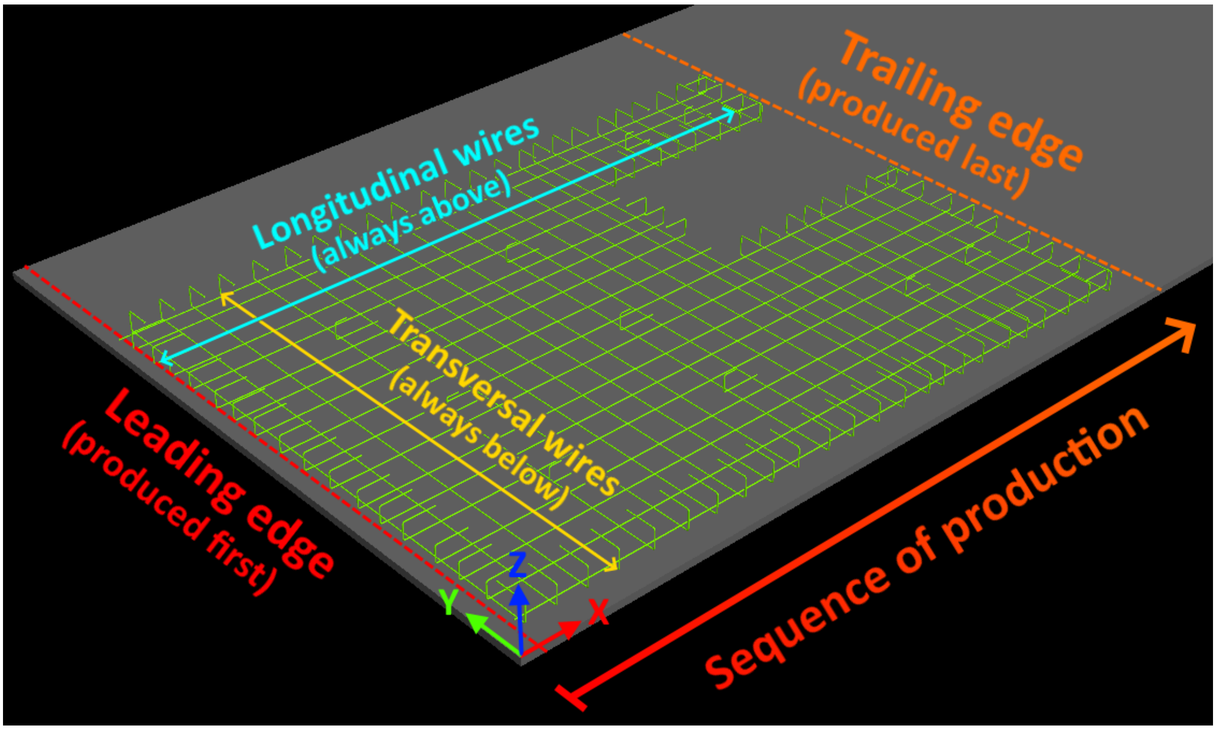
The guidelines are collected in a separate document: Nettsveisemaskin - BEST PRACTICE.

Mesh welding machine

Direction of production

The production direction for meshes and cages is now mostly determined by profit and the machine software, but the orientation in AviCAD is used as a basis and may influence the results. Orientation of loose reinforcement in AviCAD does not affect the production in any way.

The orientation of meshes in production can be viewed in the PTS received data. The picture below shows how the orientation in the program translates to the actual production direction.

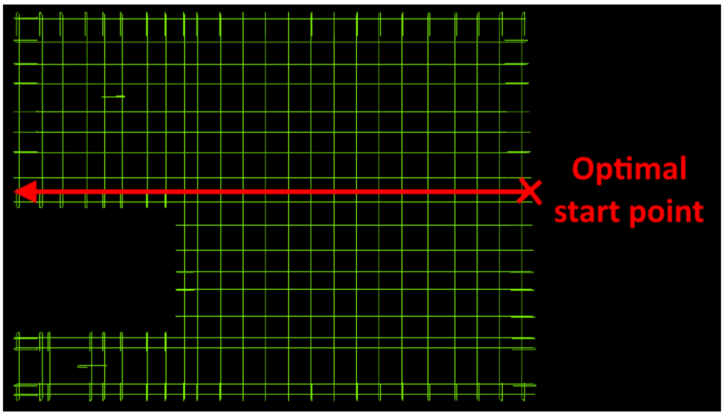


The production direction for a mesh can be forced by rotating in the production view. The element view will not be affected and show the reinforcement as it was exported.

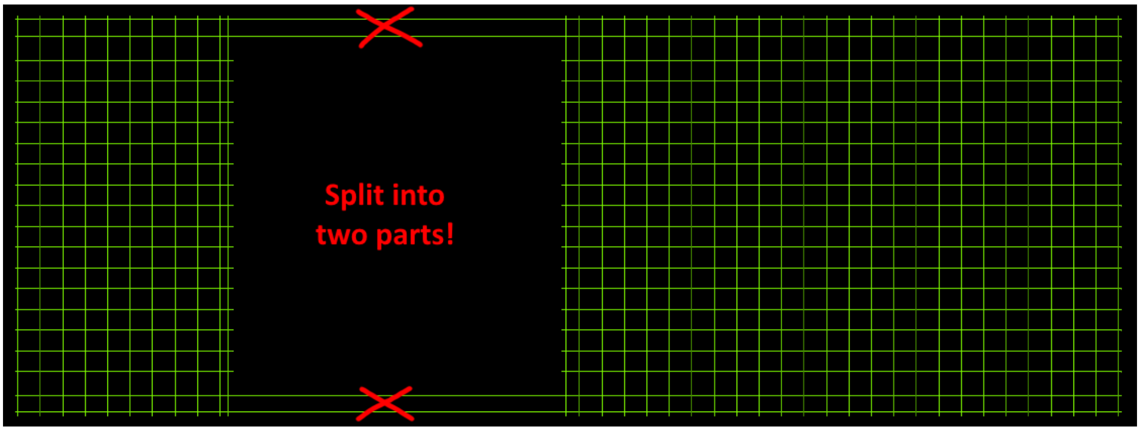
The export settings in Tekla can be adjusted to guess the optimal orientation for each mesh.

Mesh stability and optimal direction

The production is the most stable when open meshes are oriented such on pallet, that the opening is in the trailing end (produced last). Starting with opening may cause issues with welding and bending.



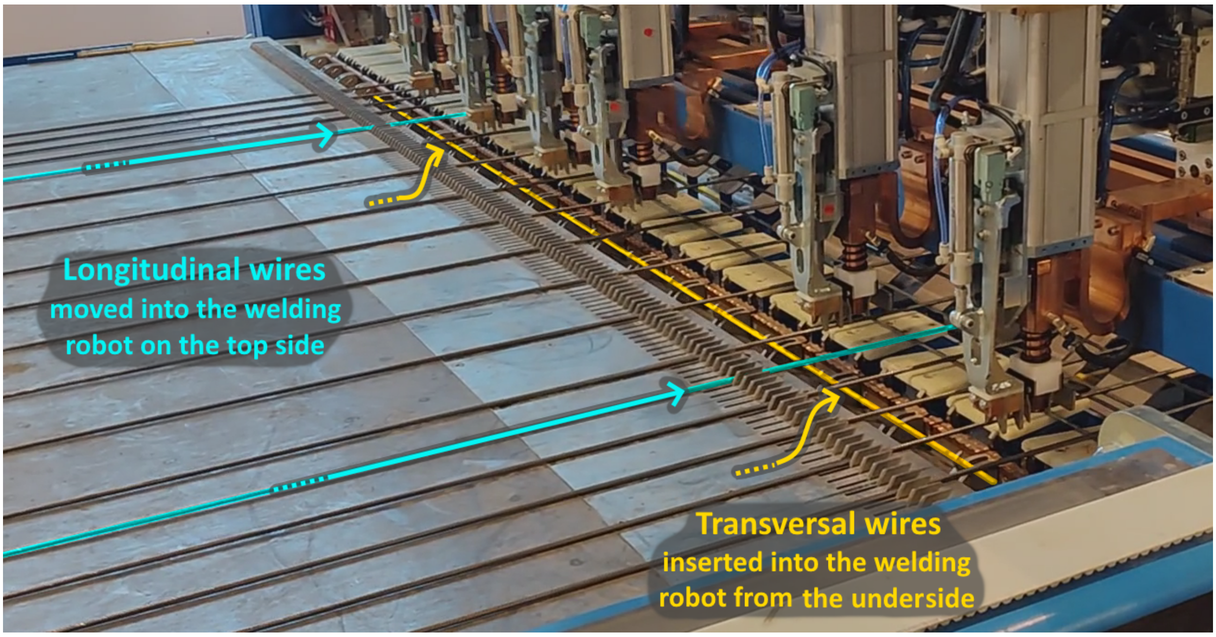
Meshes with big openings are better produced split or may cause issues in production and transport.



Order of reinforcement layers

The order of reinforcing layers is fixed. The longitudinal wires are always moved into the welding robot on the top of the pallet, while the transversal wires are always attached from below.

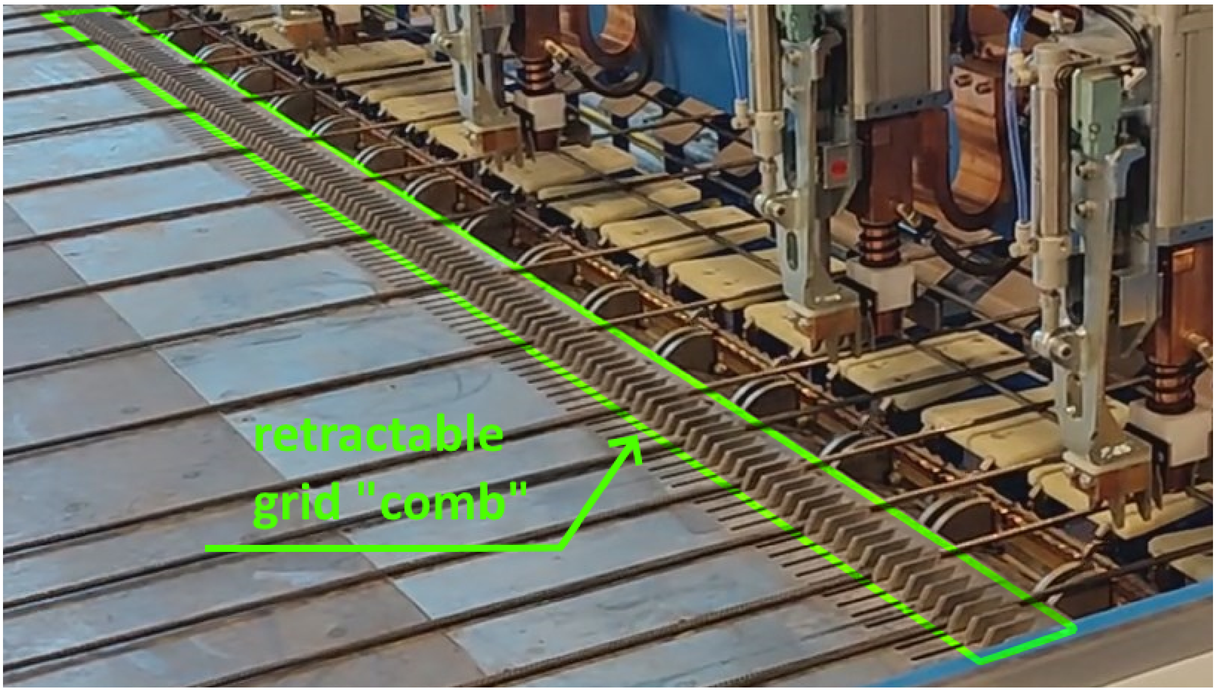
When a mesh is modelled with the transversal wires above, the machine software will turn the mesh upside down to be able to produce it. This often leads to problem with negative bending in PTS, as it is not possible to turn the mesh back once it is welded.



Rastered and gridless production

The production with grid (rastered) is much more stable. It uses a retractable grid comb that guides the longitudinal bars into the welding robot. Production with grid requires all spacings between the longitudinal bars to be a multiple of 25 mm (25, 50, 75, …).

For gridless production, the distancing device is retracted. In this mode spacings between the longitudinal wires can be varied by 1 mm (50, 51, 52, …). Almost all meshes can in principle be produced well without a grid, however bars might sometimes get crooked causing production problems, or resulting in the finished mesh being wavy.



Two additional requirements must be fulfilled for the gridless production:

1. The spacing of all longitudinal wires must be at least 50 mm.
2. The spacing of all transversal wires must be at most 450 mm.

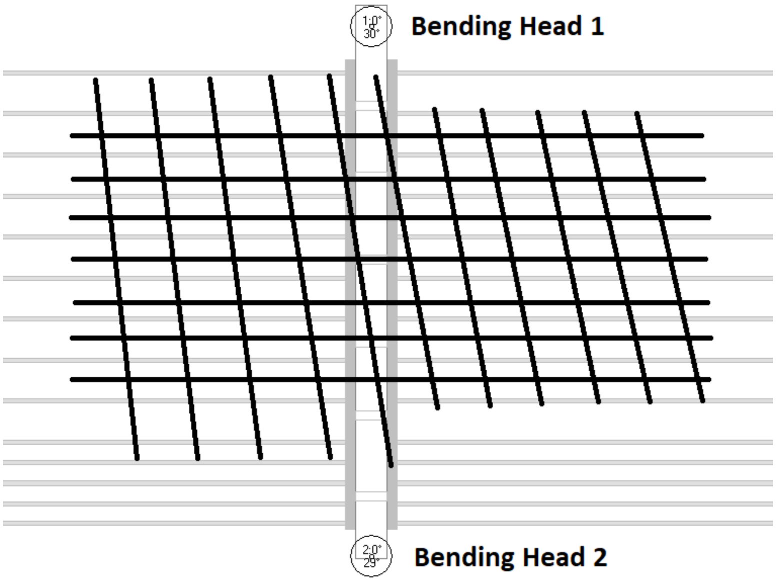
When at least one of these conditions is not fulfilled, the rastered production is forced and all longitudinal bars are automatically shifted into the grid.

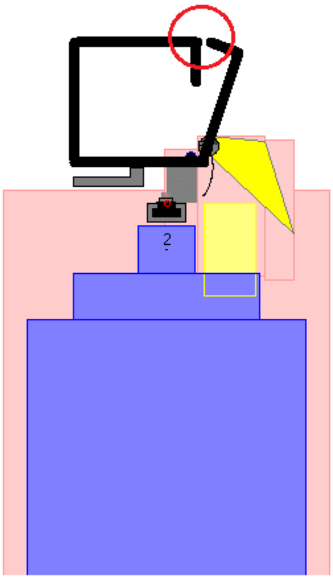
Currently changing between the rastered and gridless modes is fully automated. Because of many problems we have previously experienced in production because of unsatisfactory straightening of wire from coil, the rastered production is now used whenever possible. The production with grid is forced for all flat and bent meshes, and for cages without corner bars. Cages with corner bars are an exception, because there the positions of the corner bars must not change to ensure that the cage dimensions after bending are as close as possible to the designed dimensions.

Angle variation for transversal wires

The positioning of the welding portal affects the angle of the transversal bars. This is used to create a small offset between the start and end of each transversal bar so that when bending the cage, the ends of these bars will not collide.

The angle variation is applied automatically for all cages. When producing bent mesh with the single bending heads this is not needed as the ends of the same bar do not overlap nor could collide.





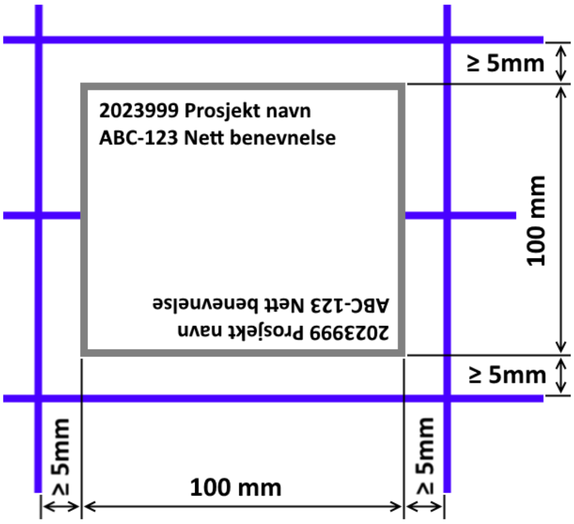
Single wire bending heads

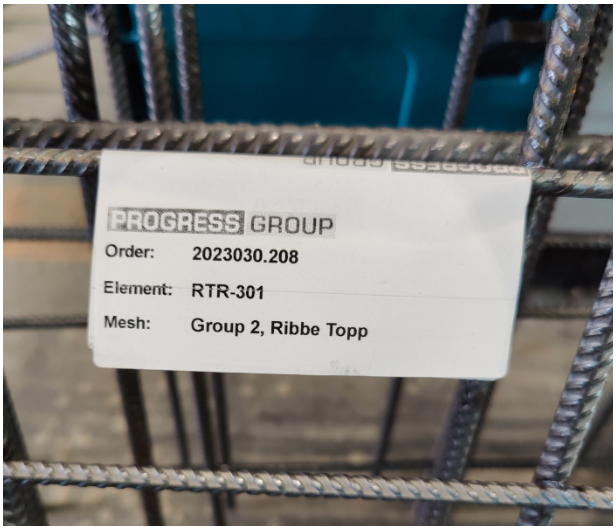
The machine is equipped with two single wire bending heads with bending dies of multiple sizes. The main principle is that all bendings are made one-by-one and it is therefore not possible to produce bendings with crossing wires.

Mesh label placing requirements

The mesh labels are used to identify the project, element, and name of each mesh or cage.

The label dimensions, placing requirements and an example label are shown below. The machine automatically prints a label for each mesh, and sticks it to a suitable longitudinal wire, if it finds one.





When a suitable label position cannot be found, the machine will print the label, and then stop and wait for the operator to place it by hand.

Design of meshes

Catalog mesh is no longer ordered

Nett	Katalog	Nettmaskin
K131	ø5cc150	**ø6cc215**
K189	ø6cc150	ø6cc150
K257	ø7cc150	**ø8cc195**
K335	ø8cc150	ø8cc150
K402	ø8cc125	ø8cc125
K503	ø8cc100	ø8cc100

Catalog mesh is no longer ordered, and all meshes are produced using the mesh bending machine.

Meshes should be modelled in Tekla using the crossing rebar groups or rebar sets. Mesh tool is discouraged as it is more difficult to adjust diameters and spacings or add bendings. The K131 and K257 must be adjusted, because ø5 and ø7 wires are not available.

The «catalog» meshes are given only for reference, statically required or minimum reinforcement should always be used. Consider setting spacings in multiple of 25 mm for production with grid.

Note that there is a difference between a catalog mesh that is ordered from an external producent and a project-wise standard mesh that is produced by us. The latter is still fine to use.

Possible wire combinations

	Ø6	Ø8	Ø10	Ø12	Ø16
Ø6	1,00	0,75	0,60	0,50	0,38
Ø8	0,75	1,00	0,80	0,67	0,50
Ø10	0,60	0,80	1,00	0,83	0,63
Ø12	0,50	0,67	0,83	1,00	0,75
Ø16	0,38	0,50	0,63	0,75	1,00

According to NS 3576-4-2005 section 5.3, the wire diameters in a custom mesh must fulfil the following condition:

$$d_{min} \geq 0{,}7\, d_{max}$$

Ratios for all wire combinations are shown to the left. We assume that Ø12/Ø8 mesh fulfils the requirement on the basis of rounding to 0,1.

Spacings between wires

The minimum and maximum spacings vary in the longitudinal and transversal directions and depending on whether the mesh is produced rastered or gridless.

Type of wires and production mode			Center-center spacing between wires				
			Ø6	**Ø8**	**Ø10**	**Ø12**	**Ø16**
Longitudinal	Min	Rastered	25				
		Gridless	50				
	Max		-				
Transversal	Min		38	40	42	44	48
	Max	Rastered	500				
		Gridless	450				

For longitudinal wires in rastered mode all spacings must be in a module of 25 mm (25, 50, 75, …). In gridless mode spacings can be varied by 1 mm (50, 51, 52, …).

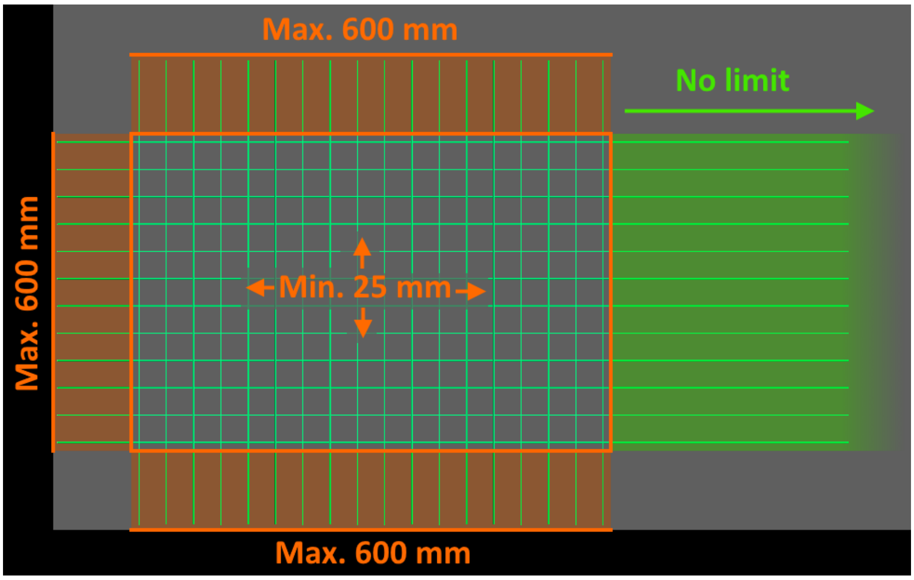
For transversal wires minimum clearance between the bars is 32 mm. The maximum center-center spacing is 500 mm in rastered mode, or 450 mm in gridless production.

The longitudinal spacing below 50 mm or transversal spacing above 450 mm in a gridless mesh gives a warning and forces rastered production. The maximum of 500 mm can be exceeded but generates a warning to ensure special attention from the machine operator.

Overhang length limits

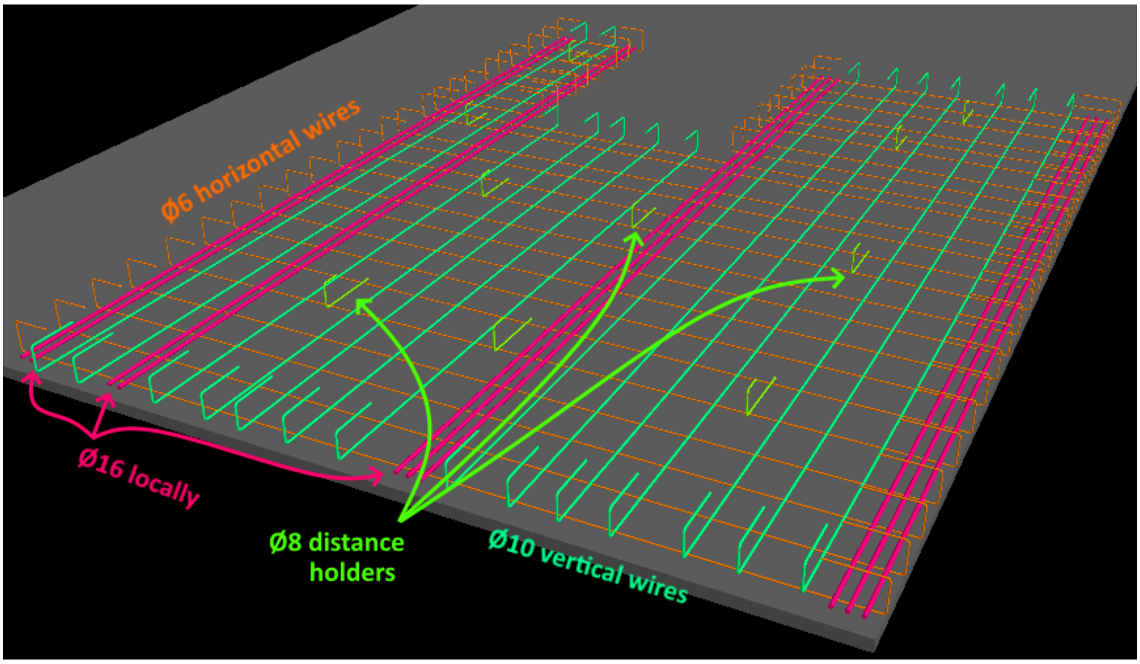
The overhang, or “flying end” length is measured from the center of last crossing wire to the bar end, including all bendings. It is generally limited to a minimum of 25 mm and a maximum of 600 mm. The picture below shows the limits in an unrolled mesh (before bending).

The maximum overhang length is not limited in the trailing end of the mesh. There any length is possible because the wires dragged behind the mesh do not cause any problems in production.



Varied diameters in meshes

All types of meshes can be modelled with varying diameters in both directions. Thicker or denser wires can be used to strengthen specific areas or as the edge reinforcement.



Note that for cages (produced flat and bent with the manual bender) mixed diameters may cause uneven bending and should be avoided, as the bigger bars require more “over-bending”. Instead adjust the spacings or include loose stirrups of any diameter to be supplied manually.

Bending of meshes

Available bending dies

The bigger bending dies require longer segments and more free space around the bending.
The possible combinations of wire diameters and bending dies are shown in the table below.

Wire \ Die	D25	D32	D40	D48	D64
Ø6	×	×	×	×	×
Ø8		×	×	×	×
Ø10			×	×	×
Ø12				×	×
Ø16					×

The machine chooses the smallest possible bending die, equal to or greater than the designed radius. Bendings that are too big are reduced to D64. Apart from that, a smaller die is never used, even if it is otherwise not possible to produce the bending.

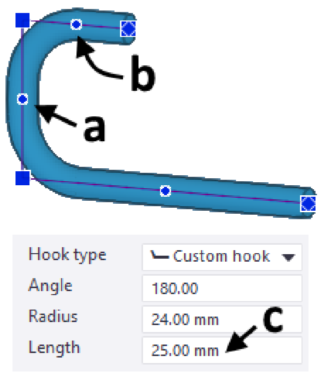
When radius changes automatically to match the available dies, it affects the real length of the bar, and for bendings above 90° also the bending "height".

Required length of bent segments

The minimum segment length depends on the bending radius (the bigger heads require longer arm), and, for bendings below 90 degrees, the actual bending angle (including overbend).

Summary of minimum segment lengths in mm for bends from 90 to 180 degrees is shown below.

Nominal diameter	**Ø6**	**Ø8**	**Ø10**	**Ø12**	**Ø16**
Actual diameter	**8**	**10**	**12**	**14**	**19**
Bending die (smallest)	**D25**	**D32**	**D40**	**D48**	**D64**
In AviCAD (nominal diameters)					
Middle segment / a	57	58	73	91	129
End segment / b	41	38	48	61	89
In Tekla (actual diameters)					
Middle segment / a	55	56	71	89	126
End segment / b	40	37	47	60	88
Hook length / c	24	16	21	29	46



The maximum segment length is not limited on its own, but the maximum overhang length should be considered. The bending height should not exceed 300 mm, or a warning will be shown.

The minimum segment length increases with the bending angle, so the bending bolt does not lose the bar during the bend. The increase becomes insignificant for bending angles from 90 degrees. Because the segment lengths are defined in terms of points along the bar centerline, in practice the required length in millimeters is the same for bending angles from 90 to 180 degrees.

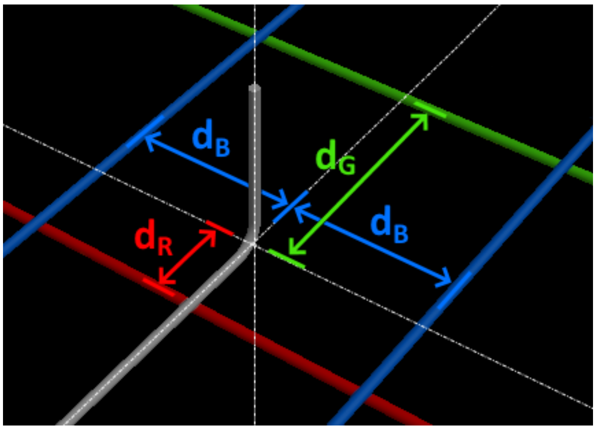
The angle correction in the machine affects the actual bending angle when overbending the bar to ensure that the angle after bending is as designed. Because of that it also affects the minimum segment length for angles below 90 degrees but can be neglected from 90 degrees and above.

Spacing around bendings

Minimum distances between center of bending and center of neighbor bar are summarized below.

The table assumes that minimum possible bending die is used and that the neighbor wire is of the same diameter as the bent wire. The decimal values are rounded up to the nearest integer.

Wire size	Min. die	Min. cc spacing in mm			
		Red d_R	Green d_G	Blue d_B	
Ø6	D25	37	71 / 73	48	46
		18	104 / 106		
Ø8	D32	46	77 / 79	50	47
		30	106 / 108		
Ø10	D40	53	83 / 85	52	48
Ø12	D48	63	89 / 91	54	49
Ø16	D64	99	101 / 104	66	53



The alternative bending mode allows placing the bending closer to the red bar but requires more distance to the green bar instead. It is available only for the Ø6 and Ø8 bars in the upper layer.

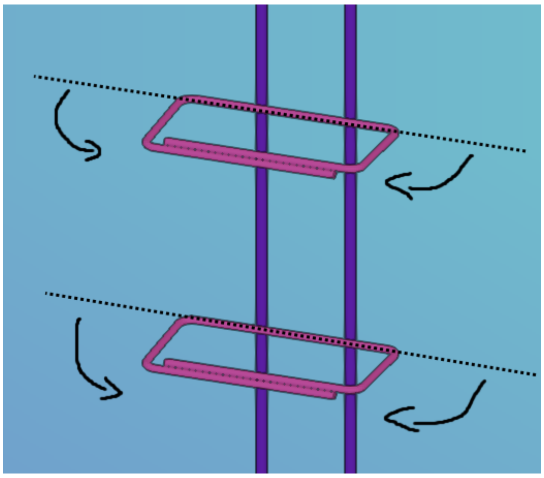
Note that Tekla uses actual diameters while AviCAD and machines work with nominal diameters. The conversion on export can shift the position of bending and additional tolerance equal to the difference should be added to the green spacing. This is the second value in that column.

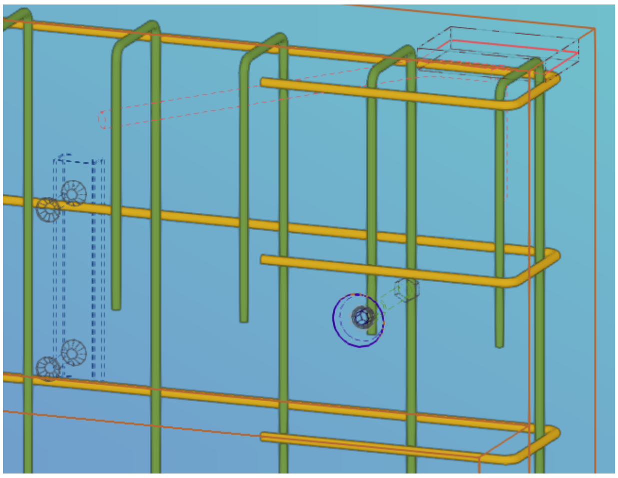
The spacings to the left and right parallel bars in blue vary depending on which side the bending head threads in. Only one side can use the smaller spacing, but the side can be chosen freely.

To adjust the spacing for different size of a neighbor bar, add `(Øother - Øbent) / 2`. For example, spacing of Ø12 bent bar to Ø6 green bar will be: `89 + (6 - 12) / 2 = 89 - 3 = 86 mm`.

Parallel and crossing bendings

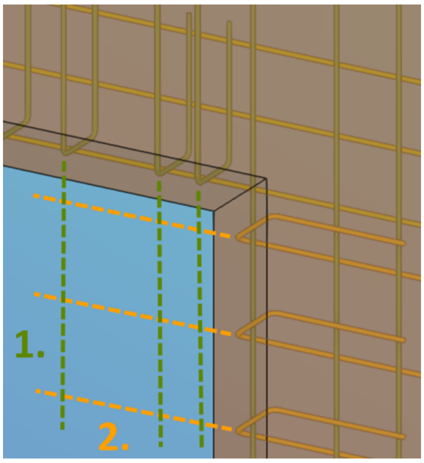
It is perfectly fine that the legs of the bending are overlapping. The machine will know the order of the bendings by checking which legs are higher in the final shape and will make these last.

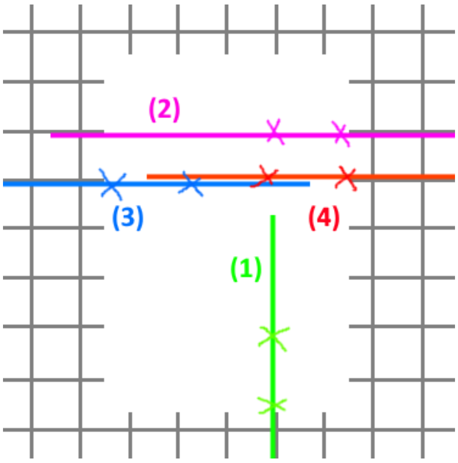




Bars that overlap before bending

Bent segments that overlap in the unrolled mesh can be produced without problems. The machine understands that the upper wires must be bent first, and the welding robot omits the crossing points, so that the bars are not welded together.



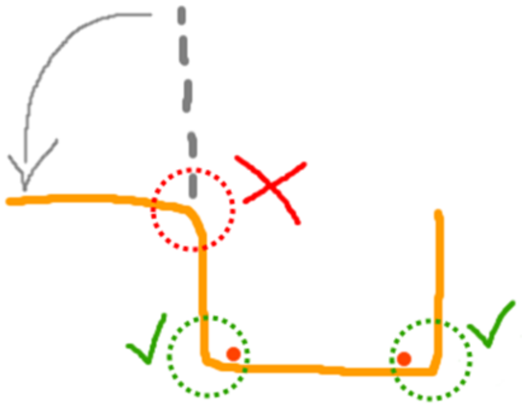


In the diagram above, the bar **(1)** is perfectly fine, there is enough space around each bending.

The bar **(2)** is fine if enough clearance is added between it and the parallel bar, and the crossing bar on the other side of the opening is below and does not block the bending.

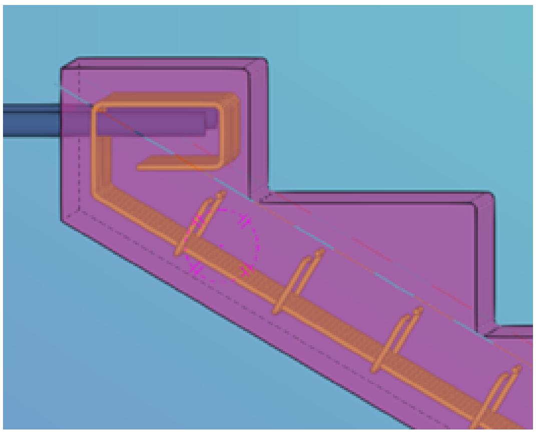
The bars **(3)** and **(4)** are fine if enough clearance is added between them (now they overlap).

Bendings in opposite directions



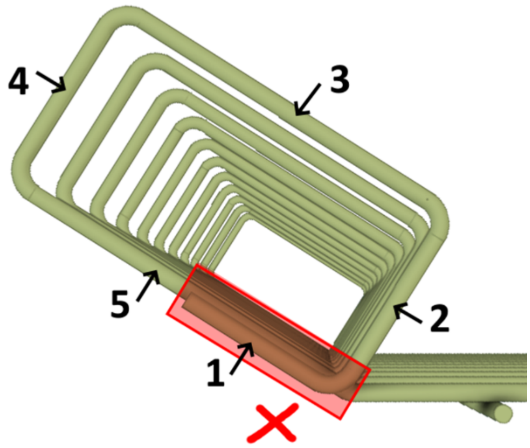
Bendings in opposite directions cannot be produced. The machine will give an error that negative bending is not allowed. The mesh could be produced, but such bending will not be made. The bar is not shortened for this bending, so the bending could be done afterwards manually. In our factory it is not possible to bend such bars manually so this should be avoided.

Non-perpendicular bendings



Non-perpendicular bendings can be produced, any angle between 0 and 180 degree is possible to bend. The length of the flying end (overhang) should be considered, the maximum is 600 mm.

Closed shapes with doubled bendings

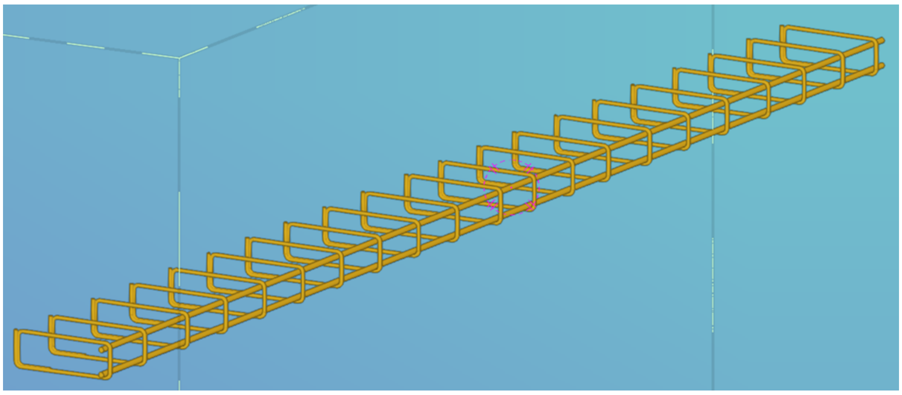


A closed shape with a doubled bending is not possible to produce using BlueMesh. The segments 1 to 4 can be bent fine, but during move to bend segment 5, the bending head would collide with the marked part of segments 1 and 2. If segment 1 was removed and segment 2 shortened or 3 lengthened to avoid hitting the bending head, it could be fine to produce.

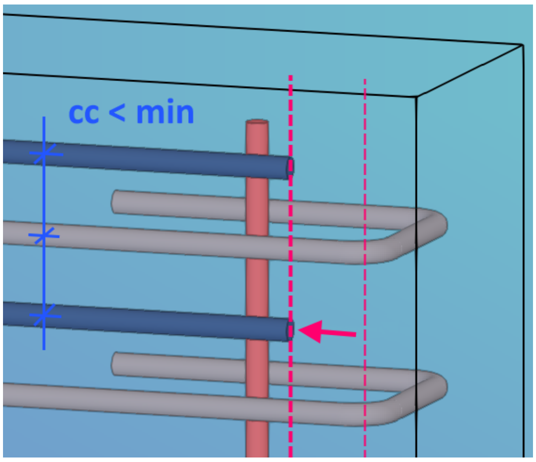
The preferred solution here is to use a separate cage to be attached later, and either a flat mesh, or a bent mesh with only the segment 5.

Bendings with corner bars

BlueMesh single bending heads cannot make bendings with corner bars in the curved section. The minimal offset to crossing bars is quite large, so when bending on BlueMesh the corner bars would need to be added manually later. However, in cases like console cage we need to add some additional long bars for stability anyway, so these are best produced flat and bent using Hambi.



Making space for the bending head



When spacing of bars parallel to the bent bar is below the required minimum (blue bar), it may not be possible to make the bendings because the neighbor bars will collide with the bending robot. Even if a rebar splitter is used to reduce the number of bendings, the remaining straight stubs will still cause the same issue.

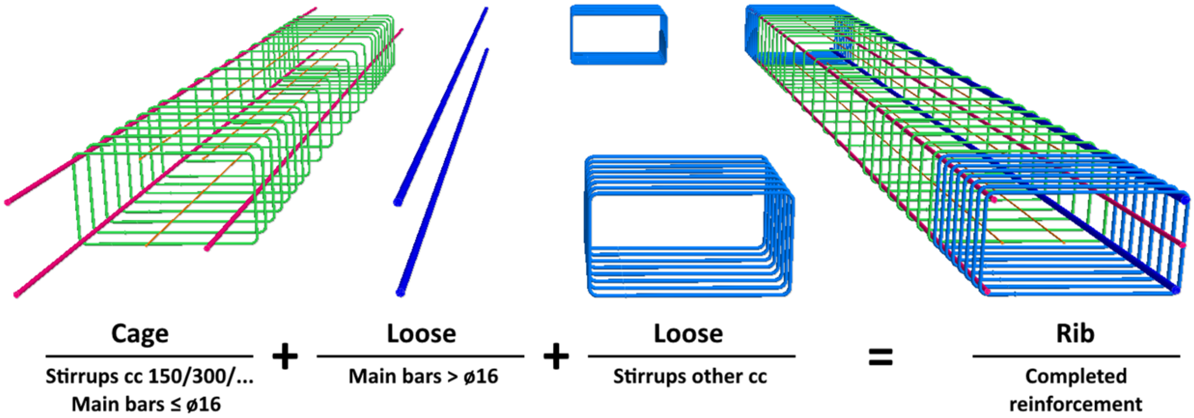
In such cases the splitter needs to be moved further away from the edge (as required for the red bar) to allow making the bendings.

Design of cages

Cages are meshes bent manually

Cage type is used when bendings in a mesh cannot be produced by the mesh welding machine, most often because of crossing rebars in the bent segments. The single wire bending heads make bendings one by one – the crossing rebars prevent it. The cages are therefore produced flat and bent manually with the Hambi mesh bender. The cage geometry can be open (U, L) or closed (□).

Because of limitations of the Hambi mesh bender described in detail below, a beam or sandwich rib reinforcement is normally produced as a combination of one or more cages and loose reinforcement welded together in the reinforcement hall. A typical example is shown below.



The reinforcement parts are produced and assembled in the following order:

1. **A cage with stirrups** spaced in a module of 150 mm (150, 300, …). Each bent bar must match a bending cam which are in spacing of 150 mm (subject to be reduced).

   Every cage should have 4 or more longitudinal wires to support it when bending. When there are fewer main bars, supporting ø6 wires should be added. The minimum for small cages is 3. As a rule of thumb, spacing of about 200 mm can be used. If necessary, the supporting wires can be later cut out, for example when they collide with the inserts.

2. **The main bars** up to ø16 should be included in the cage. Bigger bars are produced manually and then inserted and welded into the bent cage, according to the production drawing.

   Reducing the number of loose bars is beneficial, therefore it should always be considered. For example, 4ø20 bars could be replaced with 6ø16 bars etc. Remember that not all main bars must go all the way to the ends but can be placed according to the moment diagram.

3. **Denser stirrups** can be exported as loose reinforcement to be produced in Pluristar and added in the ends. Spacings should be clearly presented in the production drawings.

   Adding stirrups between cage parts prohibits integrating main bars and should be avoided. Alternatively open stirrups which can be inserted into the cage can be considered.

Production of cage is beneficial when reinforcement consists of 5 or more stirrups. Below this number it is more efficient to export everything as loose reinforcement.

The old MEP mesh bender may still sometimes be used, but in principle all cages should be designed for bending with Hambi. The MEP bender does not allow bars in the corners and requires a minimum of 40 mm center to center spacing between bending and longitudinal bar.

Geometrical requirements for cages

The cages are always bent before loose bars are welded in. The maximum size of the **longitudinal / corner bars is ø16** and of the **bent bars / stirrups is ø12**. Bigger bars must be exported as loose.

Hambi allows bending of meshes with **length up to 10,0 m** and **width up to 4,6 m** (as BlueMesh). The overall cage weight must be considered. While a dedicated crane is now available for bending, smaller cages which can be handled by two workers should still be preferred when possible.

Normally **at least 4 longitudinal wires** should be designed, while 3 is the minimum for small cages. When there are fewer main bars, supporting ø6 wires should be added in spacing of about 200 mm. The supporting wires can be later cut out, for example if they collide with the inserts.

The min and max segment lengths (including end hooks) are summarized below. Shorter segments can only be achieved with 180-degree bendings, then the bending inner height is 40 mm.

Corner bars	Minimum		Maximum
	End segment	Middle segment	
With corner bars	110 mm	110 mm	1 600 mm
Without corner bars	110 mm	120 mm	1 000 mm

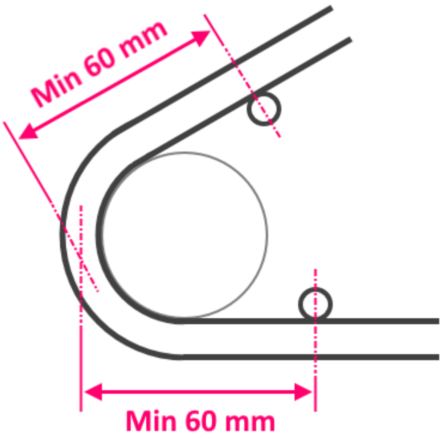
Note that the hook length specified in Tekla is the straight segment length, while the table above shows the external segment lengths. For the 90-degree hooks the length can be calculated as:

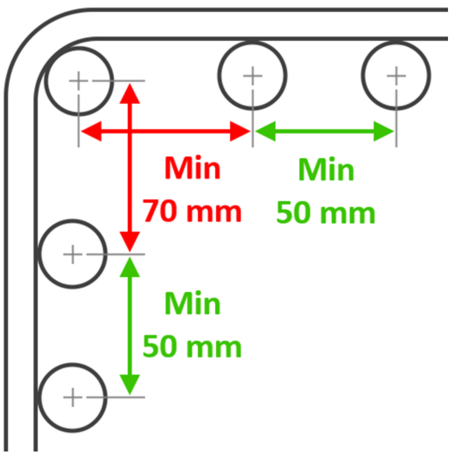
```
(length) = (external length) - (bending radius) - (bar diameter)
```

The cages must be modelled with the correct **bending diameter of 40 mm**. The positions of bars in unrolled cage depend on the modelled bending diameter. When it is different than actual diameter, the dimensions of bent cage will be wrong. In Tekla the radius should be specified instead (20 mm). The value must be set both for the rebar and for the end hooks.

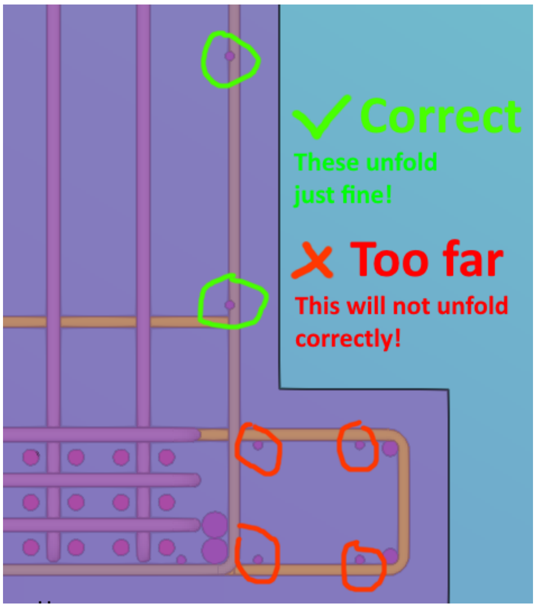
The **spacing of bending cams is fixed at 150 mm** (subject to be reduced). Stirrups can only be bent at the bending heads, not between, therefore the **possible spacings are 150, 300, … mm**. The bending cam positions can in principle be adjusted, but this is time-consuming and must be agreed with the factory in advance, and only when producing in bigger volumes.

When longitudinal bars are not directly in the corners, a minimum distance to the bending of 60 mm is required to avoid collision with the bending heads. With a bar in the corner, the spacing to the next bar must be at least 70 mm to insert the bending head. Otherwise, the minimum spacing is decided by BlueMesh. As cages with corner bars are produced without grid, this is most often 50 mm.





Unfolding of reinforcement



The two main reasons that can cause problems with the unfolding of reinforcement in AviCad are:

1. Invalid or missing reinforcement types.
2. Too big distance between long and cross bars.

This is often related to the use of actual diameters in Tekla and nominal diameters in AviCAD. There is a few millimeters of tolerance for connecting the bars, but it may be too little when there is a gap between the rebars in Tekla.

The bars that should be grouped together should therefore be modelled in Tekla completely next to each other, or even with a slight overlap.

Design of loose bars

Automatic production of rebars

The Pluristar automatic stirrup bender can produce bars of length up to 12 m, and bend wires from ø6 to ø16. Our machine is equipped with ø8, ø10, ø12 and ø16 coils and only these diameters can currently be produced automatically. Other diameters and longer bars are produced manually.

The production can be done by one of the two submachines depending on the selected bending die. Automatic die assignment is based on the rebar geometry, but it is not always right, and operators often change the bending die manually based on experience. Therefore, the die and machine shown in the PTS results will not always be correct, and subsequently some warnings may not be applicable and can be ignored. For example, too short segment length in case of stirrups for DT/SDT.

The two submachines marked in the picture above have slightly different characteristics:

- **BaMD/3D** is used for three-dimensional and more complex flat shapes. Production is slower because the bending requires operator's assistance. Shorter segments can be bend.
- **2BK_MA** is used for cutting straight bars and simple bending of longer bars. This machine is faster, because bending is fully automatic, and bars are collected by the special hooks.

Dies for bending of loose bars

The table below gives an overview of bending dies for both Pluristar submachines and wire sizes for which they are applicable. For each wire and die combination a minimum external segment length for 90° bending is given. The measurements for other bending angles will change. The requirements differ for start and middle segments (between two bendings), and there is no restriction for end segments which can be of any length.

Machine	Matrix	Size [mm]	Start segment [mm] / Wire					Middle segment [mm] / Wire				
			6	8	10	12	16	6	8	10	12	16
BaMD/3D	A	20	45	45				60	62			
	B	25	47	48	46			65	68	68		
	C	32	59	62	60	59		80	85	85	86	
	D	50		93	89	91	88		125	123	127	128
	E (circle)	35	94	98	98	99		116	122	124	128	
2BK_MA	Z	20	76	77	77			86	89	91		
	Y	32		72	72	82	82		87	89	92	96
	X	60		75	76	134			97	100	160	
	W	80			126	126	129			155	157	164
	V	100			122	121	124			156	157	164
	U (crank)	20	54	56	45			55	59	59		

The values are approximate, because they depend on the angle correction (a degree of overbent to achieve the desired bending angle), which is continuously adjusted by the operators, due to varying steel quality of different coils.

The bigger bar diameters which are not possible to produce with Pluristar are cut manually and can be bent using the B26 manual bar bender. The machine is equipped with D60 bending die and can be used for making simple bends of bars up to ø25.

Overview of possible rebar lengths

The table below shows the minimum and maximum possible bar lengths depending on the machine. Loose bars too long to produce with Pluristar are automatically assigned to be produced manually.

Diameter	BlueMesh		Pluristar		Manual		
	Min	Max	Min	Max	Min	Stock	Max
Ø6	350	10 000					
Ø8	350	10 000	50	12 000	50		12 000
Ø10	350	10 000	50	12 000	50		12 000
Ø12	350	10 000	50	12 000	50	12 000	14 000
Ø16	350	10 000	50	12 000	50	12 000	14 000
Ø20					50	12 000	18 000
Ø25					50	12 000	18 000
Ø32					50	12 000	18 000

The bars shorter than the minimum or longer than the maximum generate errors in PTS and must always be re-designed, often by splicing them into shorter overlapping bars.

The bars longer than the stock length generate warnings in PTS, as such lengths must be pre-ordered. Every use of such long bars should be agreed with the production planners in advance.

Reinforcing elements

Tolerances for bar end positions

The mesh welding machine has bar length tolerance of ±5 mm and bar placing accuracy of ±10 mm. Since each bar can be both cut shorter or longer and at the same time also placed incorrectly, the tolerances compound, and thus the total tolerance for bar ends can be up to ±15 mm. Overall the machine has a good precision, and such big deviations occur very rarely.

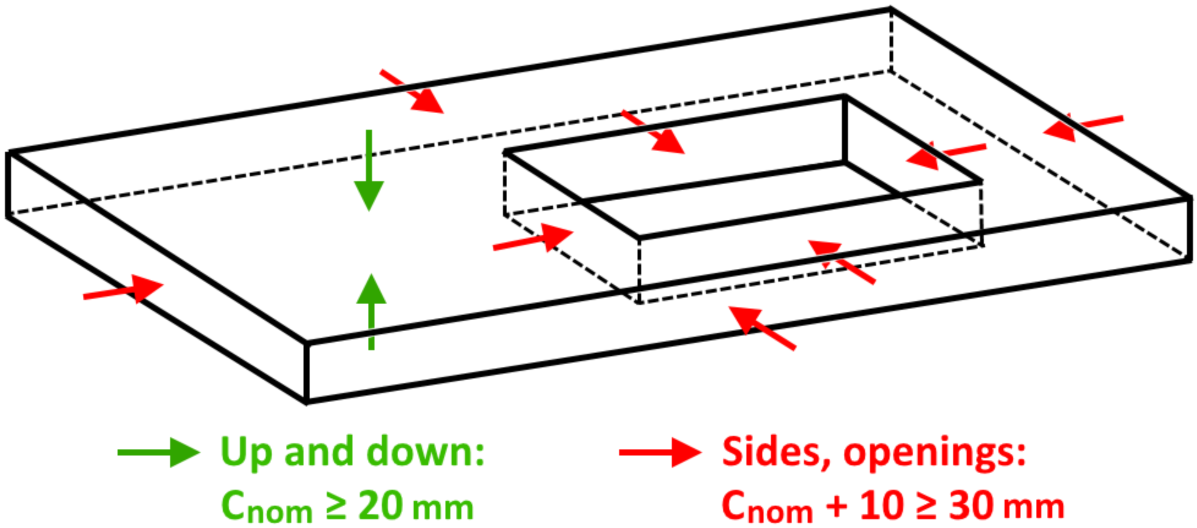
The practical tolerances that should be used in modelling are described in the following section.

Tolerances in form and concrete cover

The concrete cover **up and down in the form** can be designed according to the requirements without the additional tolerance but should be at least 20 mm (applicable for X0 and XC1). When meshes are placed on top of each other, there should be exactly 5 mm between them. This considers bending imperfections and height built-up of ribbed bars, in practice giving the overall "height" as designed.

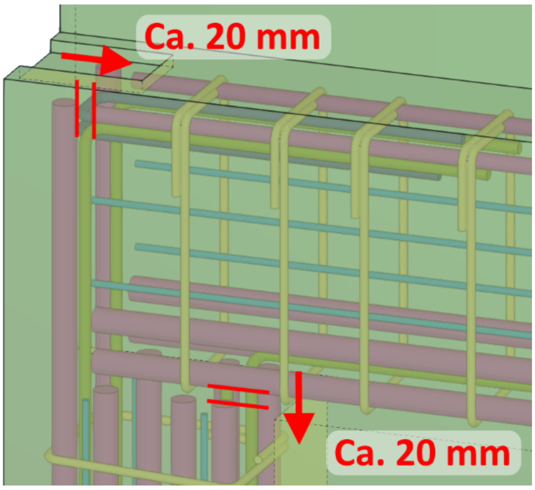
The concrete cover **in the sides of the form**, including towards openings, must consider detailing (chamfers) and should be increased by 10 mm in each side. It should always be at least 30 mm. Notches and chamfers will reduce the actual cover, and the formers will take up space available to place the mesh in the form. This should be considered when modelling the reinforcement.

The diagram below summarizes the practical tolerances for reinforcement. The concrete cover specified in the drawings should always be according to the actual requirements (C_{nom}).



The concrete cover **for stairs should be at least 30 mm in all directions**, because of the complex form and non-perpendicular bending angles, which require the additional tolerance.

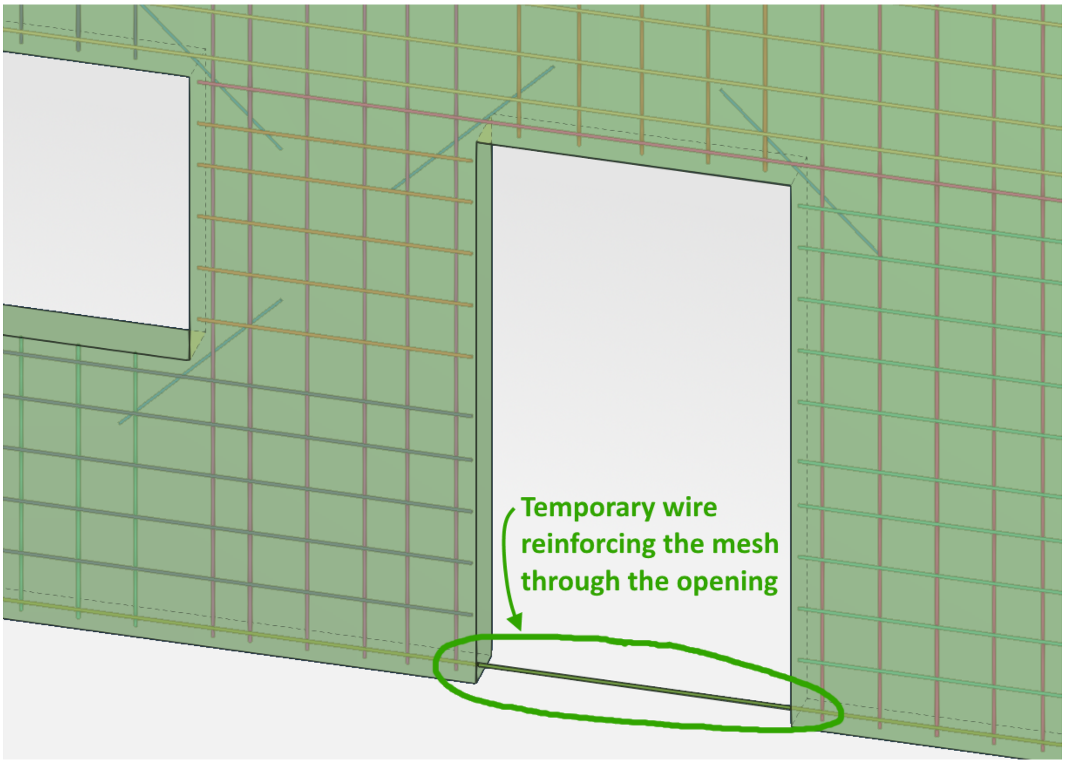
Tolerances at ends of reinforcing bars



Reinforcing bars placed towards other bars should be modelled with a tolerance of about 20 mm at the ends, to ensure that the complete reinforcement can be correctly assembled and fits in the form. This applies to both meshes, cages and loose bars.

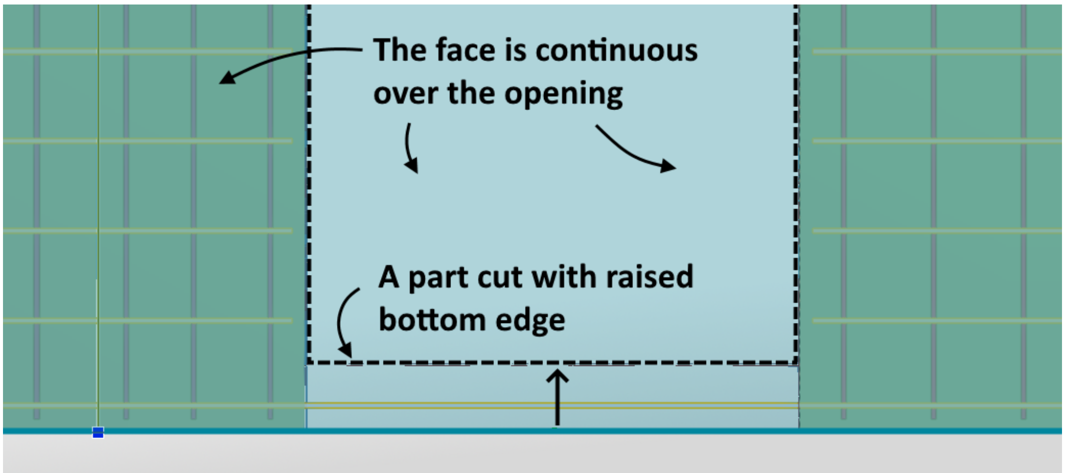
Supporting bars for openings in edge

The transport of a mesh with a big edge opening is challenging and can cause the mesh to distort or welds to break. Because of this such openings should be temporarily reinforced with a wire going through the opening and connecting both sides of the mesh.



This improves the mesh stability during handling and transport. The temporary bar can be cut away when the mesh is placed in form or even kept until the wall is installed at the construction site.

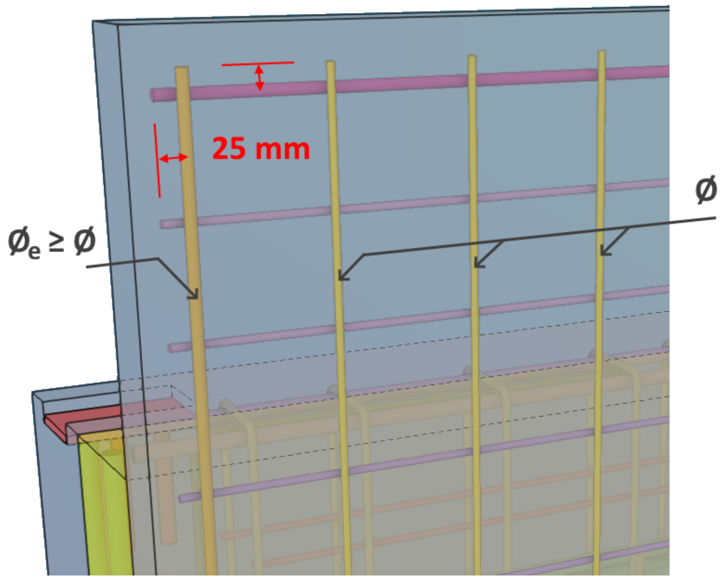
When using rebar groups such a wire is straight forward to add. In case of rebar sets, it is simplest to keep the face over the opening and use a part cut with the lower edge raised over the bottom wire.

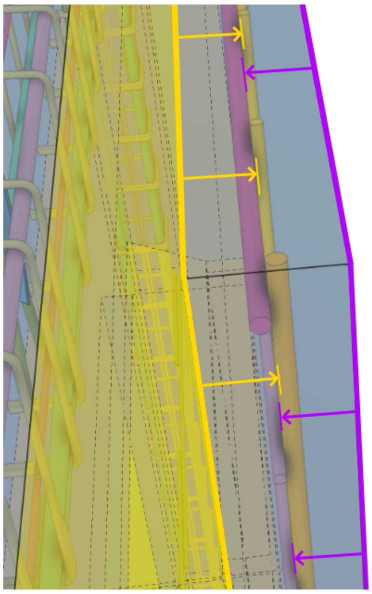


Modelling of edge reinforcement

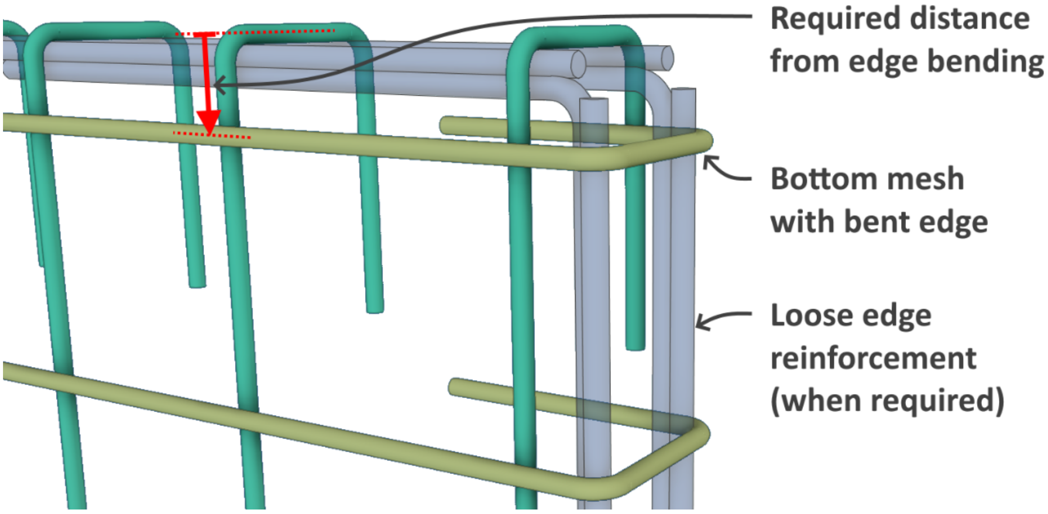
Previously, when catalog meshes were used, the actual placement of wires was uncertain, and therefore the edge reinforcement had to be added separately. With mesh welding machine the position of every wire can be fine-tuned and therefore, when possible, the edge reinforcement should be modelled as part of the mesh, also if the diameter of edge bars is bigger than in mesh. The goal is to limit the number of pieces and the amount of manual work that is needed.

It is required that crossing wires stick out 25 mm past the center of the edge bar. Bars in rebar sets are aligned towards the shell face, which creates a gap between layers when diameters vary. To keep the bars together, the outer layer can be modelled with face on the inner side of shell with adjusted concrete cover. Alternatively separate single bars can be used to model the edge reinforcement.





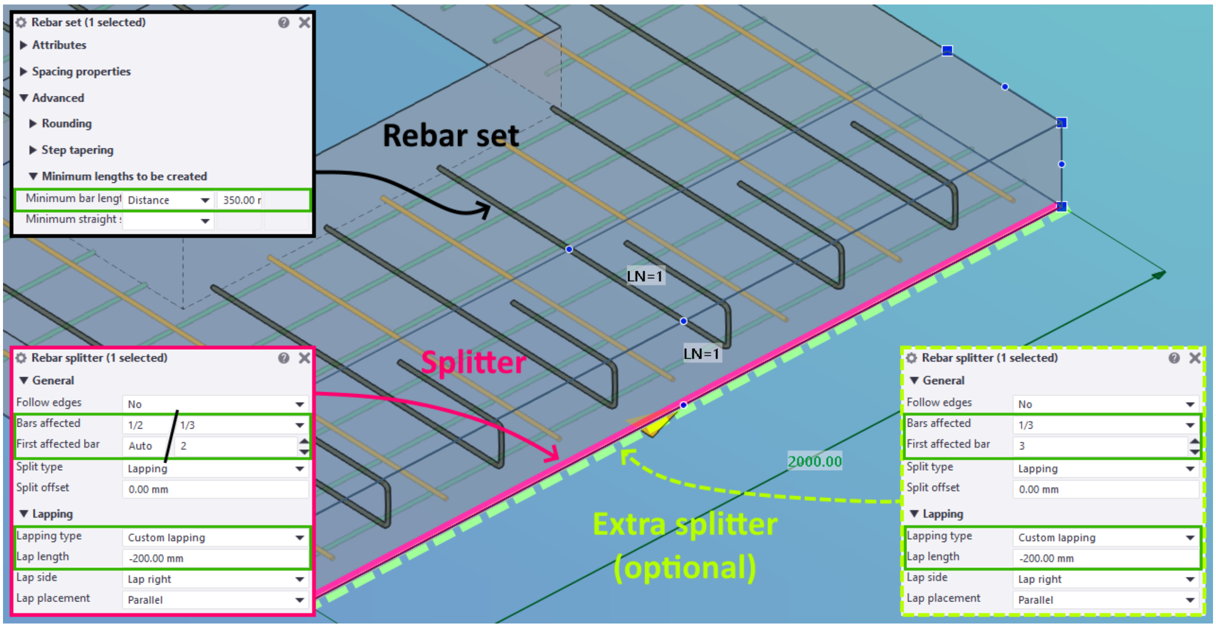
For mesh with bent edge, like bottom mesh of compact wall, the last bar must be moved further away from the edge to make space for bending. When distance from edge becomes too big it may be necessary to model the edge bars as loose reinforcement so they can be placed where necessary. The designer should also consider if the edge bars should be enclosed with stirrups. The various cases are described in Rules for edge reinforcement.



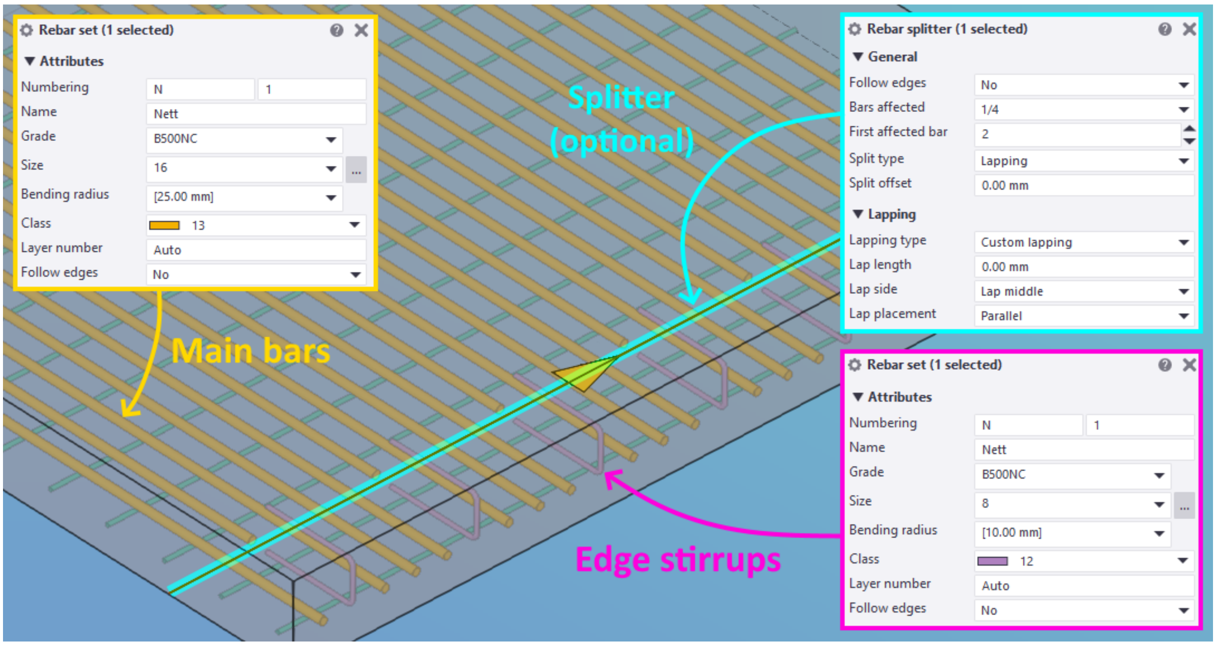
Reduction of edge stirrups amount

The number of edge stirrups should not exceed much what is necessary to support the top mesh or what is required from calculations. The reduction of edge stirrups is especially important in heavily reinforced elements, where the quantity of reinforcing steel that could be spared is significant.

The first method uses one or more rebar splitters. Minimum bar length should be defined in the rebar set, then a negative split lap length can be used to remove the unwanted parts of rebars. Additional splitter can be added in the same edge (with shifted first affected bar) to remove other ratio of edge stirrups than 1/2, 1/3 or 1/4. This method is recommended for ø6 – ø12 wires.



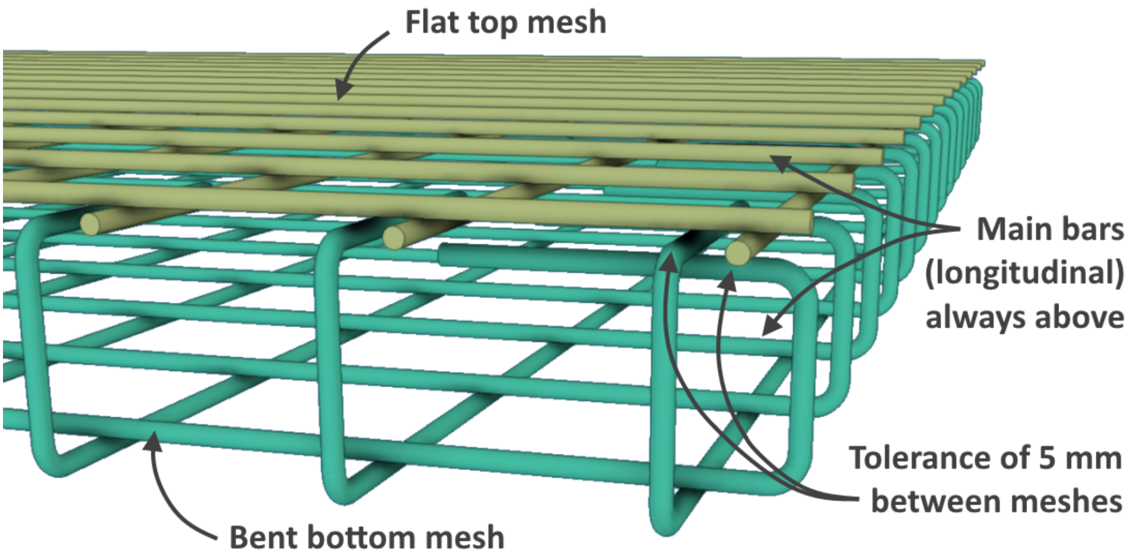
Next method uses stirrups of smaller diameter modelled separately, to reduce usage of thick wires. When there is too little spacing between the original bars, a splitter can be used to shorten some of them to make space. This method should be used for ø12 – ø16 wires.



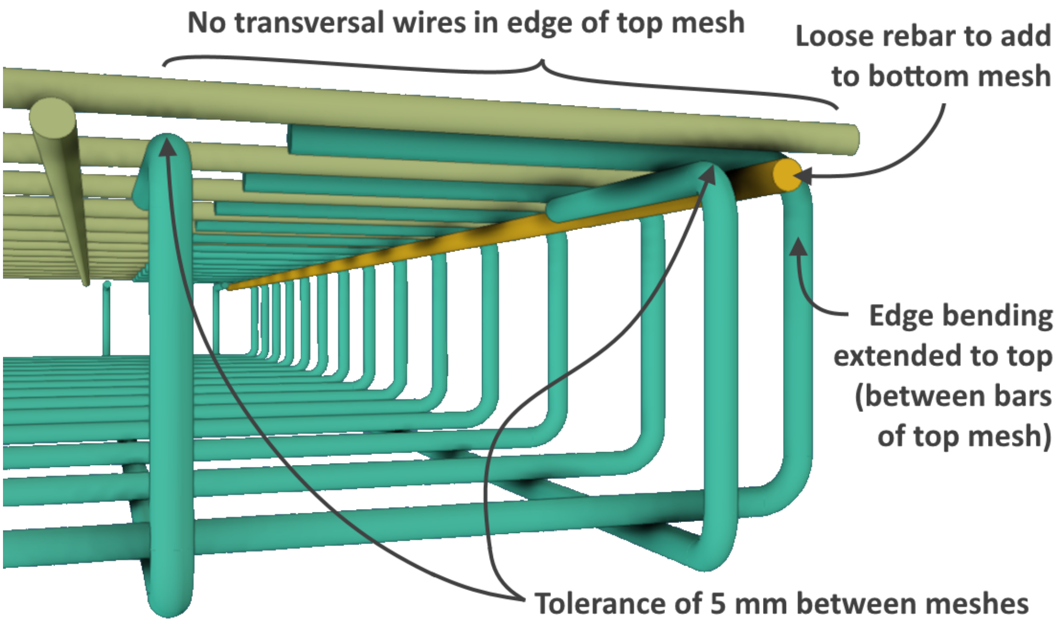
Reinforcement of compact elements

The standard solution for compact elements (walls and slabs) with two layers of reinforcement consists of a flat top mesh that can be placed on a bent bottom mesh directly as produced, without a need to supplement any additional bars. This is the most effective method and to be preferred.

The longitudinal wires should be modelled above in the form for both meshes, this simplifies handling, particularly when additional main bars must be supplemented. A height tolerance of 5 mm should be modelled between the bottom and top meshes.



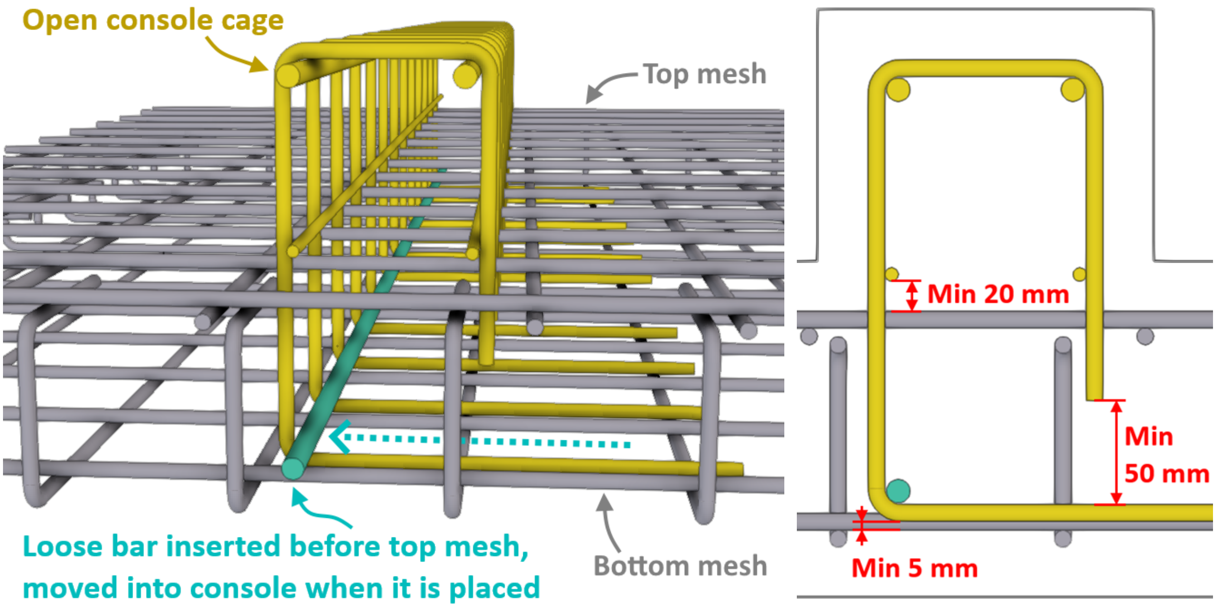
When more space is needed in the element edges, particularly when edge reinforcement is required, an alternative solution is possible. The edge bendings of bottom mesh are extended to the top and transversal wires of top mesh are omitted. Instead, a loose rebar is supplemented to the bottom mesh, to be used as a support for the top mesh. This method requires an additional production step and should therefore be used only when necessary.



Standard console reinforcement

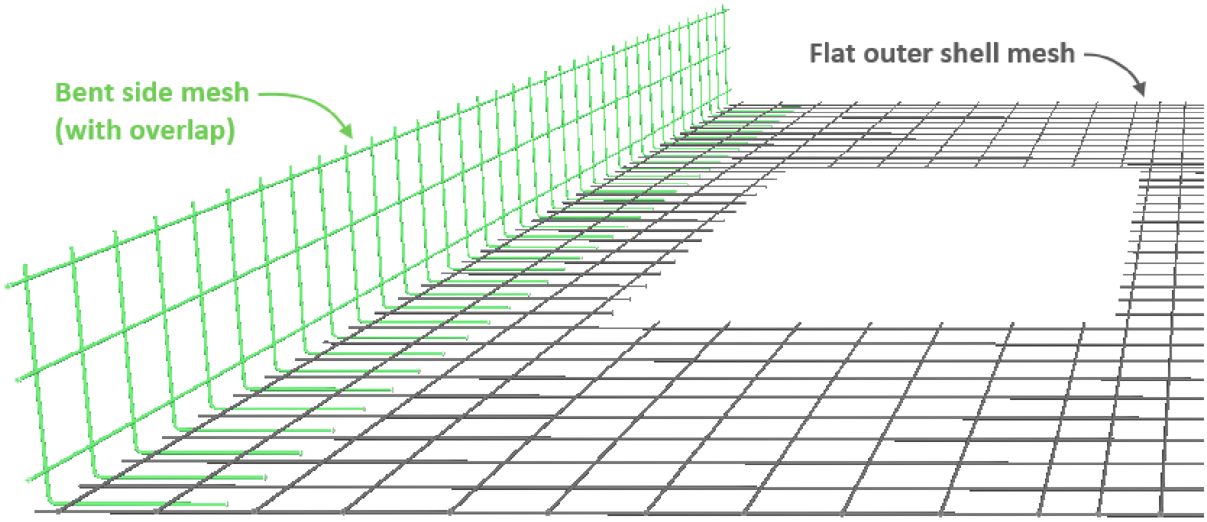
The standard solution is an open console cage that can be placed after top mesh and without cutting of the mesh wires, the gap in top mesh must be big enough. The console should include corner bars and at least two supporting wires with enough distance to the top mesh to allow correct positioning.

The loose corner bar is placed inside before the top mesh, and when the console cage is inserted, it is moved and fixed in the right place. This requires enough clearance towards the bottom mesh.



Outer shell mesh in a corner sandwich

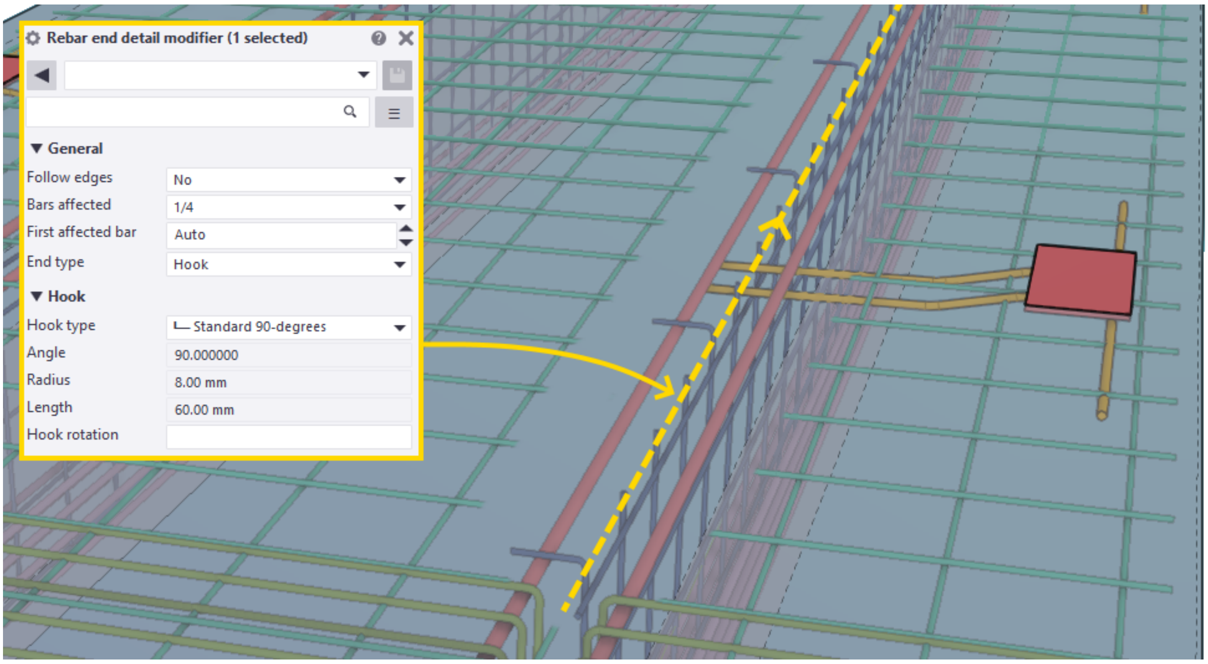
The corner sandwich walls require outer shell reinforcement in two sides. Such mesh cannot be produced as a single piece with BlueMesh because the single wire bending heads can only bend one bar at a time, what is impossible here because of the vertical bars in the side part. Such bars were often produced loose to be supplied later, but the factory prefers another solution with two separate meshes, one flat and one bent, with an overlap. This method is shown in the picture below.



Hooks for hanging of stem mesh

To simplify assembly of the reinforcement in DT elements, the stem meshes should come with hooks in the top edge in every 4th bar. The hooks are used to hang and fix the mesh in the form.

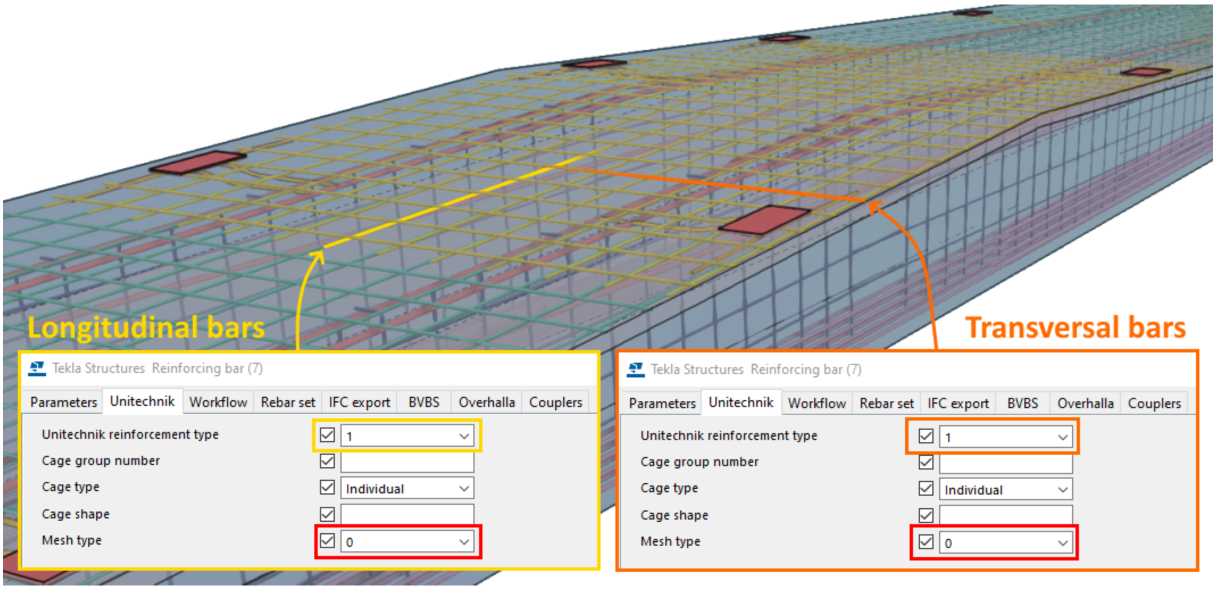
In the SDT elements such hooks should not be designed, and the stem mesh should be kept flat. The form for SDT is different and in practice the hooks hinder the assembly.



Flattening of bent mesh

Some meshes may be modelled bent but should be produced flat. A good example is a mesh in the crest of SDT element, which is modelled with a bending to follow the slope of the element in the model. The bending however does not need to be produced, because the mesh will bend naturally under its own weight. Bending it would require additional work that is not necessary.

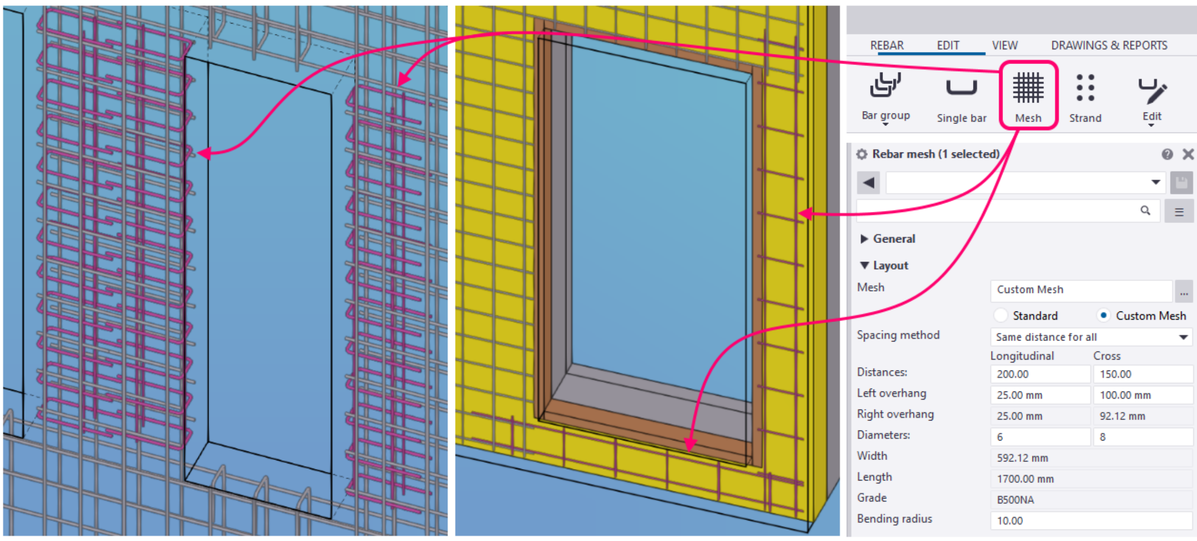
To ensure that the mesh will be flattened by the machine software, all grouped rebars should be assigned mesh type `0 = standard mesh` in the UDA dialog. The reinforcement type `1 = first bar layer` should be set for longitudinal wires and `2 = second bar layer` for transversal wires. This does not match the actual order of layers but ensures correct unrolling.



Project-wise standard mesh

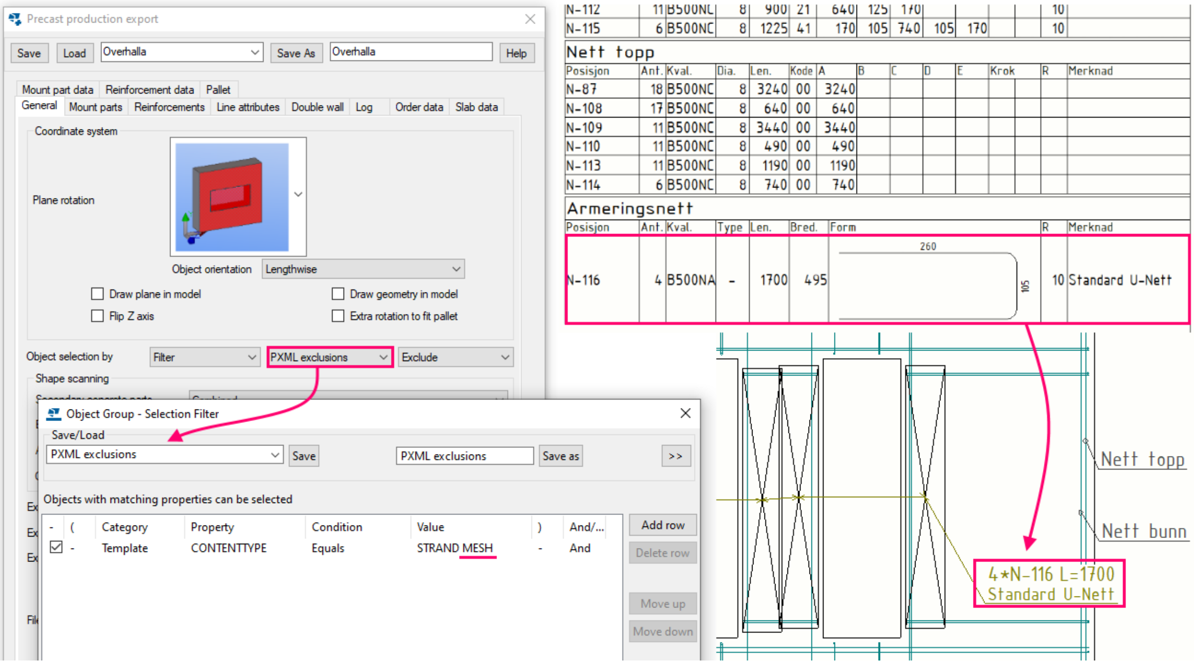
When many elements in a project use very similar mesh that vary only with a handful of parameters, it may be beneficial both in terms of design and production to define a project-wise standard mesh. Sometimes a standard mesh may be the only practical solution. This could be for example:

- U-mesh to be used around openings or as distance holder between the layers.
- Flat mesh to reinforce thin strips not producible as part of the main mesh.



A standard mesh should be first agreed upon with the production, then modelled aside the actual elements, published in a separate PXML file, and sent to the production planner. Then it can be produced in batch or on demand and cut or bend according to the production drawings.

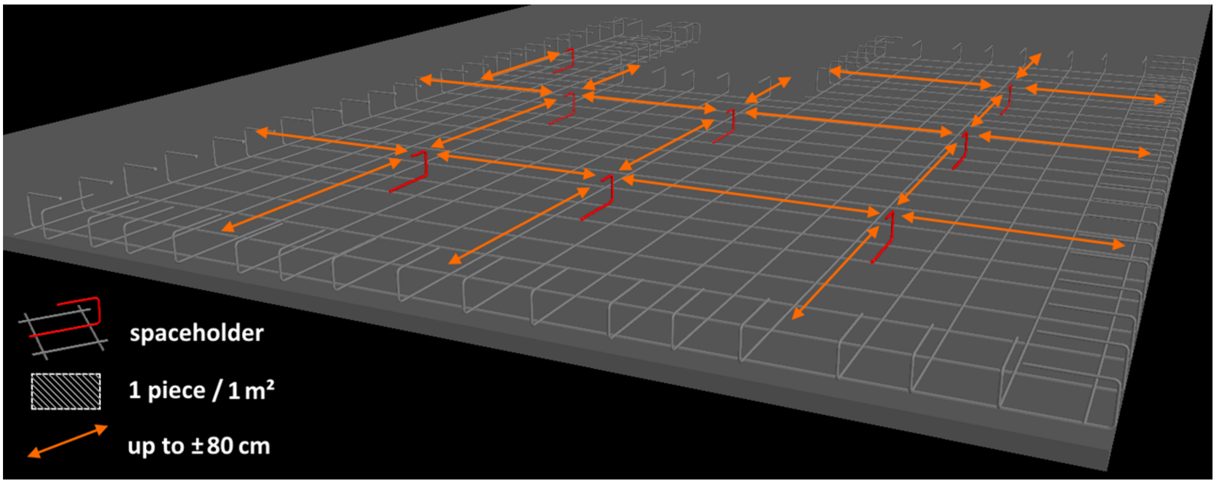
It is best to model these meshes in the actual elements to be shown and listed in the drawings, but they should be excluded from the exported element PXML-s using a filter. If otherwise described in the drawings it is also fine to not model such meshes.



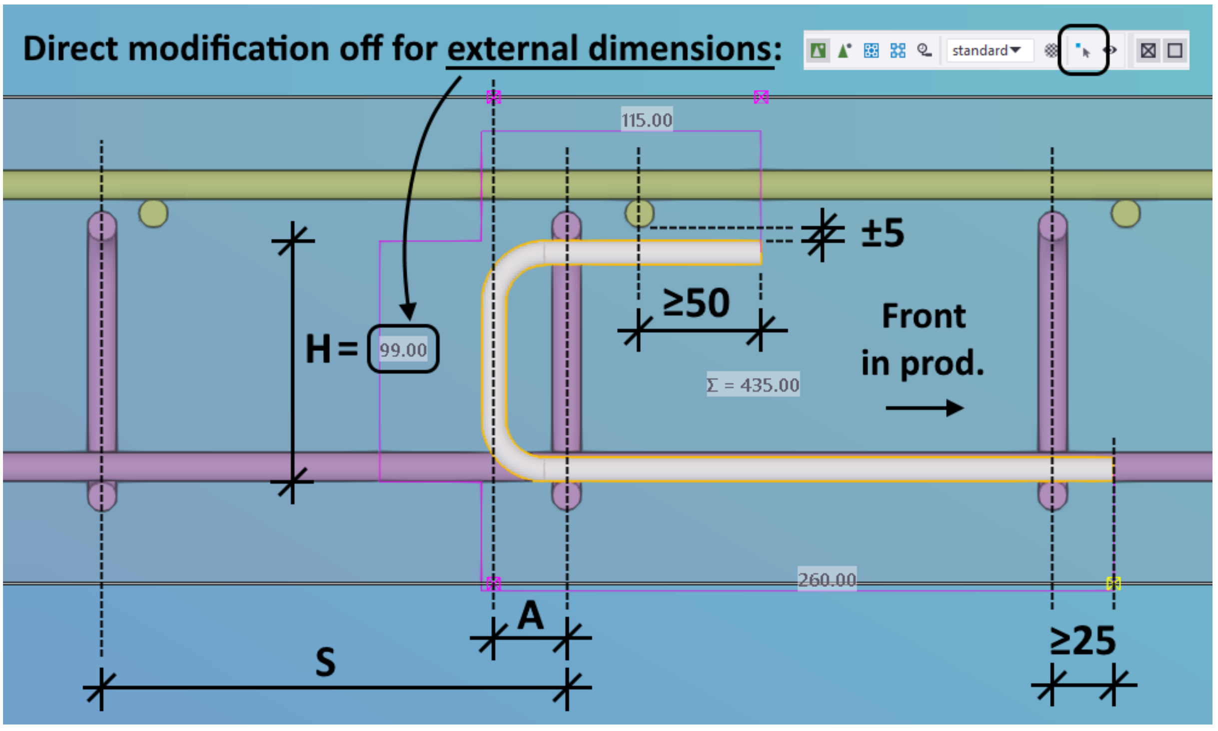
Modelling of distance holders

In compact elements the distance between the bottom and top mesh must be fixed to ensure correct concrete cover in both sides. Before, it was typically done with the U-shaped pieces of K131 mesh, but now the distance holders can be integrated directly in the mesh to reduce the manual work.

The integrated distance holders require adjustment of wire spacing and additional detailing, should therefore be considered mostly in simpler elements, or elements with bigger quantity per drawing. Otherwise, the distance holders can be designed as project-wise standard U-mesh.

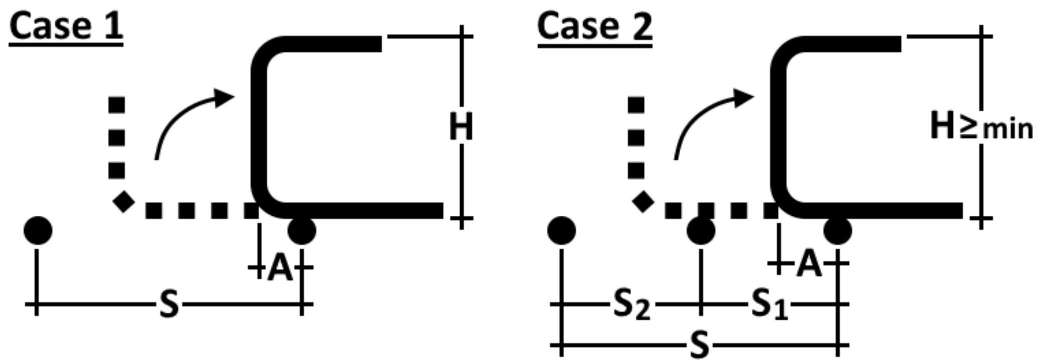


The distance holders should be Ø8/10/12 diameter matching mesh weight, 1 piece per about 1 m² of mesh area and should be spaced evenly up to about 80 cm in both directions. They should be added to the top (longitudinal) layer of the bottom (bent) mesh. The bent part should be placed behind according to the production direction to reach the minimum TMX support length.



The geometry should be adjusted such that the distance holders support the top mesh wires with at least 5 cm of tolerance. The distance holders that miss the top mesh wires are not usable.

The distance holders can be designed in two ways. In the simpler **Case 1** both bends are in one span. The more complex **Case 2** can be used in denser meshes by placing each bending in a separate span. It requires the external height **H** of the distance holders to meet the minimum specified below.



The table summarizes the geometrical requirements for both cases.

Spacer	Mesh	Case 1		Case 2					
		A_{min}	S_{min}	H_{min}	A_{min}	$S_{1,min}$	$S_{1,max}$	$S_{2,min}$*	S_{min}
Ø8	Ø6	28	**H** + 88	122	44	121	**H** - 1	**H** + 13	**H** + 134
	Ø8	29	**H** + 90	124	45	123	**H** - 1	**H** + 12	**H** + 135
	Ø10	30	**H** + 92	126	46	125	**H** - 1	**H** + 11	**H** + 136
	Ø12	31	**H** + 94	128	47	127	**H** - 1	**H** + 10	**H** + 137
	Ø16	33	**H** + 98	132	49	131	**H** - 1	**H** + 8	**H** + 139
Ø10	Ø6	50	**H** + 111	153	50	132	**H** - 21	**H** - 21	**H** + 111
	Ø8	51	**H** + 113	155	51	134	**H** - 21	**H** - 21	**H** + 113
	Ø10	52	**H** + 115	157	52	136	**H** - 21	**H** - 21	**H** + 115
	Ø12	53	**H** + 117	159	53	138	**H** - 21	**H** - 21	**H** + 117
	Ø16	55	**H** + 121	163	55	142	**H** - 21	**H** - 21	**H** + 121
Ø12	Ø6	59	**H** + 121	171	59	146	**H** - 25	**H** - 25	**H** + 121
	Ø8	60	**H** + 123	173	60	148	**H** - 25	**H** - 25	**H** + 123
	Ø10	61	**H** + 125	175	61	150	**H** - 25	**H** - 25	**H** + 125
	Ø12	62	**H** + 127	177	62	152	**H** - 25	**H** - 25	**H** + 127
	Ø16	64	**H** + 131	181	64	156	**H** - 25	**H** - 25	**H** + 131

* When $S_1 > S_{1,min}$ the value of $S_{2,min}$ is reduced and can be calculated as $S_{2,min} = S_{min} - S_1$.

The mesh wires can be shifted further apart to fulfill the minimum spacings but <u>should not be cut</u>. Cutting the wires leaves short pieces behind, which do not contribute to the element capacity, but increase steel usage and production time. In such cases it is better to remove the bars completely.

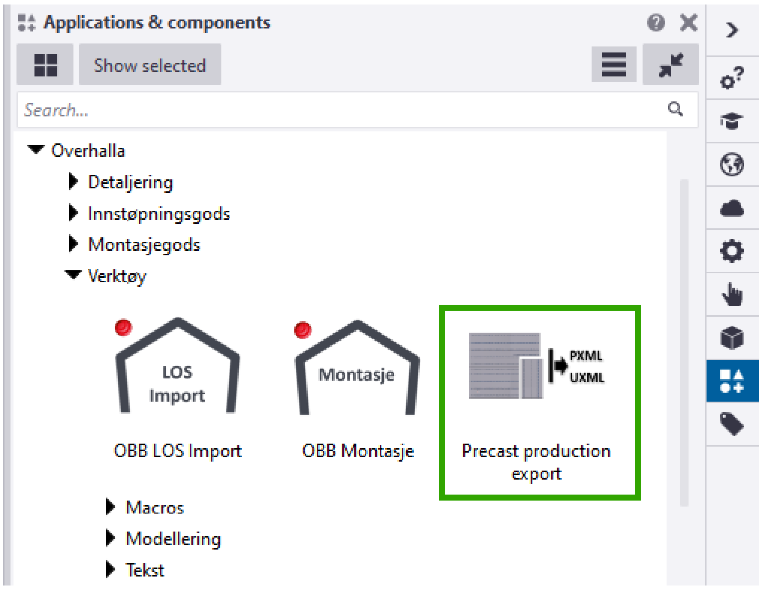
Export to PXML

Precast Production Export

The Precast Production Export extension is used to export the PXML files from Tekla. It is installed together with Overhalla Environment but can also be downloaded from Tekla Warehouse.

A detailed explanation of extension's functionalities can be found on Tekla User Assistance.

The export dialog can be opened from the Application & components catalog. It can be found under the `Verktøy` group of the `Overhalla` catalog or using the search box.



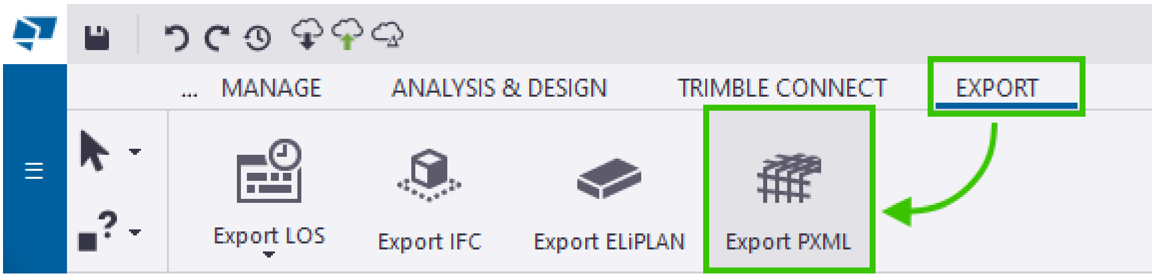
Starting the extension manually

The extension may sometimes not start properly when trying to open it from the component catalog. This is a bug in Tekla related to macros and was reported to Trimble.

Restarting Tekla fixes the problem, but the extension can also be started manually from:

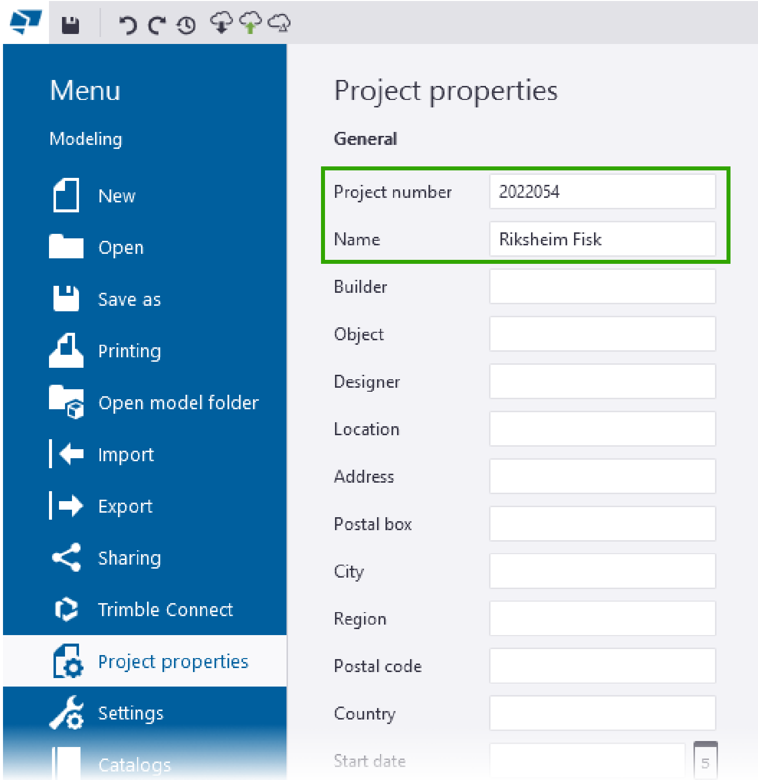
C:\TeklaStructures\2021.0\Environments\common\extensions\PrecastProductionExport

A shortcut for starting the export extension manually can be now found in the ribbon too.



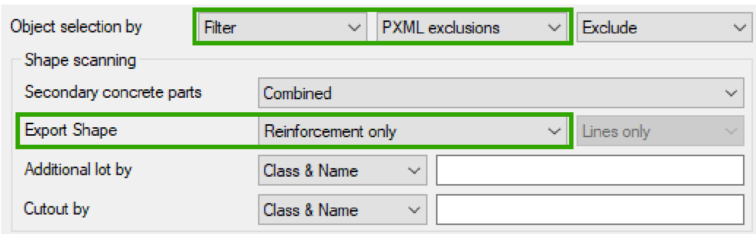
Project properties in the export

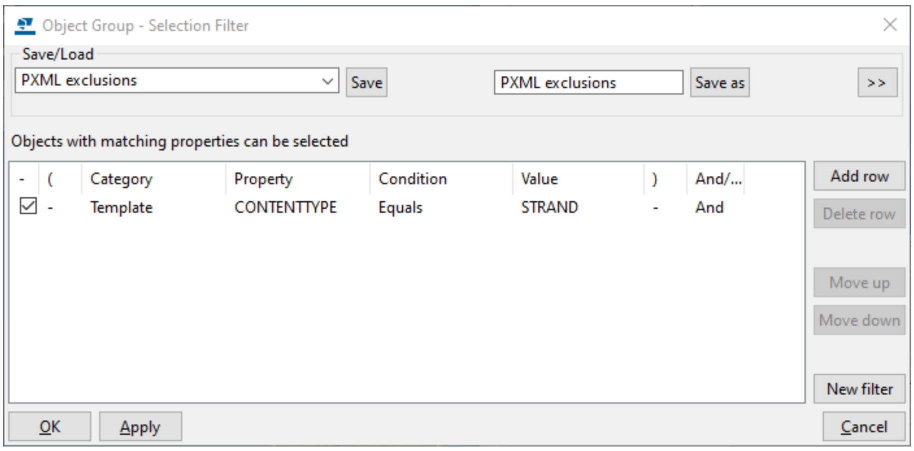
The project number and name are taken from the `Menu > Project properties` fields.



Only reinforcement is exported

The PXML can include complete element geometry, but our usage is limited to only reinforcement. In addition, strands are filtered away, so that only the rebars are exported. The filtering could also be used to, for example, exclude pre-produced lifting reinforcement.

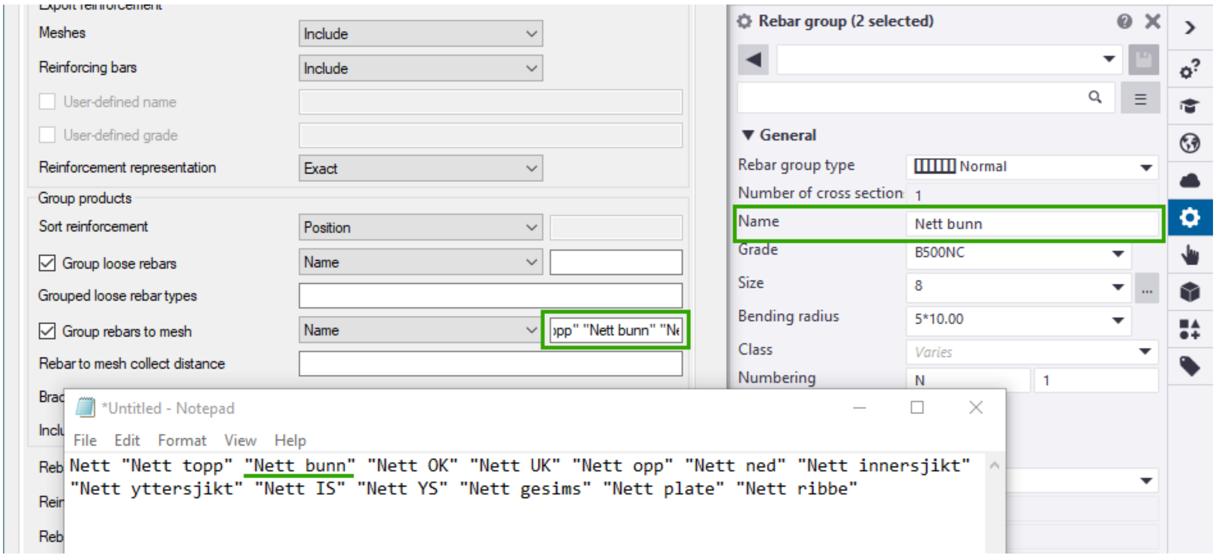




Grouping rebars into meshes

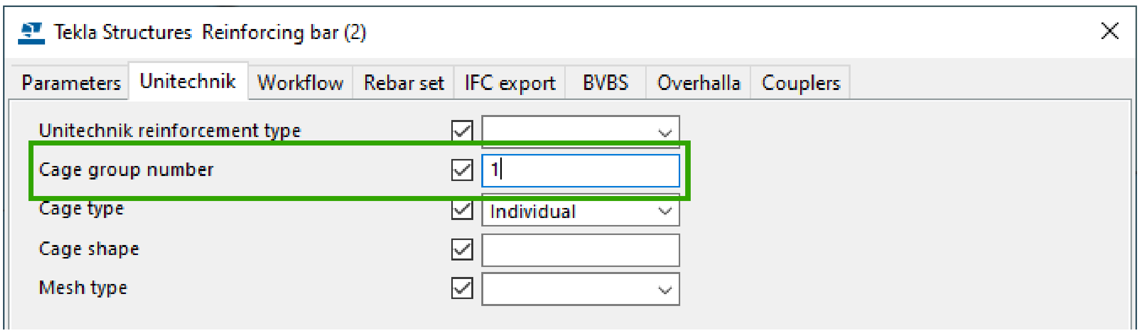
Rebars are grouped into mesh based on their name. All bars with the same name will be grouped together and unique names will become different groups. All names must be specified in the `Group rebars to mesh` field in the `Reinforcements` tab of export settings. All names should start with a word `Nett`, as this is used as a rule for grouping in the drawings.

The field is quite narrow, so values can be copied to notepad for easier editing and then copied back. Names with spaces must be quoted. Export settings can then be saved per type of element.



Grouping rebars into cages

Rebars are grouped into cage based on the value of `Cage group number` UDA found in the `Unitechnik` tab of the User-defined attributes dialog. When the field is empty, grouping into cage will not be used. Grouping into cages takes precedence over grouping into mesh, so ensure that the field is empty for bars that should be grouped into a mesh.

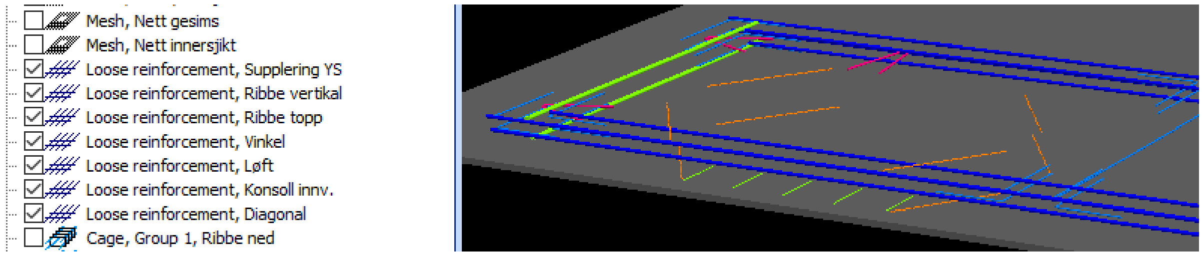


Apart from the `Cage group number` all rebars grouped into the same cage should also have identical name. Otherwise, the cages may be exported incorrectly.

Grouping of loose rebars

All rebars that are not explicitly grouped into either a mesh or a cage, will be exported as loose reinforcement. Loose rebars are grouped together by name, but the names do not need to be specified in the export settings. The names of loose reinforcement groups are visible in AviCAD, profit and machine, and can be referred to in the drawing.

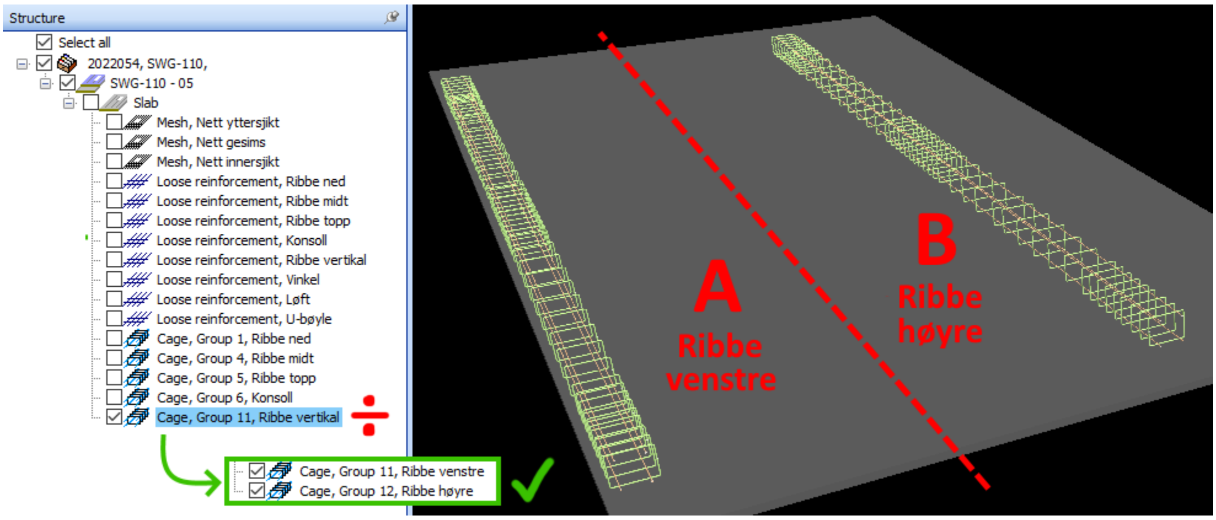
There is no strict requirement to how the loose reinforcement should be grouped. Nevertheless, using good and descriptive names can be very helpful. It simplifies checking the reinforcement in AviCAD, makes referring to correct reinforcement in the drawings easier, allows production planners to better control what is produced and when, and finally gives machine operators and reinforcement workers better overview.



Distinct mats should be separated

Distinct reinforcement parts, that will be produced separately, should also be grouped separately even if they are identical. Every individual mesh should therefore have a unique name and every individual cage a unique number.

The loose reinforcement is produced piece-by-piece and here the grouping should instead focus on the intended use of the grouped rebars, allowing production at the time when the bars are needed.

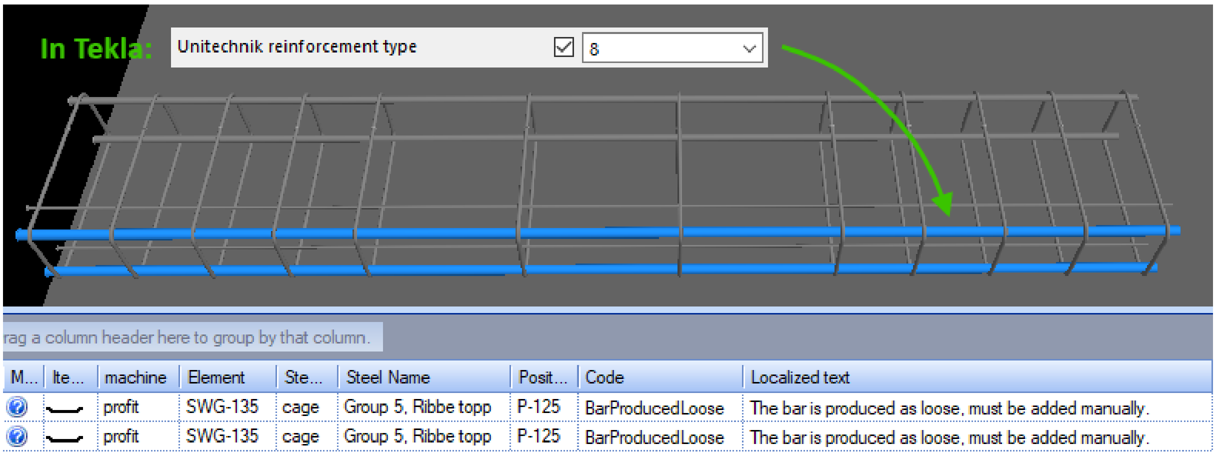


Producing some rebars as loose

When bars cannot be included directly in a mesh or a cage, for example because of the diameter too big to produce with BlueMesh, it is a normal practice to export such bars as loose reinforcement and describe in the drawing to which mesh, or cage should they be supplemented.

When there are many such parts, grouping the bars with the mesh or cage but marking them to be produced as loose can improve readability in both the drawings and the exported PXML files.

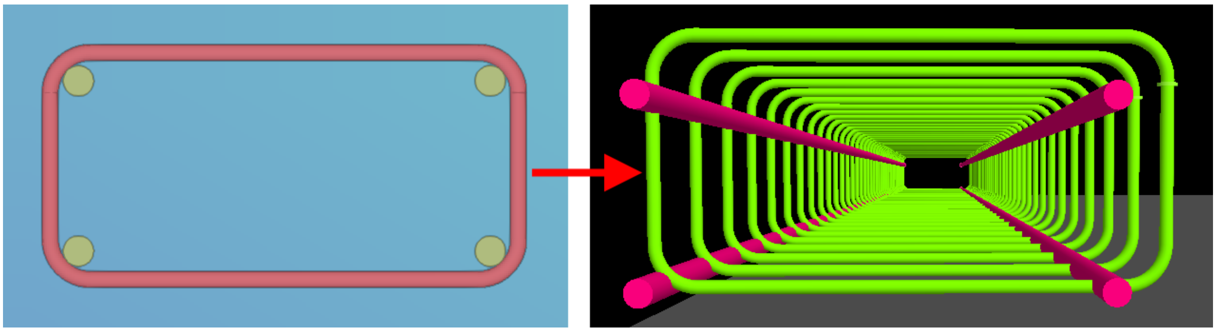
This can be done by assigning the bars which should be produced as loose the reinforcement type of `8 = loose bars`. The bars will be highlighted in blue in the PTS results for visual overview.



A whole mesh can also be marked as loose by setting the mesh type to `8 = loose mesh`. Then the whole mesh is highlighted in blue. In practice such function is rarely used, and it is not possible to do for cages, where the mesh type is not exported from Tekla.

Misplaced bars in exported cages

Sometimes the placement of bars in exported cages may be off when compared to Tekla:



When such problems occur in the export, double check that all following conditions are fulfilled:

1. Names of all rebars grouped into a single cage must be the same.
2. The `Cage type` in UDA dialog must be the same for all rebars in a cage.
   Which option is chosen does not matter, but it must be the same for all bars.
3. `Group rebars to mesh` must be checked in the export dialog.
   This option can be found in the `Reinforcement` tab and is checked by default.

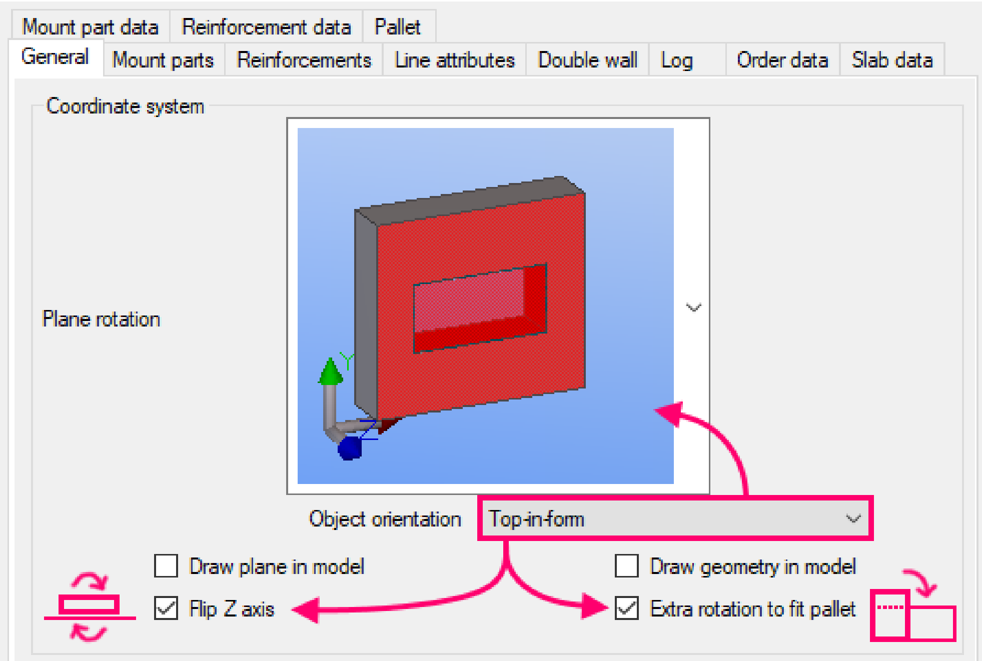
The different names of grouped rebars is the most common cause.

Element orientation on pallet

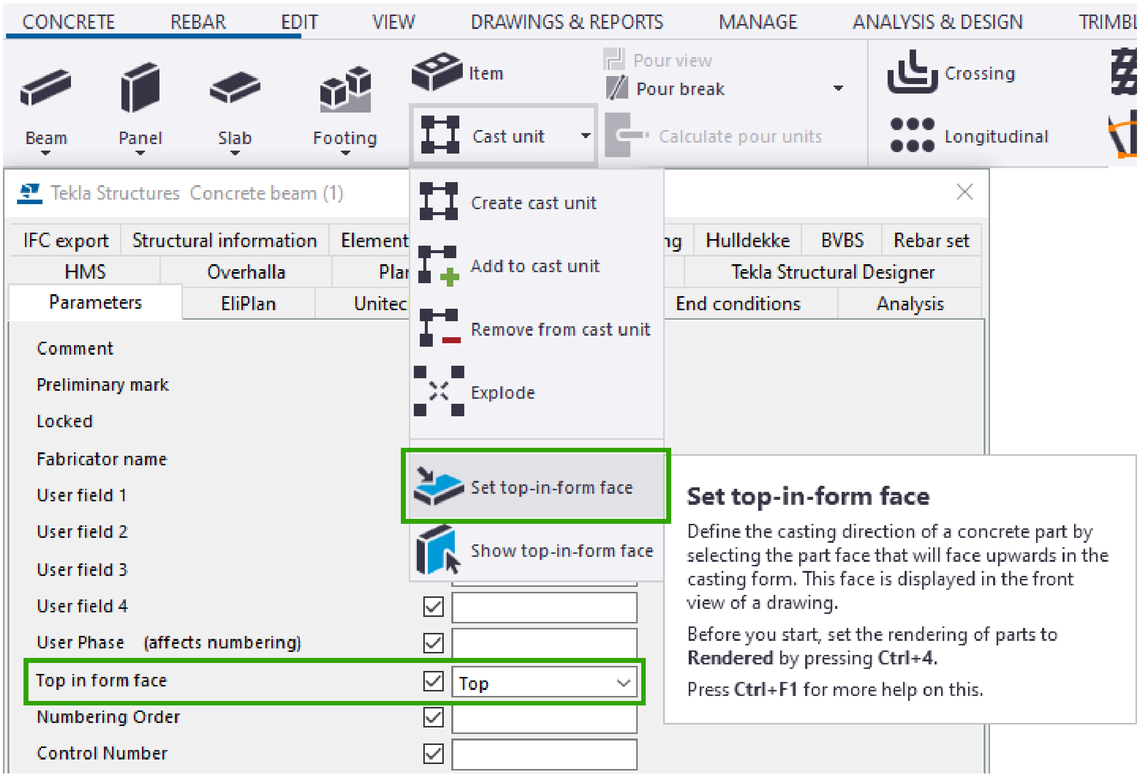
There are two modes for orienting the element on pallet that can be specified in the `Object orientation` option in the `General` tab of the export settings.

By default, the elements are oriented `Lengthwise`. This is an automated mode that will orient the element by comparing its dimensions with the pallet. The longest dimension will be along the pallet length, second longest along the pallet width and the smallest dimension will be vertical. When top-in-form face is defined, it will then decide which side of the element should point upwards.

The alternative `Top-in-form` mode can be used for better control of the orientation on pallet. When this option is selected, the export will try to orient the element as in the drawing, by using either the top-in-form face if it is set, or modelling direction otherwise. Now the `Plane rotation`, `Flip Z axis` and `Extra rotation to fit pallet` options can be used to adjust the final orientation of the exported element. Check the `Draw plane in model` option to preview the orientation without having to open the exported PXML in AviCAD.



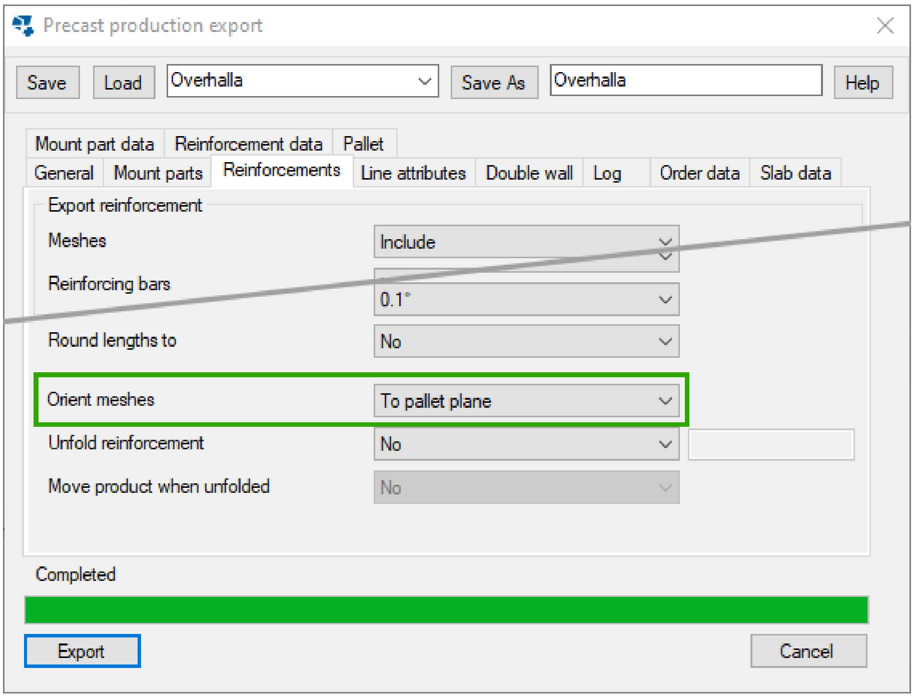
The top-in-form face can be either set visually using the `Set top-in-form face` function or selected from a dropdown in the User-defined attributes dialog.



Automatic orientation of meshes

Orientation of mesh on pallet is important and determines how the mesh will be produced. Incorrect orientation may sometimes lead to errors. The machine software will attempt automatic adjustment, but sometimes it may be necessary to change orientation manually in AviCAD.

We can also instruct the export to try to guess the best rotation for individual meshes by setting the `Orient meshes` option to `To pallet plane` in the Reinforcements tab of the export settings. Then the element will most often not look right in AviCAD, but this is not an issue.

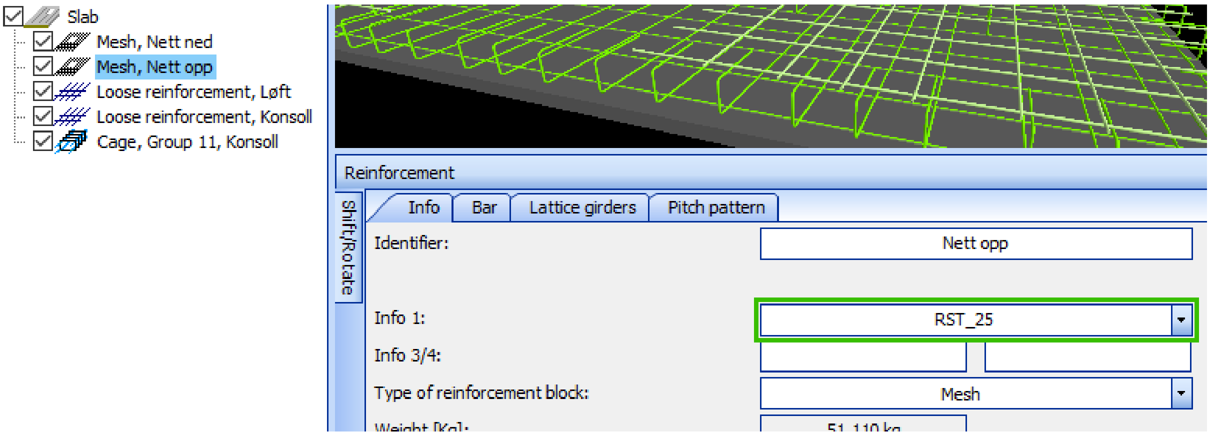


Controlling production with grid

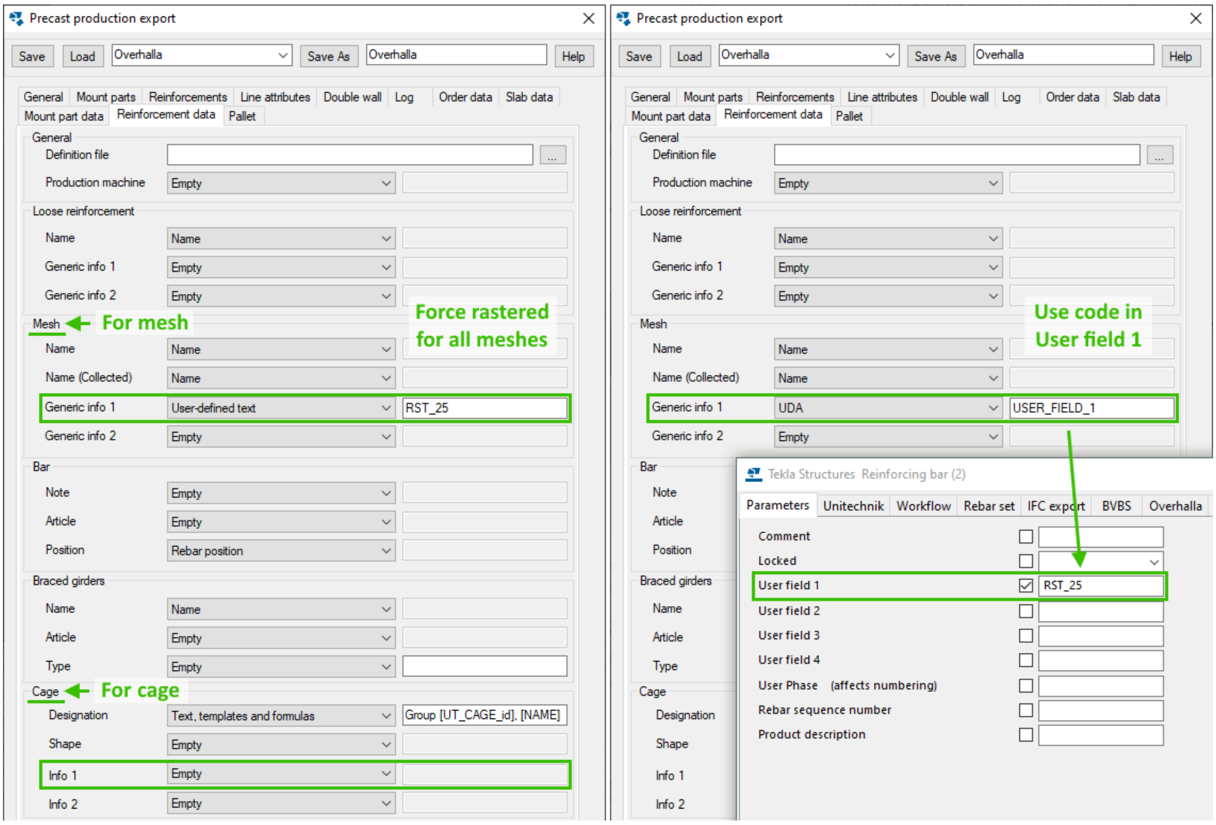
Note: Because this often led to issues, now the choice between the rastered and gridless modes is fully automatic and happens upon import of PXML to profit. All flat and bent meshes and cages without corner bars are assigned the `RST_25` flag by default, because there the small adjustments of bar positions do not matter. For cages with corner bars the flag is always cleared to use the gridless production whenever possible and avoid shifting the corner bars, which would disrupt the cage dimensions after the bending. It is therefore no longer necessary to set these flags manually.

The production with grid can be controlled by typing a special code into the `Info 1` field in the reinforcement data for a mesh or a cage:

- `RST_25` to force the rastered production even if the mesh is possible to produce gridless. Wires outside of grid will be automatically shifted into the grid.
- `RST_1` to prevent automatic switching to production with grid when gridless is not possible. In such case PTS will give `NotGridlesslyProductable` error.

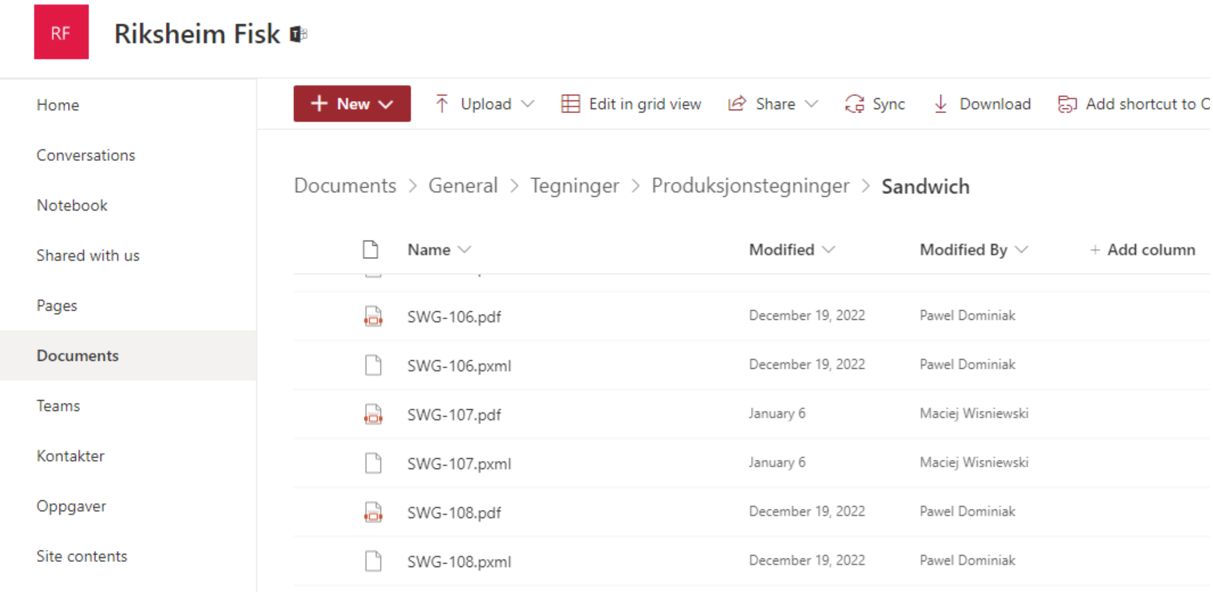


The codes can be set already during the export from Tekla, for example for all exported meshes, or for a specific mesh or cage using the value of user field from the reinforcing bars. To do so, modify the value of `Generic info 1` field under `Reinforcement data` tab of the Precast Production Export. Note that the value is defined separately for the `Mesh` and `Cage` types.



Publishing and revising PXMLs

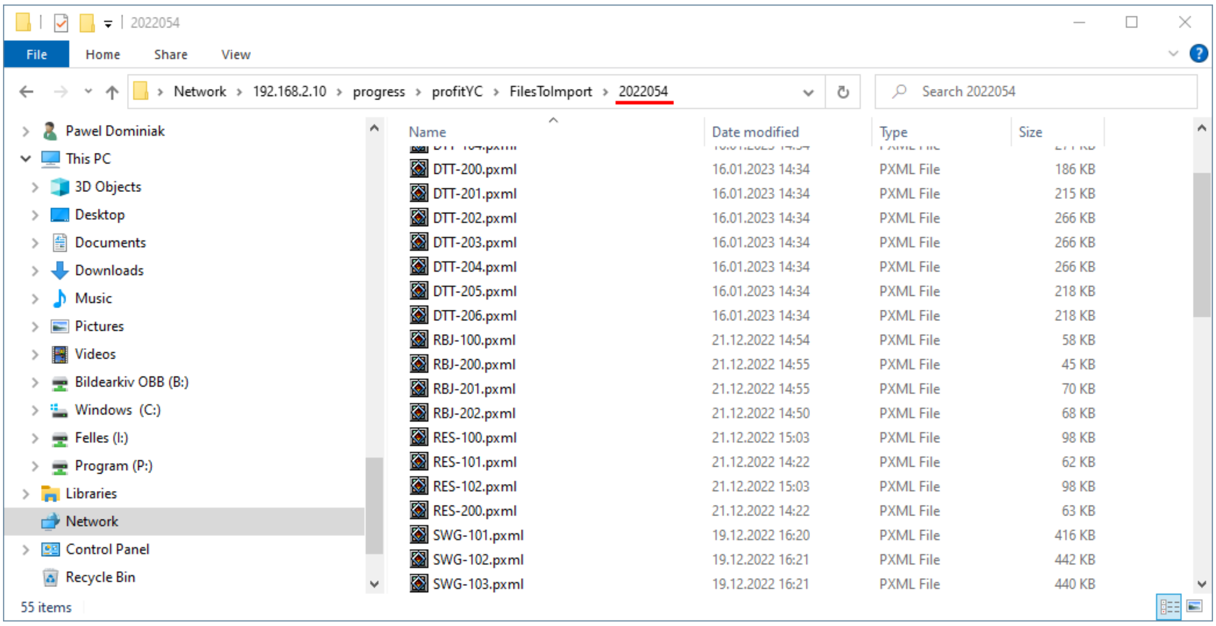
PXML files that are completed, checked and ready to be produced should be stored together with the PDF drawings in a correct element subdirectory of `Tegninger > Produksjonstegninger` folder in the project site on SharePoint. The filename should be the same as the drawing name.



In case of revisions, the existing PXML file should be updated with the new version. The name of the file should be exactly the same as before, and it must not include the revision number.

If a revision is published within 10 days of a production date, an email with a list of changed elements should also be sent to revisjonsvarsling@overhallabetongbygg.onmicrosoft.com.

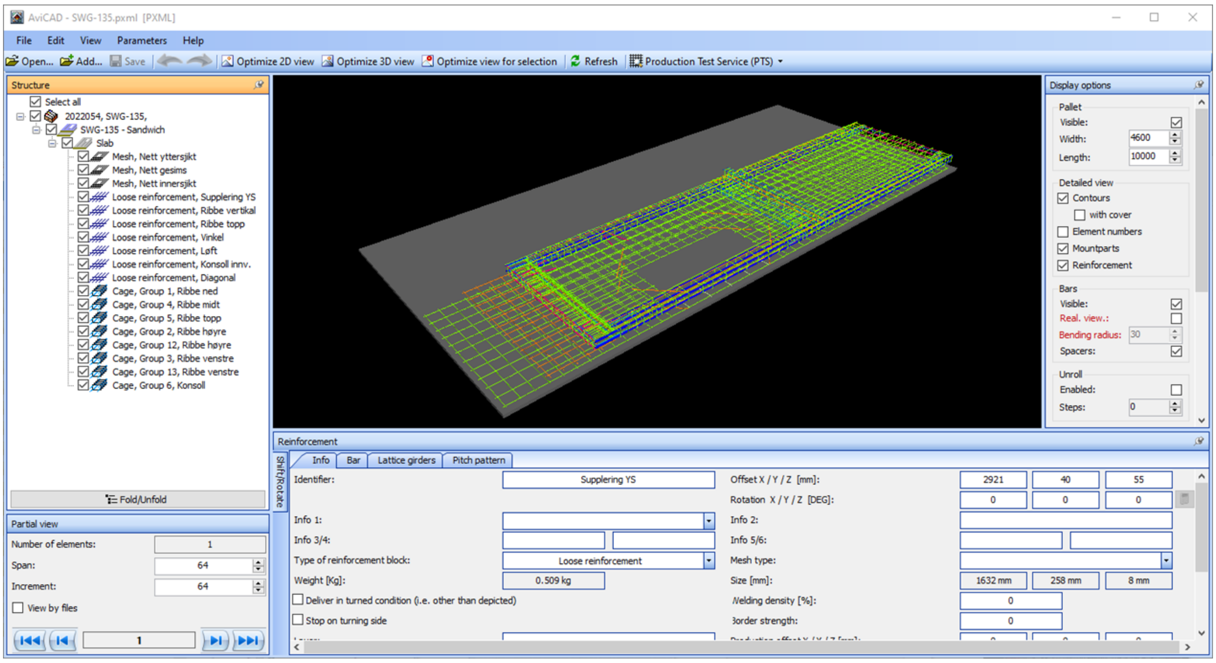
The PXML files are automatically mirrored from SharePoint to the network folder every 10 minutes. The folder location is \\192.168.2.10\progress\profitYC\FilesToImport, and the PXMLs are organized by the project number. Profit reads the reinforcement files from this directory.



AviCAD basics

Viewing and modifying PXML files

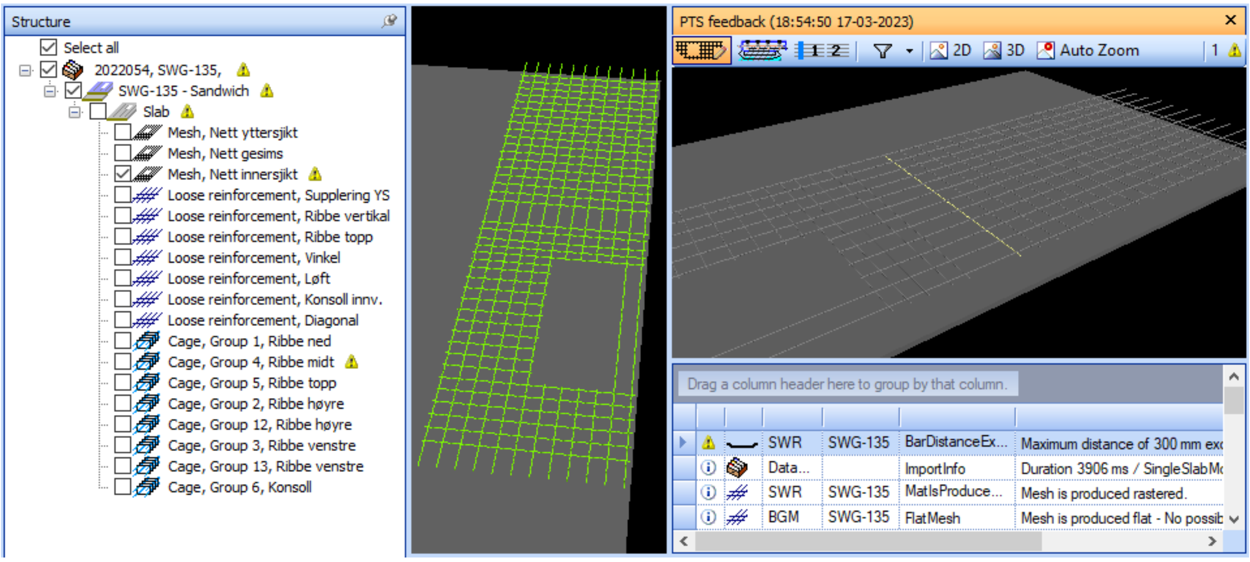
AviCAD is a program that can be used to view, check, and modify the exported PXML files.



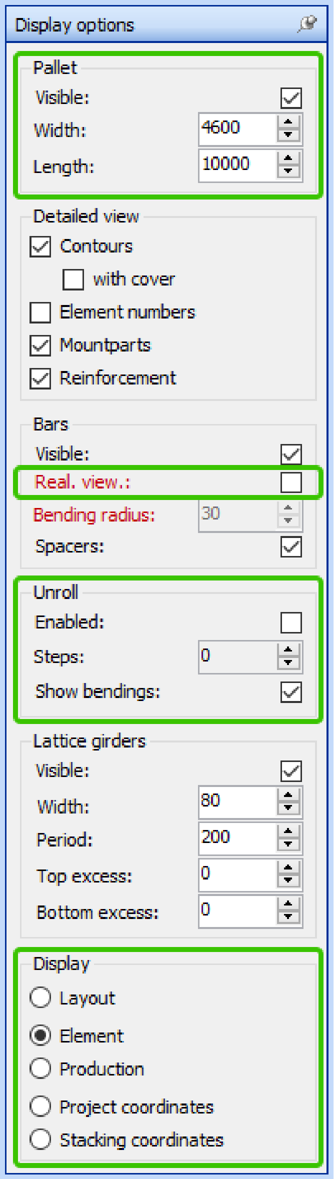
The newest version of AviCAD with Overhalla configuration can be downloaded here.

Filtering reinforcement groups

The Structure pane can be used to filter the displayed reinforcements and the PTS results by checking only the items/reinforcements that should be displayed.



Display options side pane



Pallet dimensions are used for visualization only. Changes do not affect the production. The **pallet can be hidden** from view by unchecking the **Visible** option.

Real view of bars can be enabled to display the overlapping **bendings side by side**.

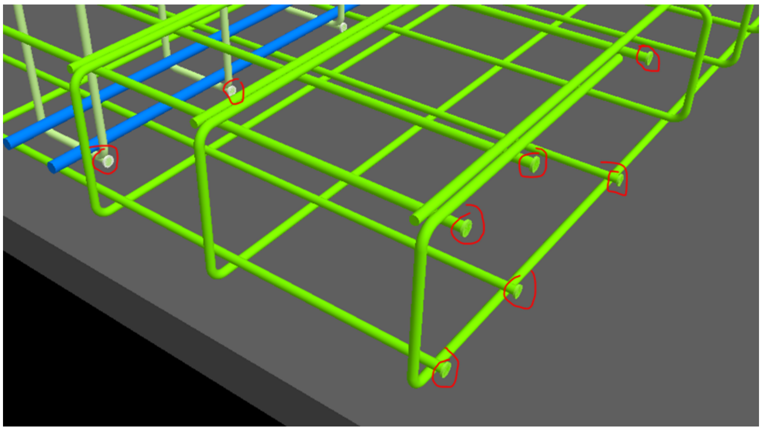
Unrolling of reinforcement. Check the **Enabled** option to unroll completely or use the **up** and **down** arrows to **step through the bending** procedure.

In PTS received data each bending action is in a separate step.

Display view can be switched to make modifications like rotation or shifting position **apply in production** but not change the element view.

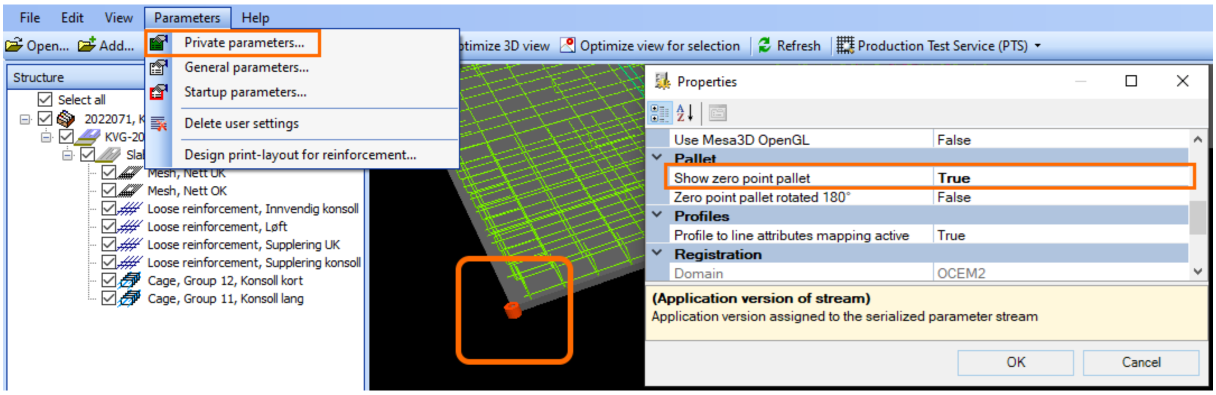
Markers for bar start points

Markers of bar start points visible in AviCAD as “heads” are just visual and do not affect production.

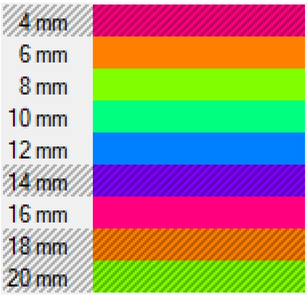


Showing zero-point of the pallet

The zero-point of the pallet can be displayed in AviCAD to show where the production starts. The program must be restarted after making this change.



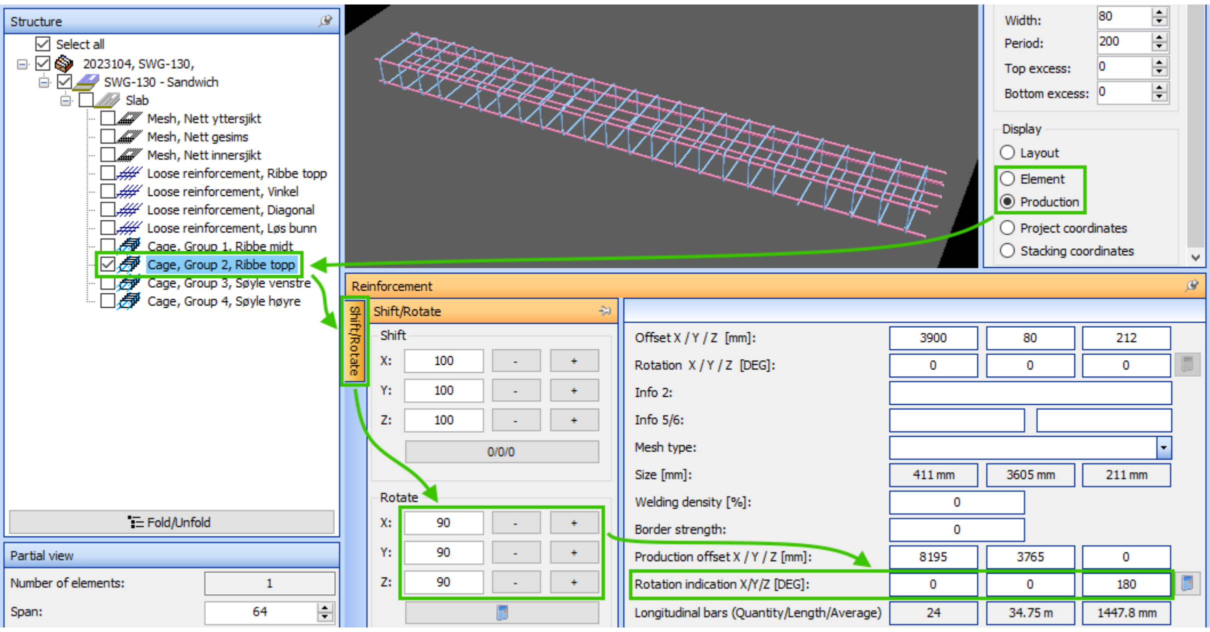
Wire colors by diameter



The colors in AviCAD indicate the wire diameters. An overview of all the colors can be opened from `View > Show legend of colors` and is presented to the left. The diameters that are not available for our machine have been greyed out.

Changing orientation of reinforcement

Many of the errors caused by invalid orientation are now automatically fixed by profit or the machine software, but in special cases it may be necessary to manually rotate a specific mesh or cage.



A recording that shows how to rotate meshes in AviCAD can be viewed here. The steps:

1. Switch to the `Production` view in the `Display options`.
2. In the `Structure` pane select the steel to rotate.
3. In the `Reinforcement` pane open the `Shift/Rotate` dialog.
4. Rotate around `X`, `Y` or `Z` axis by clicking the `-` or `+` buttons.
5. Optionally shift in `X`, `Y` or `Z` axis by clicking the `-` or `+` buttons.
6. Switch back to the `Element` view in the `Display options`.

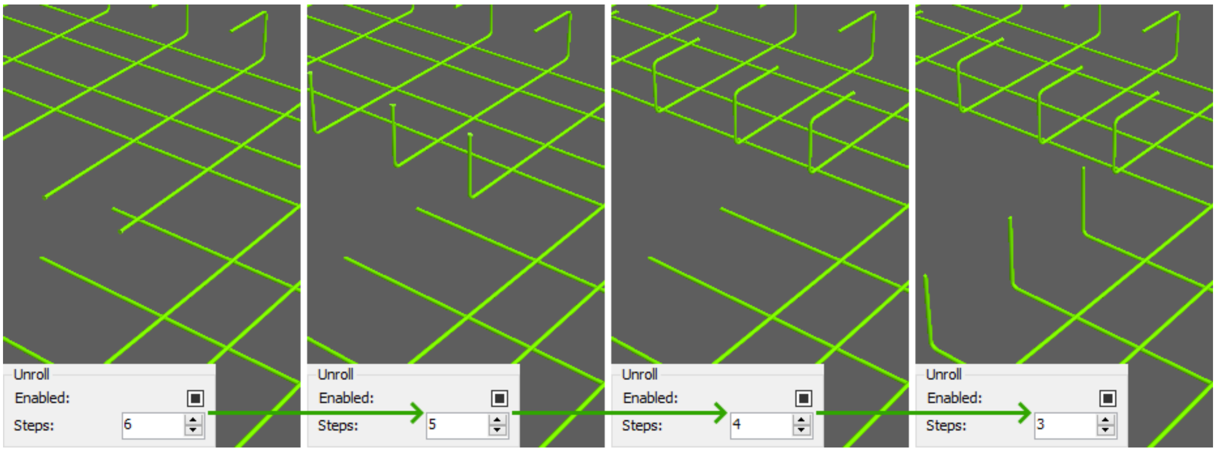
Using the `Production` view is necessary, because only then profit will not apply the same automatic rotation which may have led to errors. Additionally, this will let us not change how the element looks in the `Element` view. At least one of the `Rotation indication` fields must be non-zero. To lock the current orientation, rotate 360 degrees in any axis. The rotations can also be entered manually.

Unrolling of reinforcement

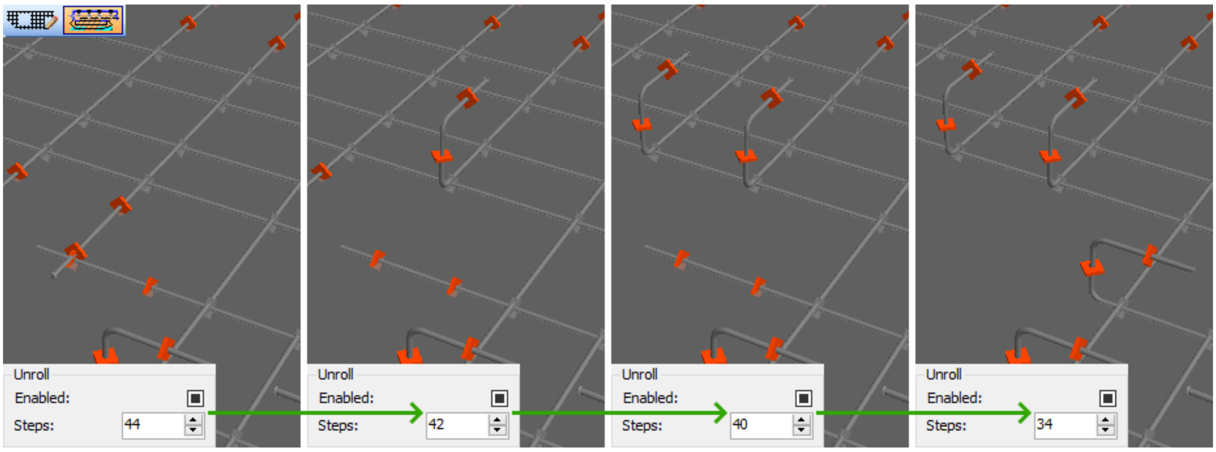
AviCAD offers an unrolling feature that can be accessed in the side pane. The mesh welding machine welds the straight wires before making the bendings, so it must first unroll or flatten every mesh. Then the optimal procedure for making the bendings is determined and every bending of every bent bar is checked for whether it can be produced without problems.

The unrolling feature can be used to check if the mesh unrolls correctly, to go through the bending process step-by-step and investigate reason for why certain bendings cannot be made, particularly the intermediate ones. Mesh can be unrolled completely by marking the `Enabled` checkbox or step-by-step using the arrow buttons or up and down keyboard keys once the field is focused.

Incorrect reinforcement types will often lead to problems with unrolling. The basic algorithm in AviCAD unrolls reinforcement edge by edge. Its results may sometimes differ from the actual bending order determined by the machine.



The actual unrolling steps can be seen in the PTS results window by enabling the PTS received data. It contains the actual order of bendings sent back from the machine. Here every bending of every bar is assigned its own step, just as it will be done during the production. Note that the invalid bendings are skipped by the machine and therefore will not be shown correctly.



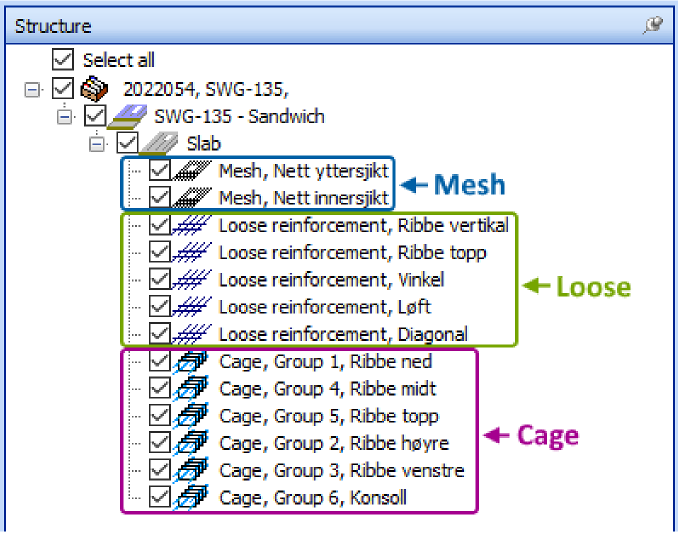
Checking reinforcement

Checking steel, mesh, and bar types

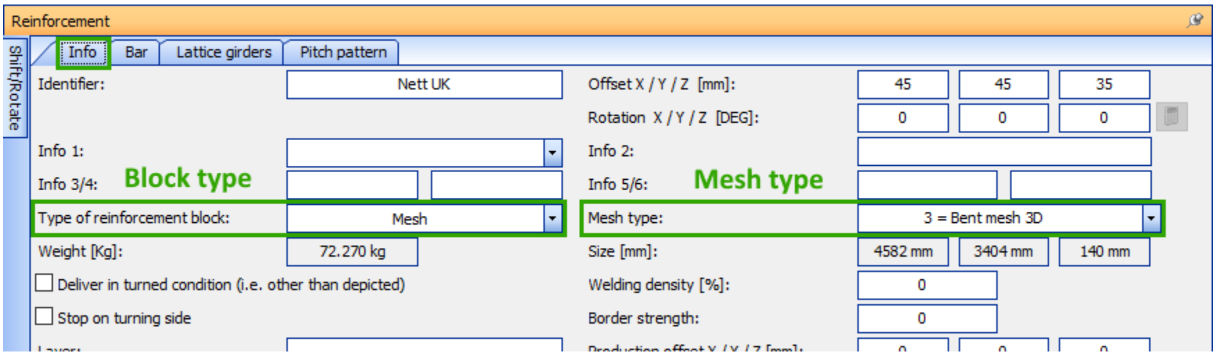
Grouping into reinforcement blocks (or steels) is visible in the `Structure` pane. The checkboxes in the left side can be used to show or hide reinforcement groups or elements from the view.

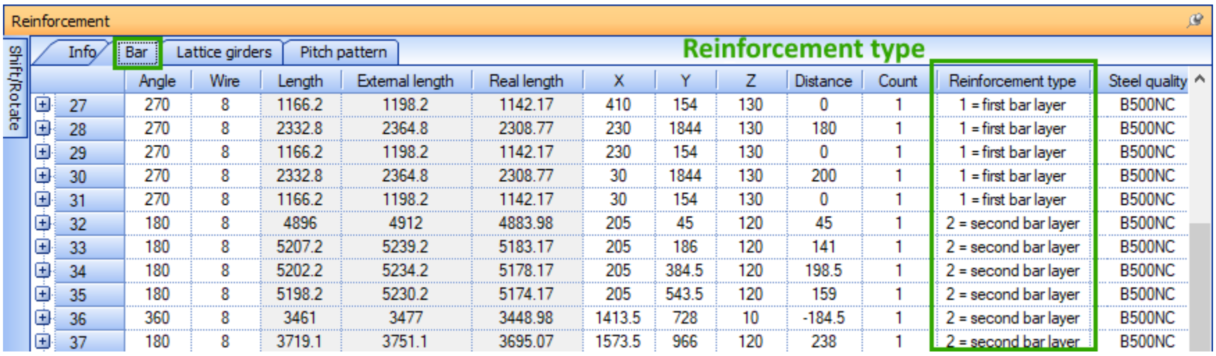
The following points should be checked for each reinforcement group:

1. If the type of the group is correct (mesh, cage, or loose).
2. If the group contains correct rebars and no rebars are missing.
3. If the block and mesh types are correct. Mesh type will be empty for cages.
4. If the reinforcement types are correct. Errors can often be spotted by unrolling.



After selecting a group, the block and mesh types can be checked in the `Reinforcement` pane, in the `Info` tab. The reinforcement type for all bars can be found in the `Bar` tab.





	Angle	Wire	Length	External length	Real length	X	Y	Z	Distance	Count	Reinforcement type	Steel quality
27	270	8	1166.2	1198.2	1142.17	410	154	130	0	1	1 = first bar layer	B500NC
28	270	8	2332.8	2364.8	2308.77	230	1844	130	180	1	1 = first bar layer	B500NC
29	270	8	1166.2	1198.2	1142.17	230	154	130	0	1	1 = first bar layer	B500NC
30	270	8	2332.8	2364.8	2308.77	30	1844	130	200	1	1 = first bar layer	B500NC
31	270	8	1166.2	1198.2	1142.17	30	154	130	0	1	1 = first bar layer	B500NC
32	180	8	4896	4912	4883.98	205	45	120	45	1	2 = second bar layer	B500NC
33	180	8	5207.2	5239.2	5183.17	205	186	120	141	1	2 = second bar layer	B500NC
34	180	8	5202.2	5234.2	5178.17	205	384.5	120	198.5	1	2 = second bar layer	B500NC
35	180	8	5198.2	5230.2	5174.17	205	543.5	120	159	1	2 = second bar layer	B500NC
36	360	8	3461	3477	3448.98	1413.5	728	10	-184.5	1	2 = second bar layer	B500NC
37	180	8	3719.1	3751.1	3695.07	1573.5	966	120	238	1	2 = second bar layer	B500NC

Production Test Service

The Production Test Service (PTS) is a function in AviCAD that checks if the reinforcement was modelled correctly and if it is producible. PTS sends a request to the server located at our factory, which uses the same software and configuration as the actual machine in order to simulate production of the reinforcement and detect possible problems, just as the machine would.

The PTS checks each individual mesh, cage, or loose reinforcement, but does not check collisions between groups or with inserts etc. Whether and how complete reinforcement can be assembled, must be checked separately by the designer, either in Tekla or in AviCAD.

When reinforcement is sent to be checked, it first goes through *profit*, which is a program we use for planning the production of reinforcement. Both profit and the machine software can perform some automatic transformations of reinforcement to fix common errors and allow production. This can be rotation or turning of the whole group, shifting wires, or adjusting the bending radius.

The PTS check results can include 4 types of messages:

1. **Information messages** give feedback about mode of production (flat, bent, gridless rastered etc.) and performed transformations (turning mat, shifting bars etc.). Normally they do not require any action but can help with checking if reinforcement will be produced as expected.

		SWR	KVG-203	MatIsProducedTurned	Mesh will be produced turned.

2. **Hint messages** are similar to information messages but are used to avoid mistakes when the production mode is non-standard. The affected reinforcement is highlighted in blue.

		profit	KVG-104	StorageItem	Mesh is a storage item - will not be produced

3. **Warning messages** mean that the mesh is producible, but often not exactly as designed (for example some bendings cannot be done), or that the production will require special attention from the operator. Designer may choose to ignore the warnings.

		SWR	FDB-116	NoValidLabelPosFound	No valid label position found.

4. **Error messages** indicate serious problems because of which the mesh cannot be produced. Errors must always be fixed before the reinforcement is sent to production.

		SWR	KVG-203	MatTooShort	Mesh too short! Minimum mesh length: 1000 mm

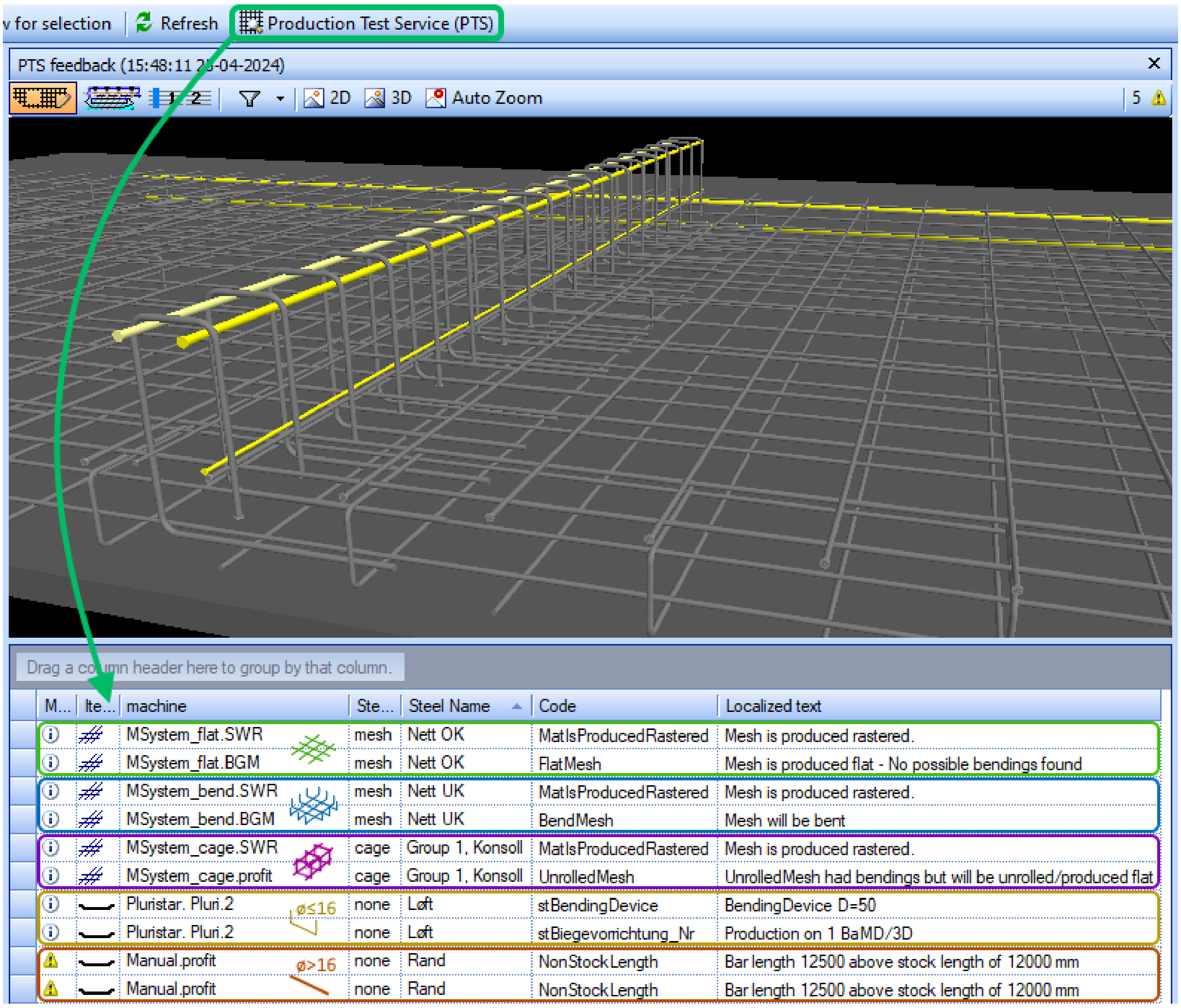
Automatic machine selection in PTS

The reinforcement can be produced by two automated machines: M-System BlueMesh for welding of meshes, and Pluristar for bending of loose bars. Because each type of reinforcement (like flat or bent mesh) has special requirements, there are more production modes or “virtual machines”:

- **MSystem flat**: Mesh without bendings or where bendings should be unrolled.
- **MSystem bend**: Mesh with bendings producible with single bender (one by one).
- **MSystem cage**: Bent mesh that is produced flat and then bent on mesh bender.
- **Pluristar**: Loose rebars up to diameter of 16 mm and up to length of 12 m.
- **Manual**: Any other loose reinforcement.

The selection of applicable production mode is done when the reinforcement is imported to profit, and the same rules apply also for PTS. More details can be found in reinforcement flows.

Running the PTS in AviCAD is currently fully automatic: the program selects a correct mode for each reinforcement, runs the tests in parallel, and then combines the results from all machines.



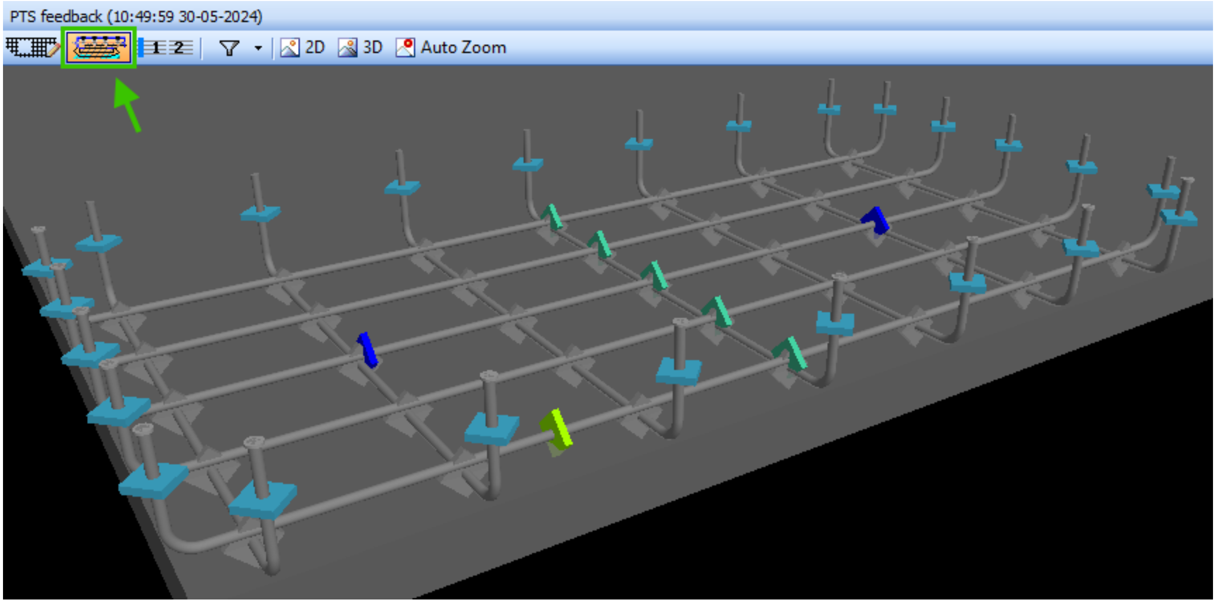
<u>The results for meshes and cages must always be checked in detail</u>. For loose reinforcement only a quick glance whether there are no errors is normally enough, most warnings can be ignored.

Viewing the PTS received data

AviCAD is used by multiple companies with different mesh welding machines. The basic algorithm for unrolling of reinforcement in AviCAD is generic and may differ from the actual order in the machine. This generic algorithm groups bendings in multiple bars into stages showing them in a single step.

The PTS feedback, apart from the check results, contains also additional information added by profit and the machine software. The feedback includes the rotation and shifting of the mesh, adjustments of bar positions, corrected bending diameters and welding points with various details including very detailed bending order, bar by bar. Each bending step contains only a single bending.

Too see all the additional information, switch display in PTS feedback to `PTS received data`.



Welding points in the PTS received data

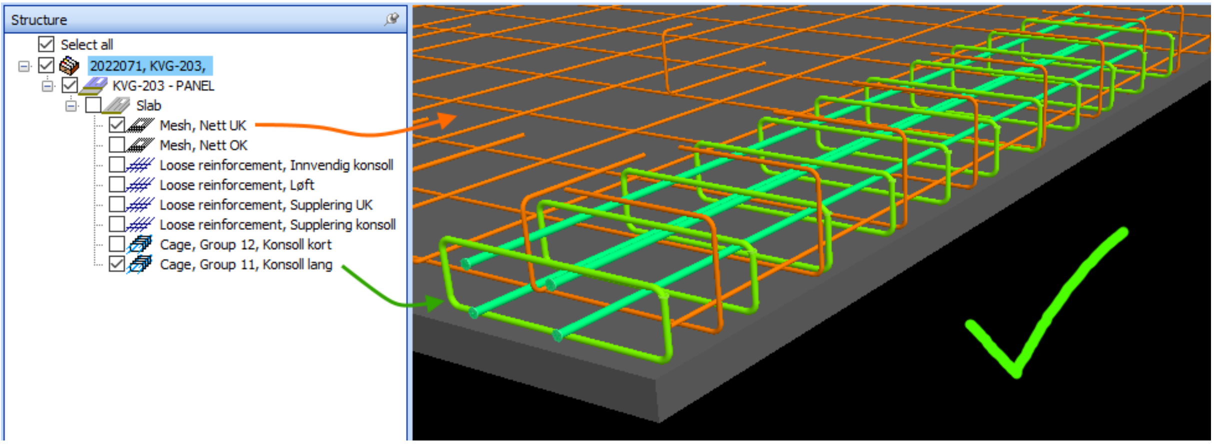
The PTS server adds welding point nodes in its response which can be viewed when the PTS received data is enabled in the PTS results window, see above. These nodes store additional details about the bars, crossing points and bent segments. Each welding point has a type indicated by a number and is shown in a different color in the PTS feedback pane.

The overview of the welding point types, colors and functions is shown in the table below.

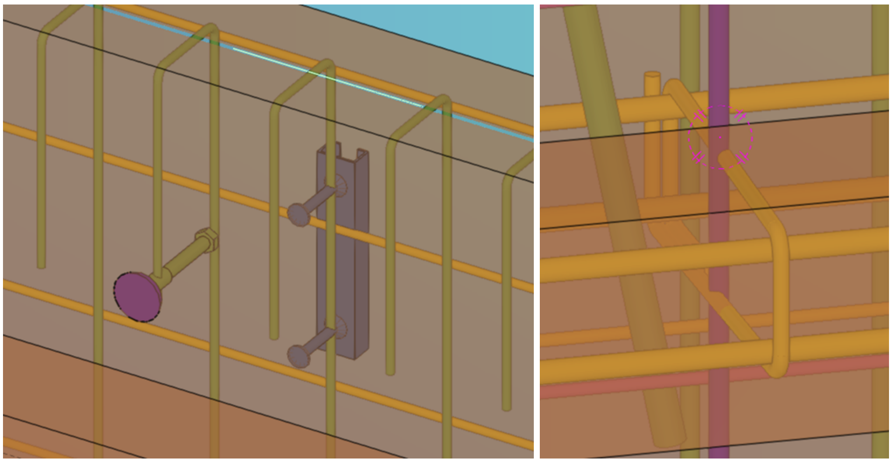
Type	Node	Description
1	Welded point	Contains welding parameters for the node
0	Undefined	Node without a defined type
-1	Not welded point	Marks crossing point not to be welded
-2	Label position	Marks position where the mesh label is placed
-3	Color point	May be used to specify the color if multiple colors are available
-17	Main segment	Defines the central and stable segment during production
-23	Bending step	Contains sequence of bendings determined by the machine
-29	Guided point	The guided bar follows a guiding bar when it is moved or bent
-30	Guiding point	The guiding bar when moved or bent is followed by guided bars
-100	Gripping position	Marks position where a gripper grips the mesh

Collisions with other groups and inserts

The machine software and PTS checks all reinforcement groups on one-by-one basis. Each group is tested separately from other groups because the production of every group is independent as well.



The software does not detect collisions between different reinforcement groups or between the reinforcement and, for example, inserts. This kind of issues must be checked by the designer visually. This should preferably be done already when modelling in Tekla, because reinforcement placement in AviCAD may not be the same as in the actual element depending on the export settings.

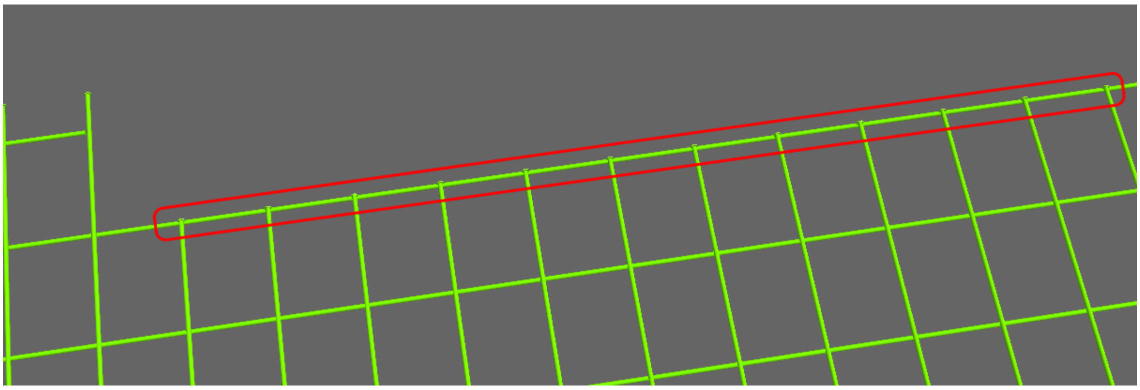


Omitted welding points and other issues

Not all issues with the reinforcement can be detected by the PTS.

For example, in the mesh below the topmost wire would not be welded to most of the bars because of too little overhang. The PTS does not show any errors or warnings, because the requirement of minimum 2 welding points is fulfilled. The PTS cannot know if this was done on purpose, in general the mesh can be produced as designed, and thus there are no warnings.

This is however not ideal, and probably not wanted by the designer. The topmost wire can break or bend when the mesh is handled or transported. Therefore, keeping in mind the requirements and a certain degree of visual control is always necessary.



Understanding PTS results

Overview and severity of PTS results

Multiple companies use PTS with different machine types, but many messages are now adjusted to our specific setup. The designers, production planners and machine operators must understand the reasons and consequences of various PTS results and know which messages can be ignored.

The message of High severity must always be fixed, as they mean that either mesh is not producible, or that it will differ significantly from the design, for example because bendings cannot be made.

Message	Explanation
NotImplementedSteelType	Selected mesh type is incorrect
PTS TIMEOUT	Internal error or file too big
PTS_Faulted	Internal error in PTS, please report
MatInconsistent	Problem with mesh geometry
MatTooLong	Mesh is too long to be produced
MatTooShort	Mesh is too short to be produced
NoTransportPositionsFound	Mesh cannot be moved through bending portal
OutsideOccupationFrame	Mesh is too wide to be produced
TooLessMatWidth	Mesh is too narrow to be produced
BarNotLyingCorrectOnFeedingFinger	Multiple short bars in a line can fall from feeder
BarTooLong	Bar is too long to be produced
BarTooShort	Bar is too short to be produced
InvalidBarDiameter	Bar size is not available
OverMaxStickOut	Bar has too long overhang
ReinforcementDegreeInvalid	Bar is not perpendicular to other bars
SamePosition	Bar overlaps with another
stWireNotfound	Bar size is not available
stzBarTooshort	Bar is too short to be produced
TooLessCrossPoints	Bar has too few weld points
UnderMinStickOut	Bar has too short overhang
UnderMinStickOutOnCrossingBar	Bar causes too short overhang in crossing bars
BendBlockedByBar	Another bar blocks the bending
BendingBlockedByAnother	Blocked by another bar which was not bent
NegativeBendingNotAllowed	Bending must not point downwards
NoBendingDeviceFound	There is no die which could make the bending
→ _BendingSetNotReachable	Bending cannot be reached by the robot
→ _CollisionBetweenBendings	Bending die would collide when bending
→ _NoBoltQuotaFound	Bent segment is too short
→ _NoThreadInOutFound	Not enough space to insert bending die
SegmentTooShort	Segment too short to bend with Hambi

The messages of Medium severity should always be checked because they may cause problems during production and require special attention from the operator. When left unfixed, they do not cause significant changes in how mesh is produced, so can be ignored after a consideration.

Message	Explanation
BendMeshDimensionExceeded_Height	Too high mesh may crash with machine parts
MayNotStableStackable	May get stuck or require frequent offloading
NonStandardSpacing	Stirrup positions should match bending cams
NoValidLabelPosFound	Production pauses to attach label manually
OnlyOneTransportPositionFound	One gripper is not enough for bigger meshes
ProducibleAsMesh	Check if reinforcement was exported correctly
Productable180Degree	Try rotating the mesh 180 degrees
WeldingPointDistanceExceeded	Distance between bars or welds may be too big
BarMovedCollisionAZ	Bar moved to avoid collision with claws
BarTranslatetInXDir	Bar moved to reach minimum spacing
MovedToRaster	Bar moved into grid comb
NonStockLength	Length not in stock but possible to order
Y_UnderMinDistance	Forces rastered production
stDrehwinkel	Bending must be perpendicular to bar plane
stWinLimited	Bending angle reduced to be producible
stzLenSetTooShort	Segment too short for selected bending die

The messages of Low severity inform about the production mode, insignificant changes in geometry, or common warnings that do not normally cause problems and in most cases can be ignored.

Message	Explanation
BarProducedLoose	Bar will not be produced as part of mesh
MeshProducedLoose	Mesh will not be produced, only loose bars
stBiegevorrichtung_Nr	Assigned Pluristar submachine
StorageItem	Pre-produced mesh from storage
BendMesh	Bendings will be produced by BlueMesh
FlatMesh	Mesh has no bending to produce
MatIsProducedRastered	Mesh bars will be moved into grid
MatIsProducedRasterfree	Mesh will be produced without grid
MatIsProducedTurned	Mesh will be produced turned upside-down
MeshMovedCollisionAZ	Mesh moved to avoid collision with claws
UnrolledMesh	Mesh had bendings but will be produced flat
BarMovedAwayFromCorner	Bar moved to avoid collision with bending die
BarMovedIntoCorner	Bar snapped into corner for better results
UnderTMXMinCarryLen	Short bar is lengthened slightly
BendingDeviceChanged	Bending diameter corrected for Hambi
BendingRChanged	Assigned die changed bending height
stBendingDevice	Assigned Pluristar bending die
stBendingDeviceChanged	Actual bending die is smaller than designed
stBendingDMDifferent	Bending diameter was reduced

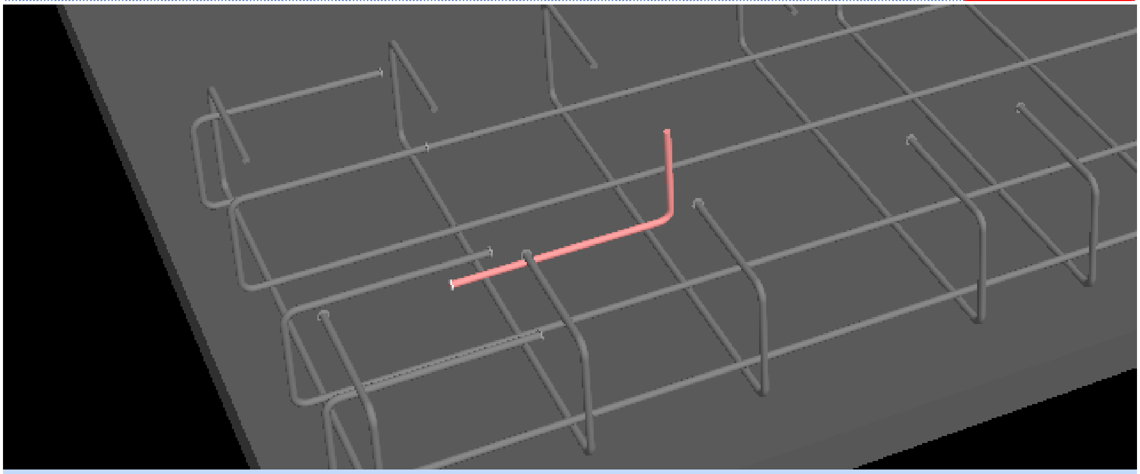
Other messages, not listed above, should be assumed severity depending on their type:
Error → High, Warning → Medium, Hint and Info → Low.

Transformations may cause errors

It is important to keep in mind that the automatic transformations done by the machine software may in some special cases cause other PTS errors. In the example below the real length of the bar visible in AviCAD is 351.99 mm but despite that, the PTS shows `BarTooShort` error for this bar.

Angle	Wire	Length	External length	Real length	X	Y	Z	Absolute X / Y / Z	d X / Y / Z
	8	358	366	351.99	190	294.5	10	235 / 340 / 45	246 / 0 / 112

Bending angle	Rotation angle	Segment length	Segment external length	Real segment length	Bending radius
		246	250	243	
90		112	116	109	10



Drag a column header here to group by that column.

		SWR	KVG-203	BarTooShort	Bar does not meet the minimum producible length of 350 mm
		DataCtrl		ImportInfo	Duration 5767 ms / SingleSlabMode = True
		SWR	KVG-203	MatIsProducedRasterfree	Mesh is produced grid-free.

The reason is that the bar is designed with the bending radius of 10 mm, but the smallest bending die available for Ø8 is D32. This causes change of the bending radius which makes the bar a little shorter, the real length becomes 349.42 mm, lower than the minimum requirement of 350 mm.

Automatic cage adjustments

Since the introduction of Hambi mesh bender, we have experienced problems with the incorrect cage dimensions after bending due to imprecise modelling of corner bars in Tekla, and inherent errors in export of cages caused by the conversion from the actual to nominal rebar diameters.

Another problem was that cages were often designed incorrectly. When the modelled bending diameter was different than the diameter of bending heads, the real cage and segment lengths would deviate from the design, affecting the dimensions after bending. Often the bent segments were too short and thus impossible to bend, or stirrups were placed such, that some would not match the bending heads, requiring time-consuming adjustments. See requirements for cages.

Because of these problems all cages are now automatically adjusted upon import of PXML to profit or when testing the reinforcement in PTS. The bending diameters are corrected, and corner bars snapped precisely into the center of bending or moved past the bending cam. Additionally, the segment lengths and stirrup positions are checked to be readily producible.

The table below gives an overview of PTS messages generated by the cage adjustments.

Message	Explanation
⚠ BarMovedIntoCorner	Bar was snapped to bending center to improve bending precision
⚠ BarMovedAwayFromCorner	Bar was shifted past cams to avoid collision when bending
⚠ BendingDeviceChanged	Bending diameter was adjusted to cams diameter of 40 mm
⚠ NonStandardSpacing	Stirrup positions do not match standard spacing of bending cams
✖ SegmentTooShort	Segment is too short and would slip from the bending arm

The `BarMovedIntoCorner` and `BarMovedAwayFromCorner` do not require any changes, because perfect accuracy is not possible in Tekla due to export issues. The `BendingDeviceChanged` can be ignored when not causing other problems, like too short mesh. The `NonStandardSpacing` should be corrected unless done on purpose in agreement with the factory. Only the `SegmentTooShort` errors are “critical” and must always be corrected.

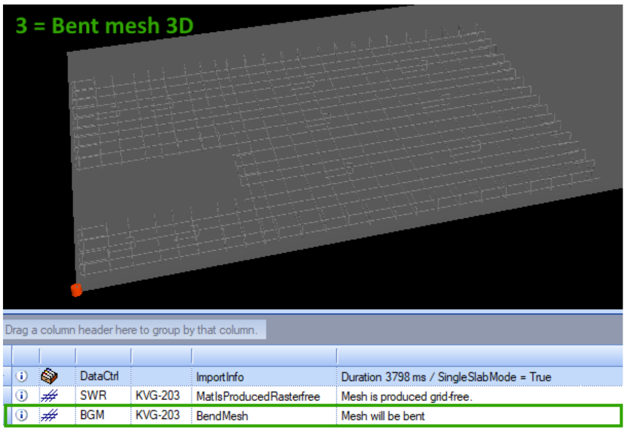
Additionally, the `MovedToRaster` and `BarMovedCollisionAZ` messages are shown as warnings when they occur in cages and designer should check if the positions of corner bars are not affected. For meshes these messages are only informative, as there they rarely cause any issues.

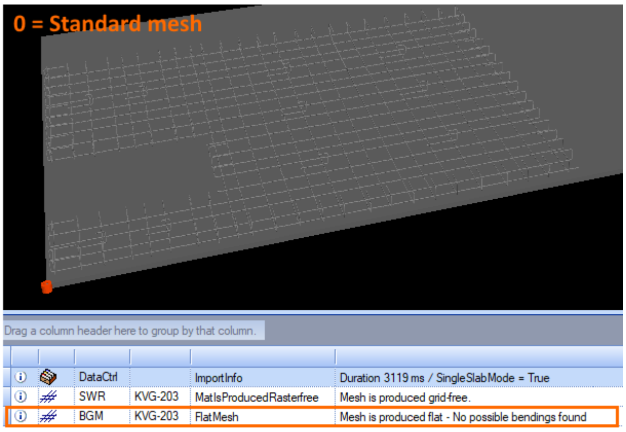
Production bent, unrolled or flat

A mesh modelled with bendings may also be produced flat. For example, cages are purposedly produced flat and later bent manually on a mesh bender. In general, when the mesh type is set to `3 = bent mesh 3D`, the bendings are produced and when it is set to `0 = standard mesh`, the bendings, if present, are straightened and the mesh is produced flat.

PTS shows info messages as an additional control if the correct production mode is used:

- `BendMesh` when mesh will be produced with bendings by BlueMesh.
- `FlatMesh` when mesh has a flat geometry and no bendings to produce.
- `UnrolledMesh` when mesh had bendings but will be produced flat (unrolled).





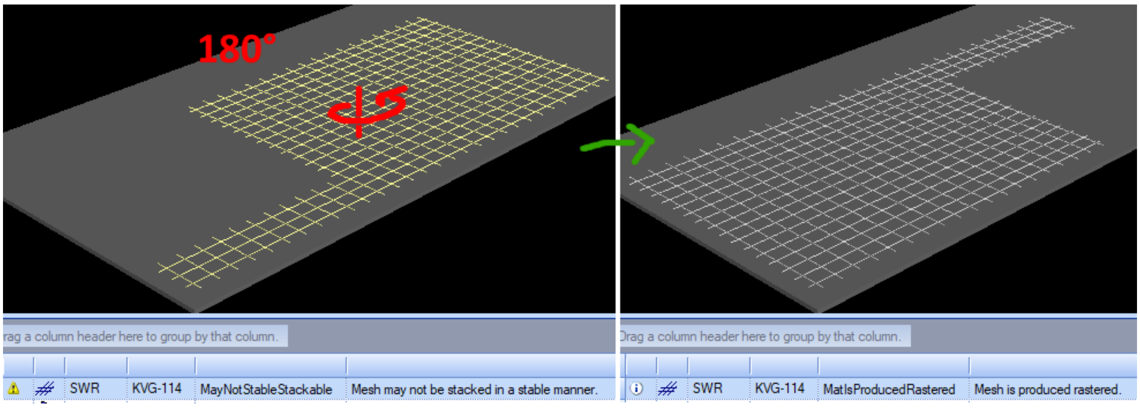
Mesh may not be stacked in a stable manner

The `MayNotStableStackable` warnings can indicate two potential problems with the mesh.

The flat meshes are extracted from the machine using a staple wagon, that uses claws to grab the mesh in the front and pull it away from the welding robot to the flat mesh outlet. In the outlet the meshes are normally first stacked on top of each other before such stack needs to be unloaded.

In the example below the narrow part of the mesh is in the front and the staple wagon would need to pull the mesh off-center, which could cause the mesh to rotate during the transport. A wide mesh could potentially hit some part of the machine or get blocked. Even a small rotation could cause stability issues when meshes are stacked on top of each other in the outlet.

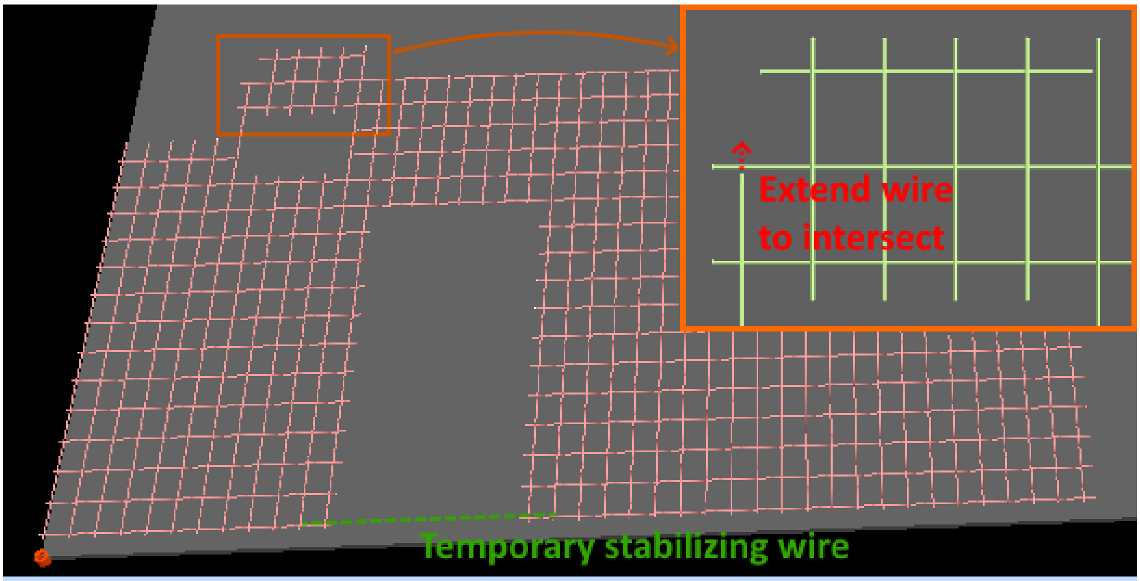
In this case it is best to rotate the mesh 180° around the Z axis, so the wide part is in the front.



For more complicated meshes these warnings may not always be possible to solve. The meshes are entirely producible so in special cases the warnings can be ignored. The production will require special attention from the operator and may be slower because of more frequent unloading.

There is no way to extract the mesh

The `MatInconsistent` errors mean that the machine sees the mesh too unstable to extract. When pulled by the extractor it could be bent or distorted. Every "region" of the mesh must be connected with at least 2 welding points to the other parts of the mesh.



Drag a column header here to group by that column.

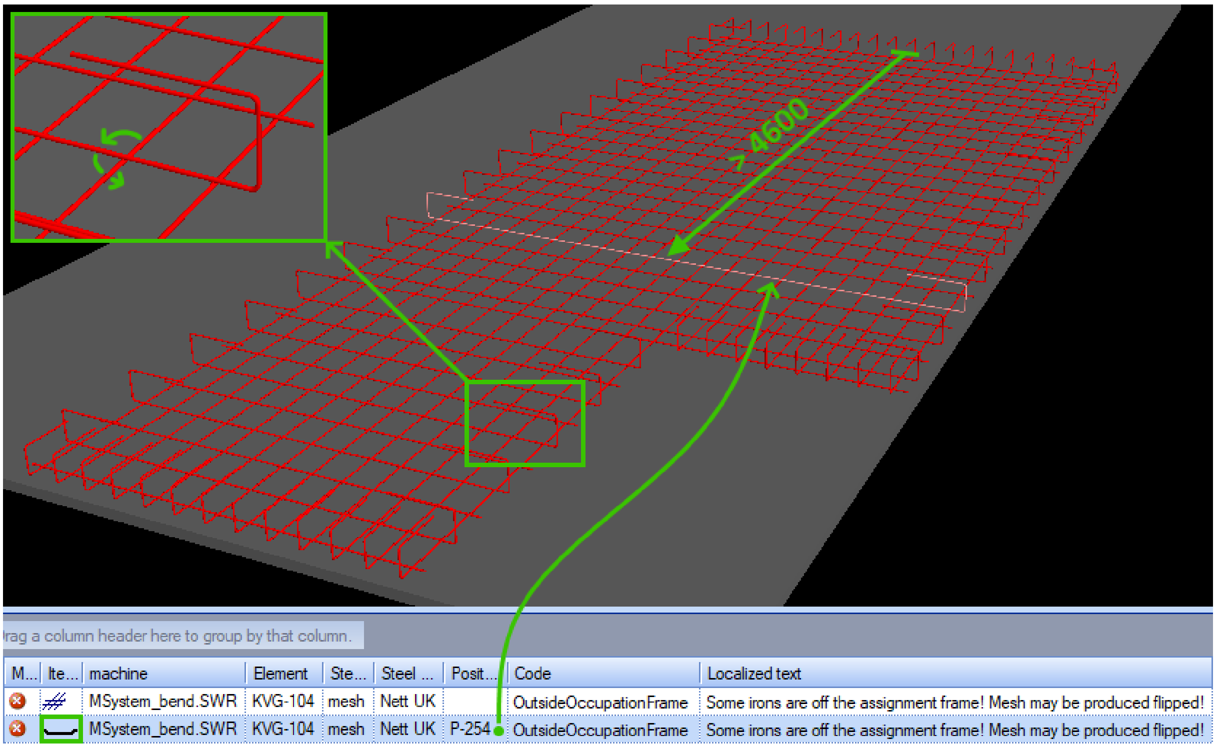
⊗		SWR	KVG-302	MatInconsistent	There is no way to extract the mesh!
ⓘ		DataCtrl		ImportInfo	Duration 9198 ms / SingleSlabMode = True
ⓘ		SWR	KVG-302	MatIsProducedRasterfree	Mesh is produced grid-free.
ⓘ		SWR	KVG-302	MatIsProducedTurned	Mesh will be produced turned.
ⓘ		BGM	KVG-302	FlatMesh	Mesh is produced flat - No possible bendings found

Here the transversal wire should be extended past the intersection to add a welding point, as marked in red. The error would then disappear, but the mesh would remain susceptible to distortion and difficult to transport because of the big opening with an open edge. Therefore, the bottommost wire should be continued through the opening, as marked in green.

Some irons are off the assignment frame

The `OutsideOccupationFrame` errors can be confusing at first, because they figuratively mean that some longitudinal bars are placed outside of the pallet, for example when a mesh is too wide. The error usually comes twice in the results for each problematic mesh, once for the whole mesh, and once for the first bar which lies outside of the pallet.

Such bars should normally be easy to spot but in practice these errors often come for a mesh that at a first sight fits inside the pallet just fine. The actual problem here is the layer order. The longitudinal bars must always be above, and when they are not, the machine rotates the mesh perpendicularly to how it was designed, and thus it no longer fits within the pallet. The solution is to swap the order of layers, such that the longitudinal bars are placed above the transversal bars.

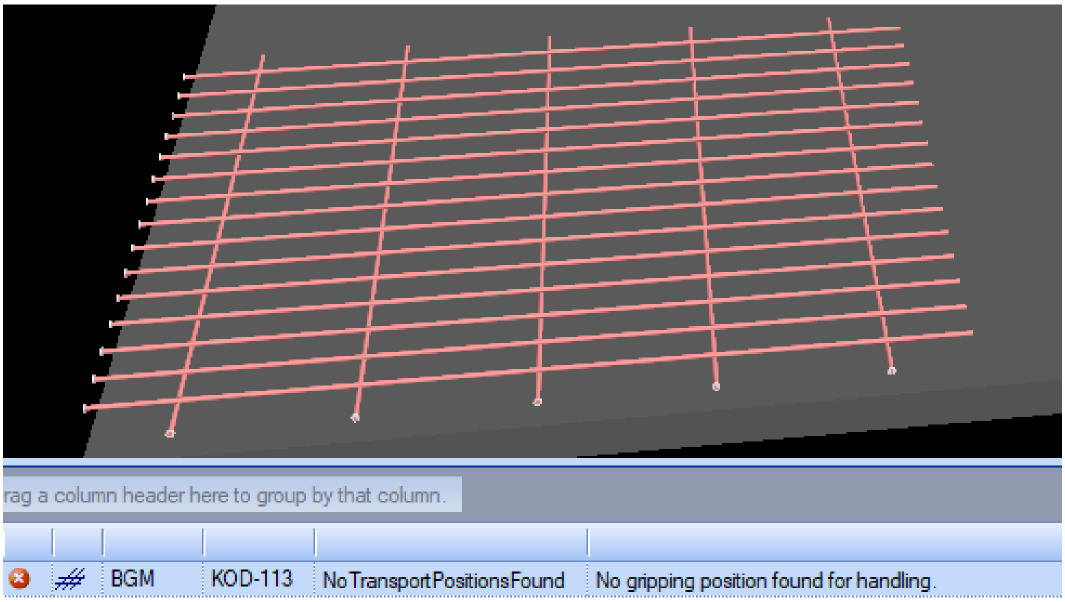


No gripping position found for handling

The `NoTransportPositionsFound` errors occur when none of the four grippers that transport the mesh in the bending portal can find gripping positions. The `OnlyOneTransportPositionFound` warnings occur when only one such position can be found. The transportation with one gripper is not stable, but may be sufficient for small, flat meshes. For bent meshes it should be avoided.

The grippers always grip the longitudinal bars at the crossing points or between them. The minimum center-center spacing that gives enough space for grippers can be calculated as `49 mm + Ø / 2`. The minimum spacing is bigger towards the bent bars because the gripper robot is bigger above the tongs and the bent bars require overbending, getting closer to the gripper during the bend.

The errors happen most often when the spacing of longitudinal bars is too narrow. Increasing a few spacings should allow the machine to grip the bars. Other possibility is to rotate the whole mesh 90° around the Z axis to try gripping in the perpendicular direction, but it may not always be possible.

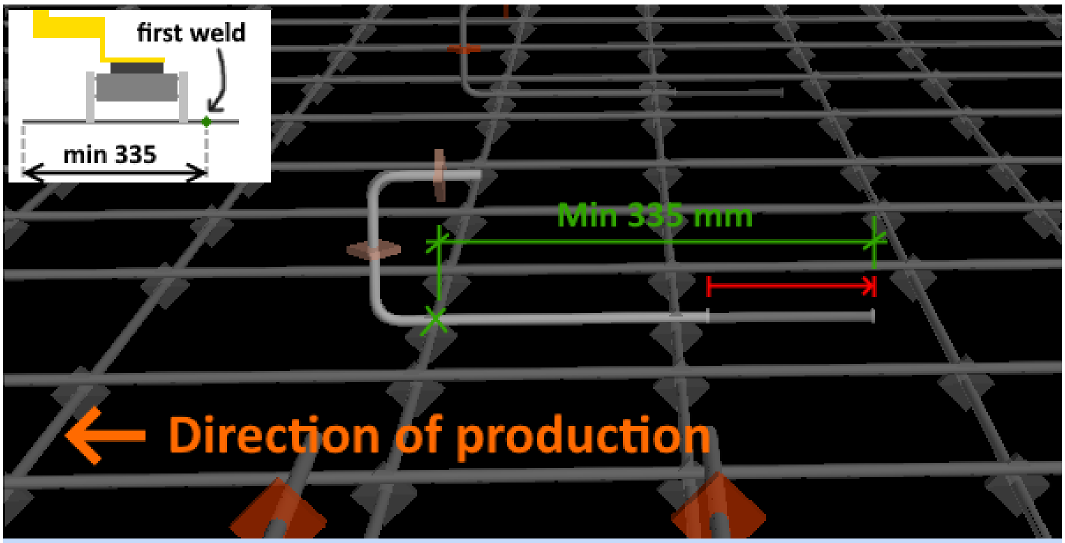


Previously such errors could occur when the gripper tongs collided with the green transport sliders, but now the software automatically shifts the whole mesh to allow gripping.

TMX minimum support length not reached

The `UnderTMXMinCarryLen` messages inform that the bar was extended to meet the minimum required distance of 335 mm from the first welding point to the end of the bar. This is because the magnets that transport the longitudinal bars to the welding portal must have enough space to hold the wires properly. It requires no further action as the extension of bars is done automatically.

However, in case of the mesh below an alternative, better solution is possible. To avoid extending the bars, either whole mesh, or the space holders could be rotated 180° around the Z axis, so that the bent part is at the trailing end and can be used by the magnets.

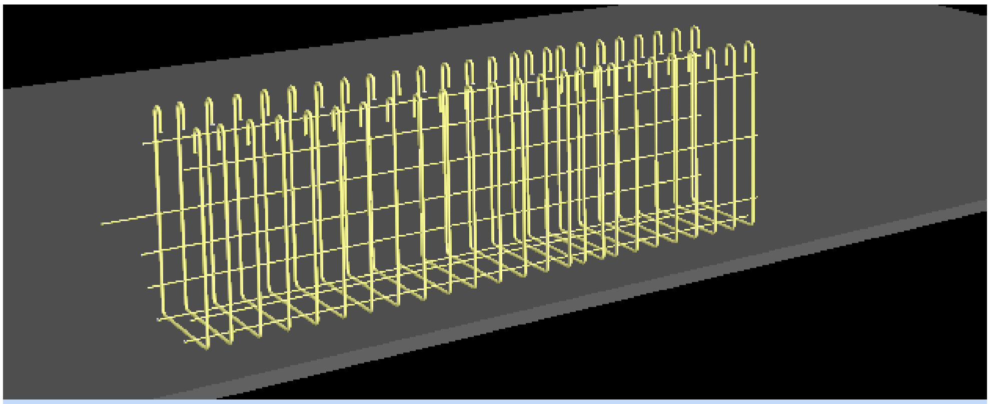


Drag a column header here to group by that column.

	ⓘ		SWR	KVG-203	UnderTMXMinCarryLen	TMX minimum support length not reached. Extend bar.
▶	ⓘ		SWR	KVG-203	UnderTMXMinCarryLen	TMX minimum support length not reached. Extend bar.
	ⓘ		SWR	KVG-203	UnderTMXMinCarryLen	TMX minimum support length not reached. Extend bar.

No valid label position found

The `NoValidLabelPosFound` warnings occur when the label cannot be placed as described here. The machine will print the label and stop so the operator can place it by hand. When possible, such stop should be avoided – it is often enough to modify the spacings of bars slightly. In the example below two bars were lengthened to make place for the label, but there is one more thing to consider.



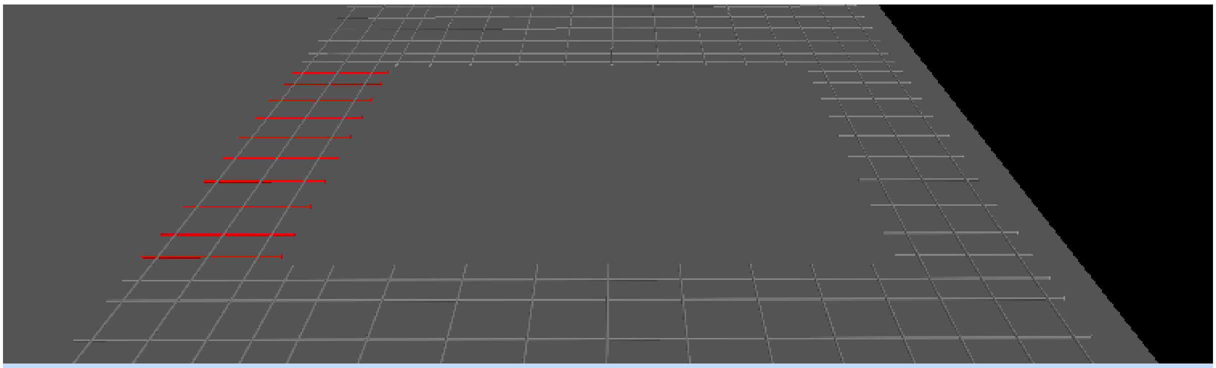
Drag a column header here to group by that column.

▶	⚠		SWR	FDB-116	NoValidLabelPosFound	No valid label position found.
	ⓘ		SWR	FDB-116	MovedCollisionAZ	Y position has been shifted to prevent collision with the grippers of AZ2.
	ⓘ		SWR	FDB-116	MatIsProducedRasterfree	Mesh is produced grid-free.

The extractor grips the first bar, and it would destroy the label placed beneath it. Rotating the mesh 180° around Z axis solves this problem, because then the label will be placed at the end.

Iron cannot be placed correctly on the feeder

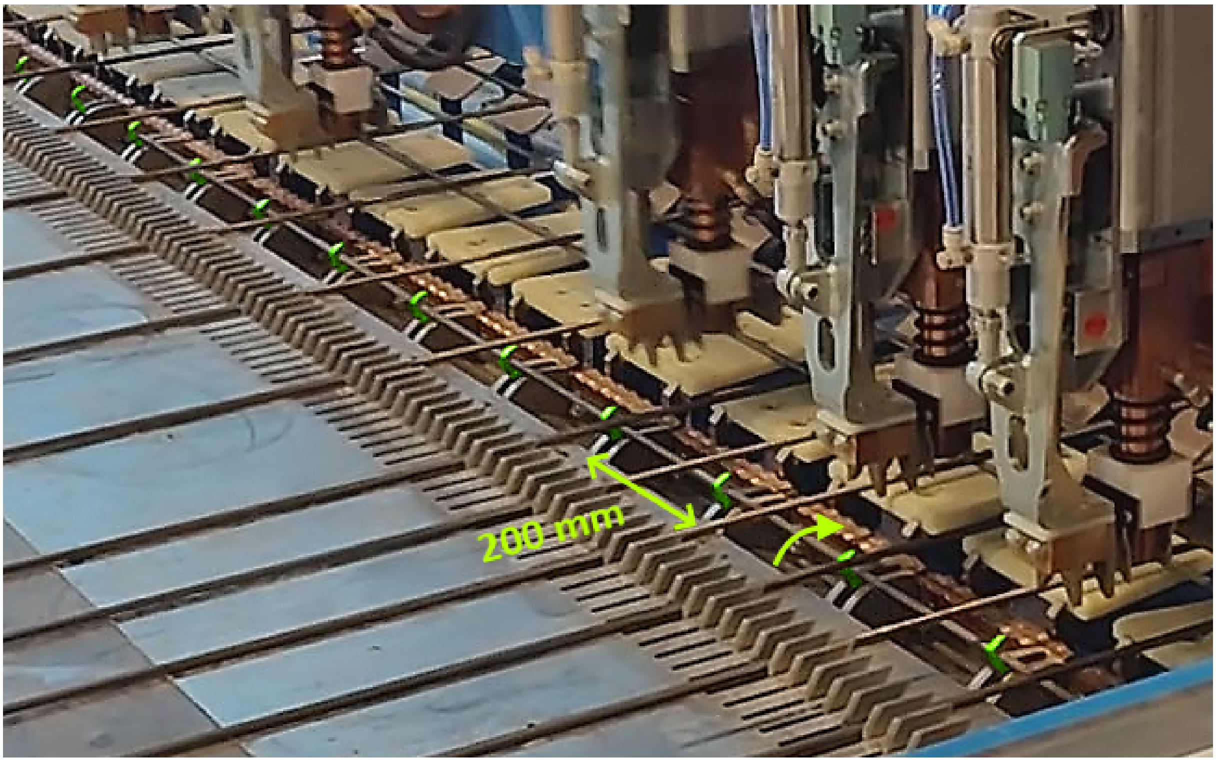
The `BarNotLyingCorrectOnFeedingFinger` errors happen when multiple short transversal bars are placed in the same line. The problem here is that such bars, depending on their positions, could fall from the feeding fingers which move the transversal bars onto the counter electrode.



rag a column header here to group by that column.

M...	Ite...	machine	Element	Ste...	Steel Name	Pos...	Code	Localized text
		MSystem_flat.SWR	SWG-140	mesh	Nett yttersjikt	N-57	BarNotLyingCorrect OnFeedingFinger	The iron cannot be placed correctly on the feeder finger (-> lengthen the iron to facilitate safe feeding).

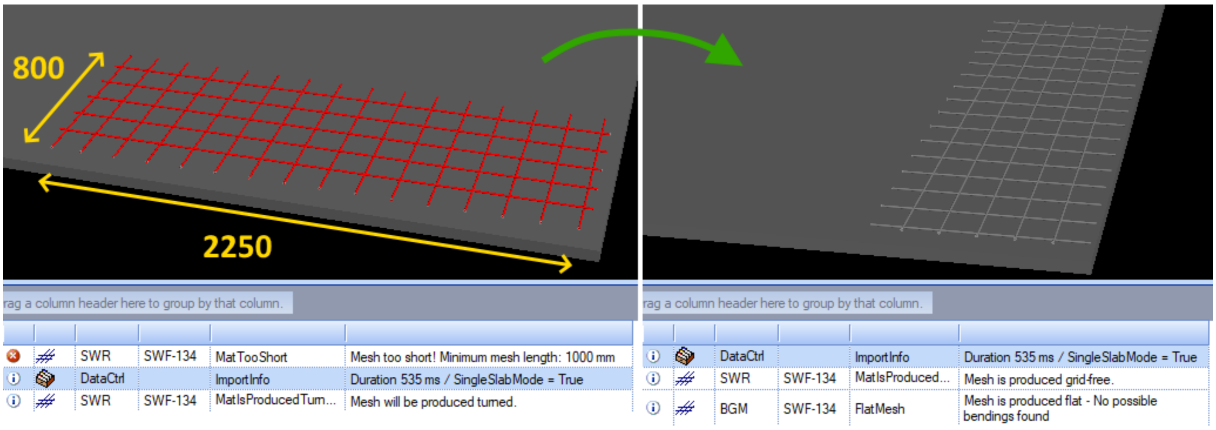
When the bars are long enough to be placed onto 3 or more feeding fingers (marked below in green), they are stable and will never cause such problems. But if the bars are very short and positioned such that only 2 fingers are used, they could fall down when placed off-center. If there is only one such bar in a line, the machine can solve this by moving the whole mesh. But if there are two or more bars in a line, this is not always possible. The spacing between the feeder fingers is 200 mm.



To solve such problems, the bars lengths or their relative positions in the same transversal line must be adjusted. This can be done by trial and error in AviCAD, and once a solution is found, a similar change should be done in Tekla for all such bars.

Mesh too short

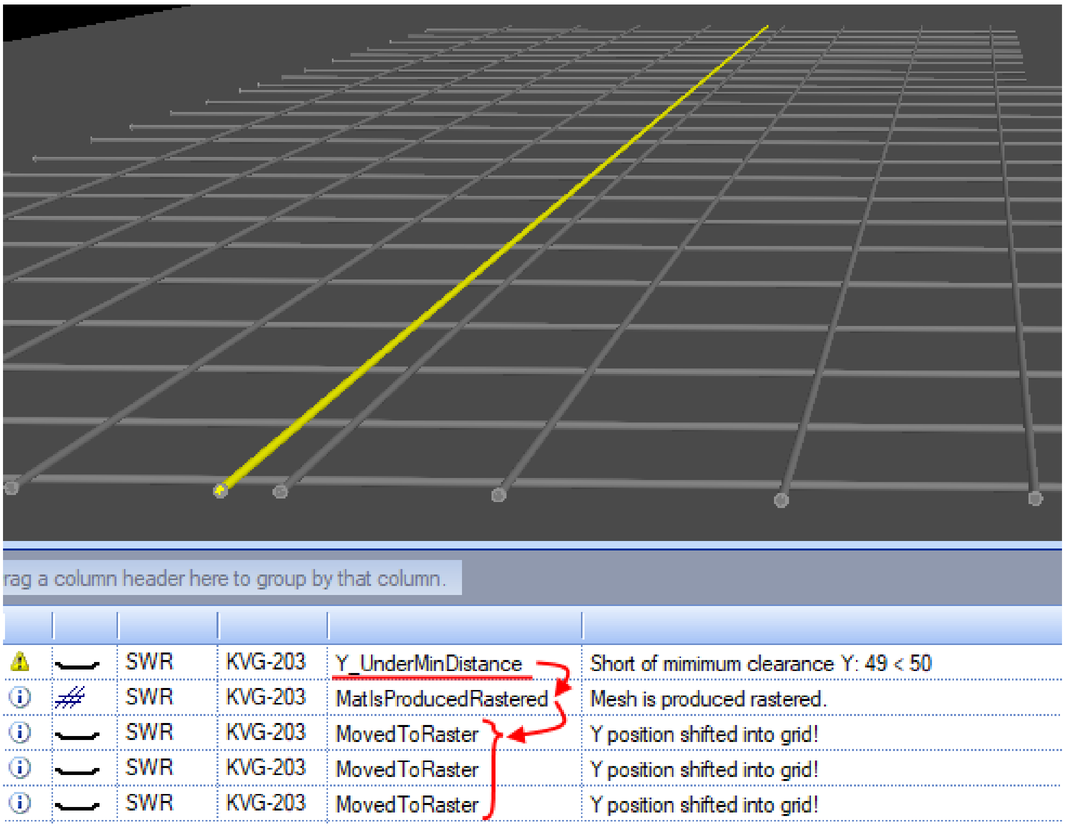
The `MatTooShort` errors are caused by the mesh not fulfilling the minimum length requirement. Because the machine attempts to produce meshes as oriented in AviCAD, this can often be fixed by rotating the mesh 90 degrees in the Z axis. The rotation can be done manually in AviCAD or automatically during export.



Meshes that do not meet the length requirement in any direction cannot be produced using the BlueMesh. It is possible to produce these as bigger mats to be cut into required pieces, but it must be coordinated with the factory on a case-by-case basis.

Short of minimum clearance

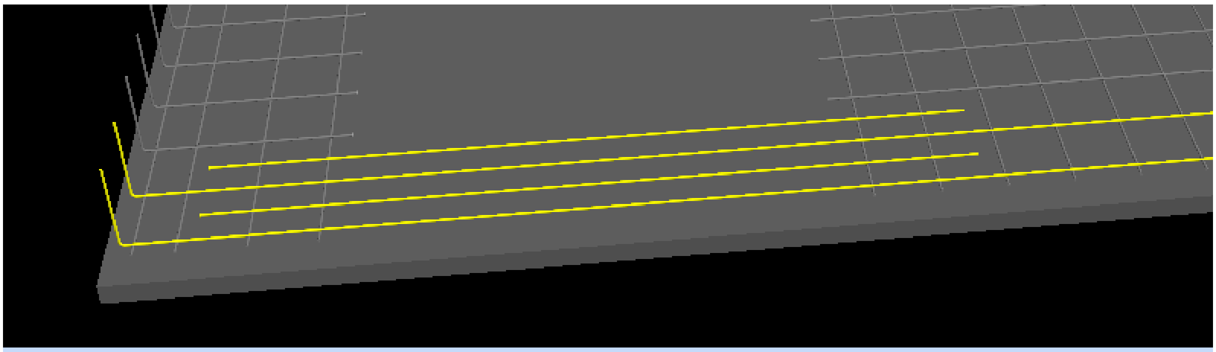
The `Y_UnderMinDistance` warnings inform that the spacing between longitudinal wires is below the minimum necessary for the gridless production. This forces the rastered production, which in turn results in the automatic shifting of wires into the grid.



Maximum welding point distance exceeded

The `WeldingPointDistanceExceeded` warnings occur when spacing between transverse wires exceeds 450 mm in a mesh produced gridless, or 500 mm in a mesh produced rastered (with grid). Spacings above such limit are often acceptable and the warnings can be ignored, but special attention is required from both the designer and the machine operator.

For the specific mesh below the spacing is too big, and it is better to split it into two separate pieces.

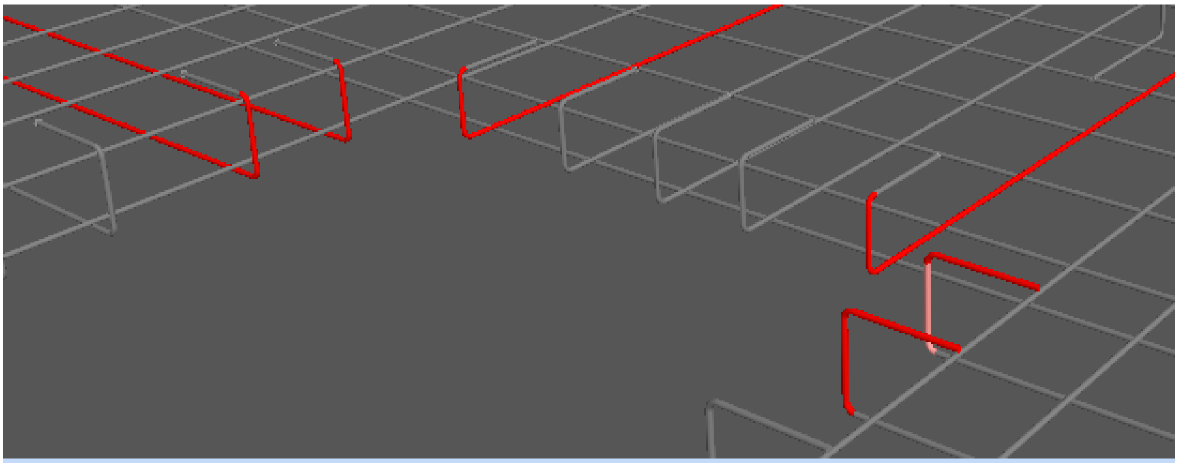


rag a column header here to group by that column.

M...	Ite...	machine	Element	Ste...	Steel ...	Posit...	Code	Localized text
⚠		MSystem_flat.SWR	SWF-101	mesh	Nett YS	P-64	WeldingPointDistanceExceeded	Maximum welding point distance of 500 mm exceeded! Current distance: 1200 mm
⚠		MSystem_flat.SWR	SWF-101	mesh	Nett YS	P-64	WeldingPointDistanceExceeded	Maximum welding point distance of 500 mm exceeded! Current distance: 1200 mm

Bending / double-bending not available

The `NoBendingDeviceFound` errors occur when the machine cannot produce a bending, often because there is not enough space for the bending head. The production of the mesh is possible, but the bendings with errors are skipped, so in principle all errors of this type should be fixed.



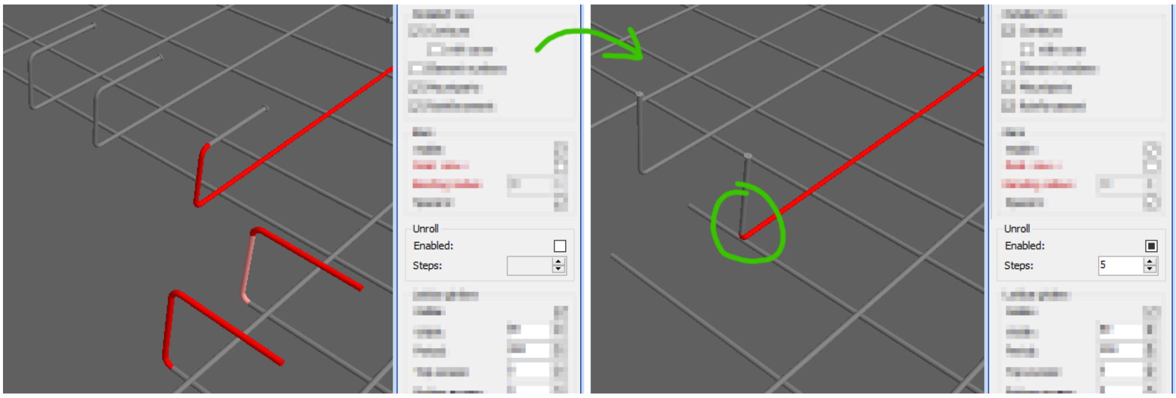
rag a column header here to group by that column.

⊗		BGM	KVG-203	NoBendingDeviceFound_CollisionBetweenBendings	Bending / double-bending not available. - No potential bending equipment found - collision on a bend or between the bends.

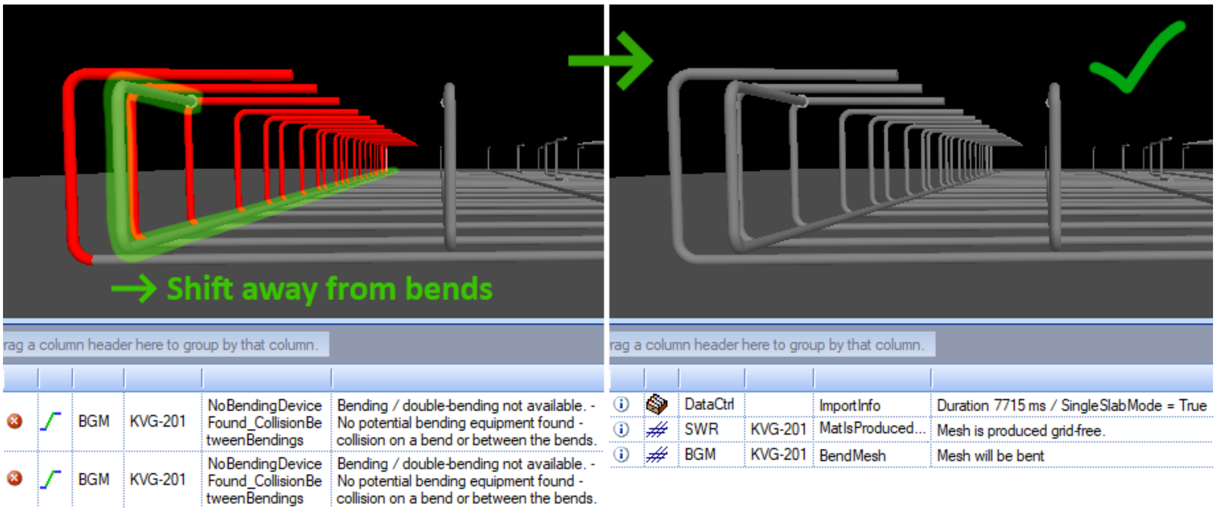
There are 4 different types of the `NoBendingDeviceFound` errors:

1. `NoBendingDeviceFound_BendingSetNotReachable`: None of the bending robots can reach the bending. For our machine both robots always have enough space.
2. `NoBendingDeviceFound_NoBoltQuotaFound`: The bending bolt cannot bend the bar because the bent segment is too short.
3. `NoBendingDeviceFound_CollisionBetweenBendings`: The bending head collides with another bar during bending or at the move from one bending to the next on the same bar.
4. `NoBendingDeviceFound_NoThreadInOutFound`: There is not enough space around the bending to insert (thread in) or to thread out the bending head from the bar.

When the error concerns an intermediate bending, the unrolling feature can be very helpful.

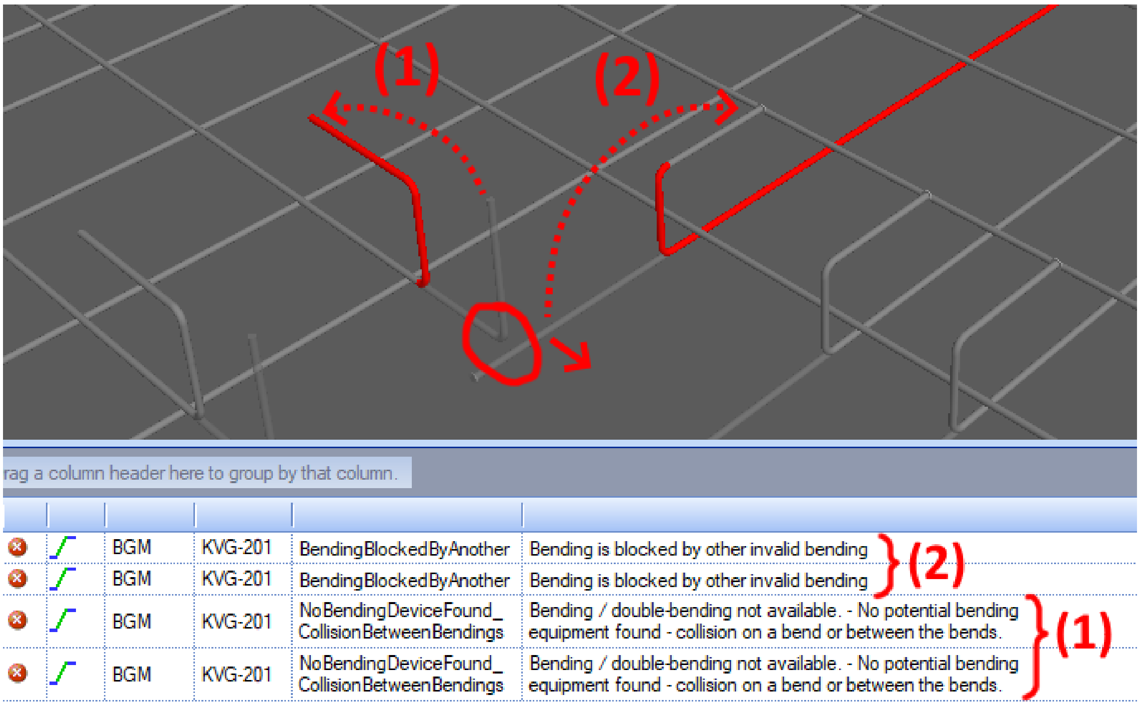


When the error comes for all bendings in an edge or in an opening, the problem is often not with the highlighted bent bars, but with the perpendicular edge bar that comes too close to the bendings, marked below in green. Moving it slightly towards inside of the mesh fixes the errors.



Bending is blocked by other invalid bending

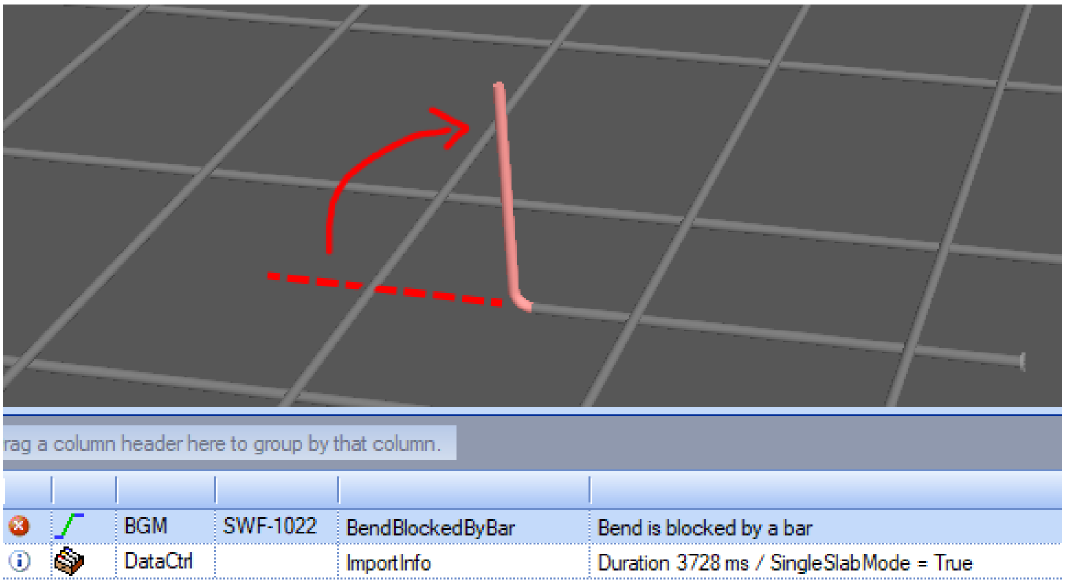
The `BendingBlockedByAnother` errors occur when a bending cannot be made, because its path is blocked by another bar that could not be bent. Here it is important to identify the original problem and fix it first. In the case below, the bending (1) cannot be made because it is too close to the perpendicular bar (circled). Shifting this bar away makes the bending (1) possible, which in turn opens the space necessary to make the bending (2) as well and fixes both errors.



⊗		BGM	KVG-201	BendingBlockedByAnother	Bending is blocked by other invalid bending	(2)
⊗		BGM	KVG-201	BendingBlockedByAnother	Bending is blocked by other invalid bending	(2)
⊗		BGM	KVG-201	NoBendingDeviceFound_ CollisionBetweenBendings	Bending / double-bending not available. - No potential bending equipment found - collision on a bend or between the bends.	(1)
⊗		BGM	KVG-201	NoBendingDeviceFound_ CollisionBetweenBendings	Bending / double-bending not available. - No potential bending equipment found - collision on a bend or between the bends.	(1)

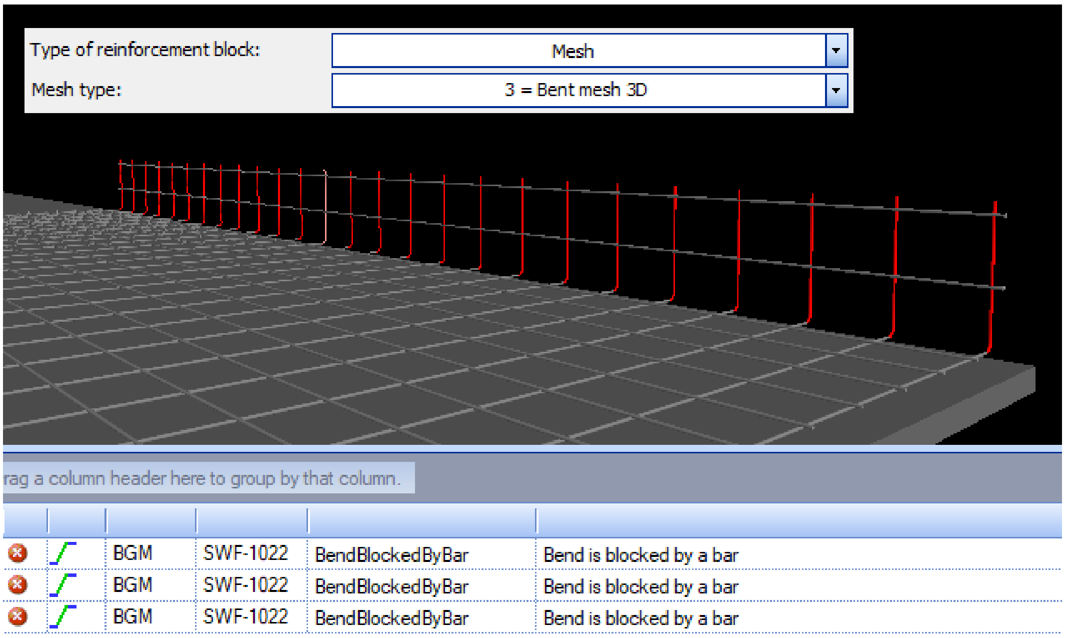
Bend is blocked by a bar

The `BendBlockedByBar` errors normally indicate that the erroring bending cannot be made because of a crossing wire that blocks its path. The reason in this specific case is rather clear.



⊗		BGM	SWF-1022	BendBlockedByBar	Bend is blocked by a bar
ⓘ		DataCtrl		ImportInfo	Duration 3728 ms / SingleSlabMode = True

Another situation where these errors occur are bendings with crossing wires. Here it may not be immediately obvious which bar is blocking the bending, but it is important to understand, that mesh is first welded flat, and the bars are always bent one by one. Then every one of these bendings would be blocked by the two crossing wires kept in place by the other bars.



The possible solutions to produce a mesh of this type are:

a) Export as a cage to be produced flat and bent manually afterwards.
b) Export the crossing wires as loose reinforcement to be added manually.
c) Split the mesh into two parts, one flat and another bent with some overlap.

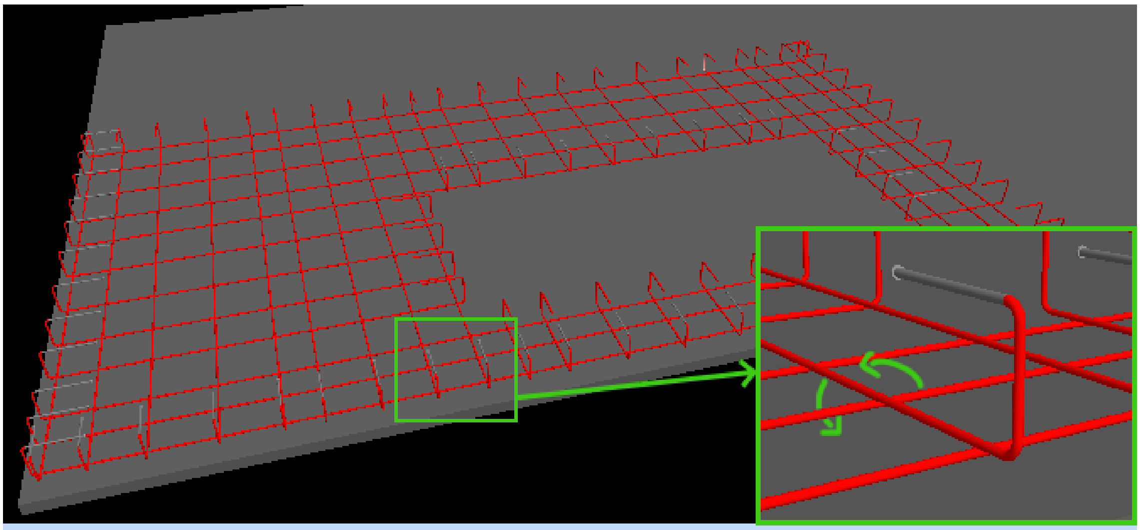
The variant c) is preferred by the factory as it is simplest to assemble and benefits from automated production to the greatest extent. It should be used whenever possible.

Negative bending not available

The `NegativeBendingNotAllowed` errors mean figuratively that the downward (negative) bendings cannot be done as the machine is capable of only making upward (positive) bendings.

When first encountered this can be confusing, as all bendings are clearly pointed upwards. Here it is important to notice the “Mesh will be produced turned” message further down. The transversal bars are always welded from the underside but here they are modelled above the longitudinal bars. Because of that the machine rotates the whole mesh 180° in order to produce it. That causes all bendings to point downwards which makes them impossible to make.

The solution here is to swap the order of layers, so that the transversal bars are below, and the longitudinal bars are above, such that the mesh does not need to be turned. Smaller meshes could alternatively be rotated 180° around the Z axis, so that the transversal bars become longitudinal.



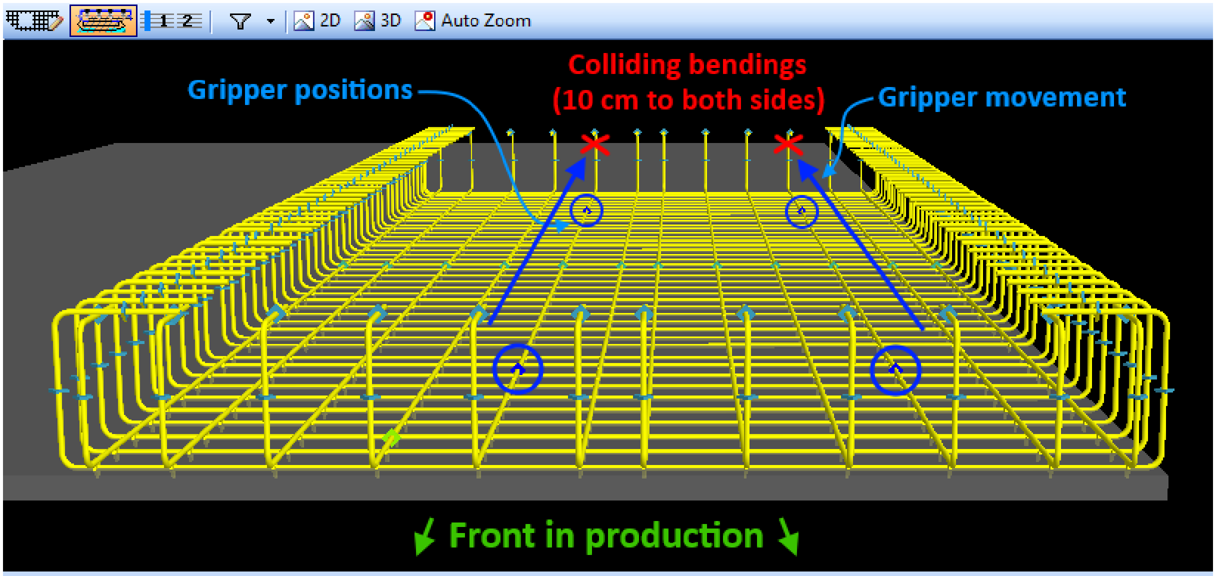
rag a column header here to group by that column.

⊗		BGM	KVG-201	NegativeBendingNotAllowed	Negative bending not available
⊗		BGM	KVG-201	NegativeBendingNotAllowed	Negative bending not available
ⓘ		SWR	KVG-201	MatIsProducedTurned	Mesh will be produced turned.

Limitation bent mesh: height

The `BendMeshDimensionExceeded_Height` warnings inform that the mesh may collide with the machine parts because of its height. Ideally, the mesh height should be reduced, but the production is often possible with special attention from the operator and using reduced speed.

The fixed parts of the machine like the steel frames give a clearance of about 500 mm, but there is only 300 mm under the grippers in their topmost position and thus the PTS warns above this height. The grippers are optimized to move back parallel to the pallet after they release the mesh and move to the side only after they come past the trailing end of the mesh. Because of that it will often be the bendings in the end that collide with the grippers.



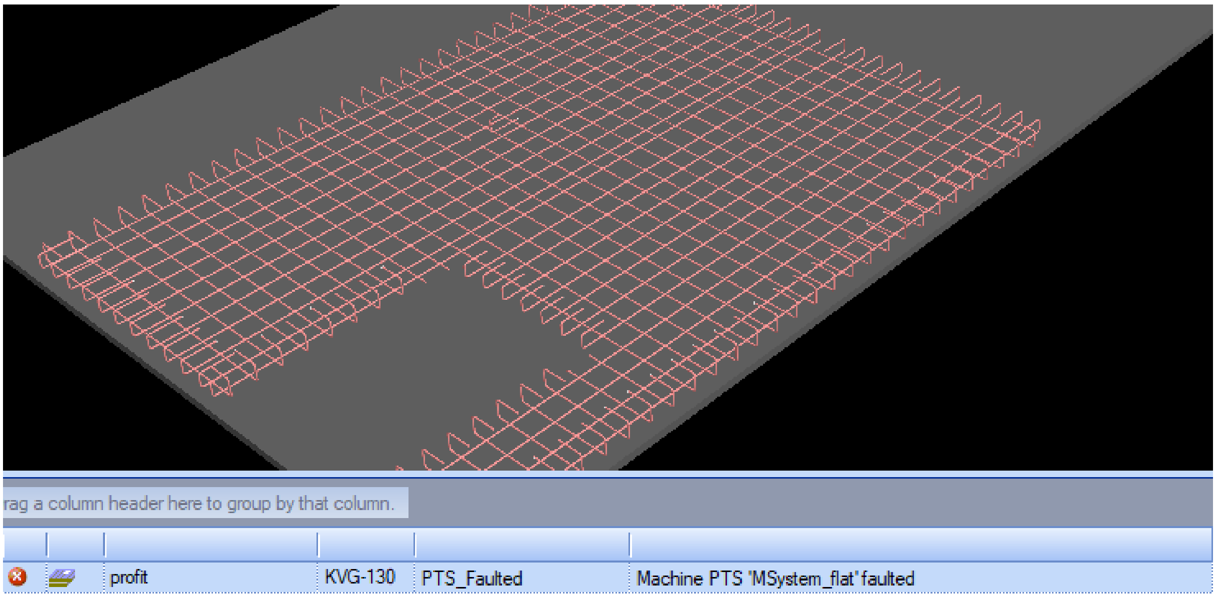
rag a column header here to group by that column.

M...	It...	machine	Element	Ste...	Steel Na...	Code	Localized text
⚠		MSystem_bend.BGM	KVG-127	mesh	Nett bunn	BendMeshDimensionExceeded_Height	Limitation bent mesh: height 321 oversteps 300

When height of the mesh cannot be reduced, removing, or lowering the bendings in the end of the mesh can help to avoid the problems with gripper collisions. The gripper positions are shown in the PTS received data and at least the bendings behind these positions should be removed. There must be at least 10 cm of free space in both directions from the centerline of each gripped bar.

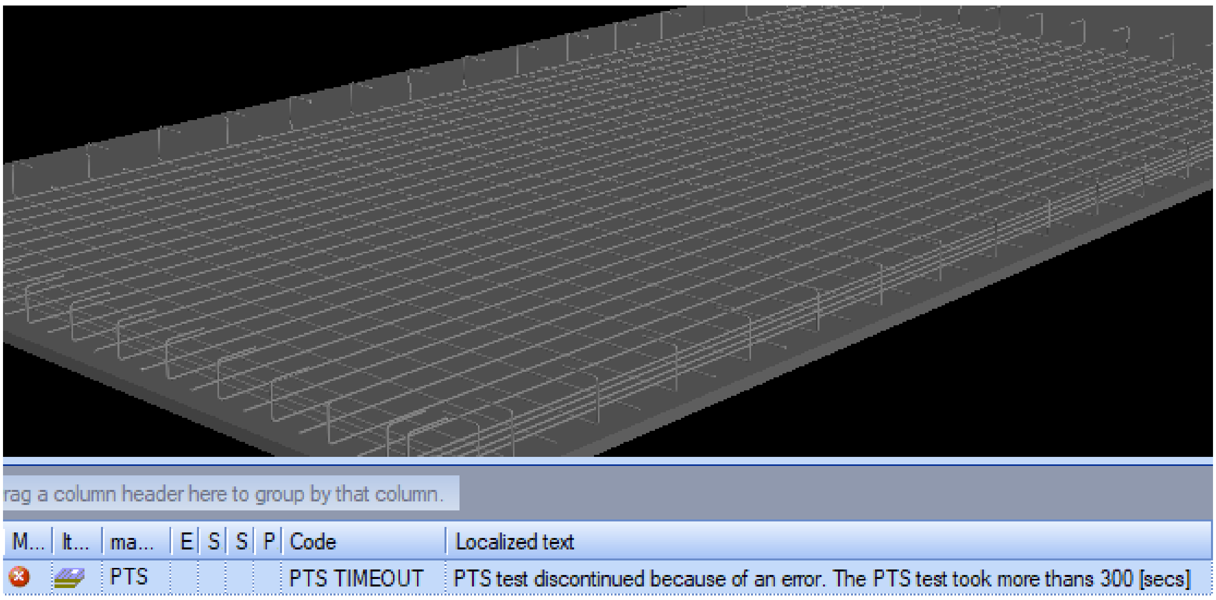
Test internal errors and timeouts

The `PTS_Faulted` errors occur when the Production Test Service crashed when checking the file. When such problems occur for an element, but not for other ones, it can mean that some special edge case is not handled correctly. Such errors should never occur, so if you see one, please report it immediately, including the tested PXML file.



The `PTS TIMEOUT` errors indicate other internal errors or problems with the PXML file. A control of a single file should normally take up to 15 or so seconds. If the test is still running after 5 minutes, the algorithm is terminated to prevent blocking the server for other requests.

The common reason for these errors is rebar sets duplicated dozens or hundreds of times due to error in Tekla caused by Edge and Corner reinforcement plugin used in element with added material. PXML files for even very heavy reinforced elements should not be bigger than 500-600 KB.

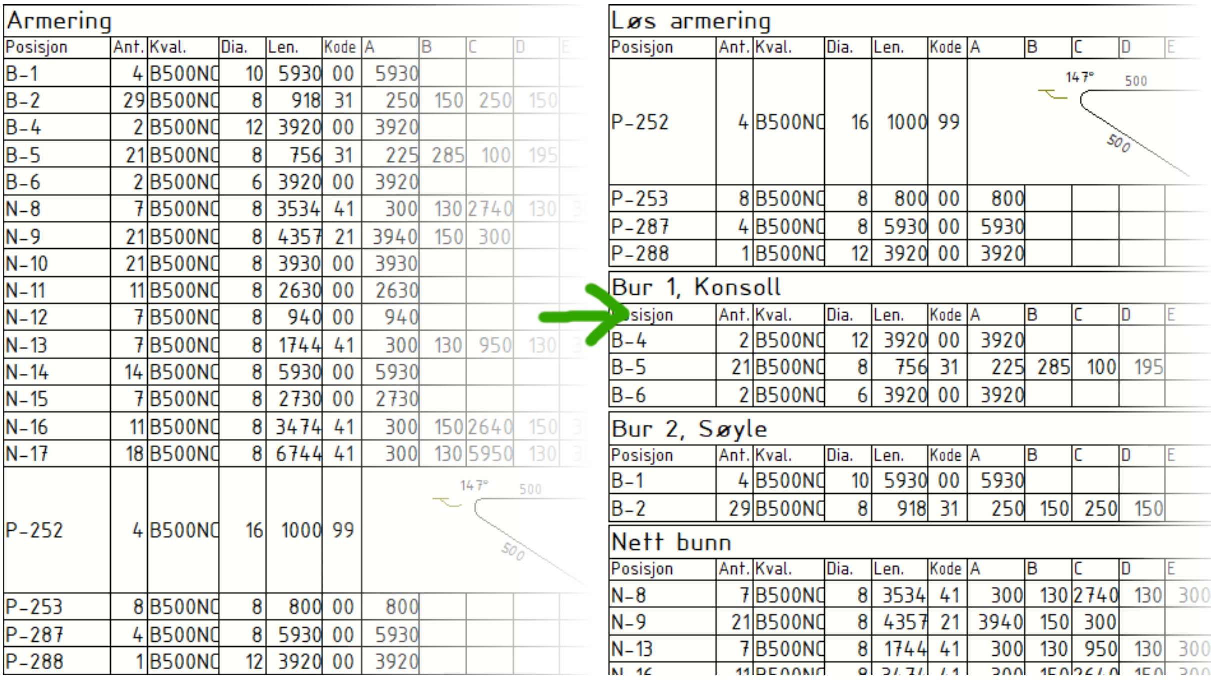


Production drawings

Enable grouping in the bending schedule

The bending schedule in production drawings should be grouped in a similar way as the PXML file.

Note that grouping of meshes works slightly different than in Precast Production Export, because drawings cannot access the mesh names specified in the export dialog. Therefore, all rebars with names that start with the word `Nett` are grouped together. Cages are grouped according to the `Cage group number`, exactly as in the export.



Armering

Posisjon	Ant.	Kval.	Dia.	Len.	Kode	A	B	C	D	E
B-1	4	B500NC	10	5930	00	5930				
B-2	29	B500NC	8	918	31	250	150	250	150	
B-4	2	B500NC	12	3920	00	3920				
B-5	21	B500NC	8	756	31	225	285	100	195	
B-6	2	B500NC	6	3920	00	3920				
N-8	7	B500NC	8	3534	41	300	130	2740	130	
N-9	21	B500NC	8	4357	21	3940	150	300		
N-10	21	B500NC	8	3930	00	3930				
N-11	11	B500NC	8	2630	00	2630				
N-12	7	B500NC	8	940	00	940				
N-13	7	B500NC	8	1744	41	300	130	950	130	
N-14	14	B500NC	8	5930	00	5930				
N-15	7	B500NC	8	2730	00	2730				
N-16	11	B500NC	8	3474	41	300	150	2640	150	
N-17	18	B500NC	8	6744	41	300	130	5950	130	
P-252	4	B500NC	16	1000	99	147° 500 500				
P-253	8	B500NC	8	800	00	800				
P-287	4	B500NC	8	5930	00	5930				
P-288	1	B500NC	12	3920	00	3920				

Løs armering

Posisjon	Ant.	Kval.	Dia.	Len.	Kode	A	B	C	D	E
P-252	4	B500NC	16	1000	99	147° 500 500				
P-253	8	B500NC	8	800	00	800				
P-287	4	B500NC	8	5930	00	5930				
P-288	1	B500NC	12	3920	00	3920				

Bur 1, Konsoll

Posisjon	Ant.	Kval.	Dia.	Len.	Kode	A	B	C	D	E
B-4	2	B500NC	12	3920	00	3920				
B-5	21	B500NC	8	756	31	225	285	100	195	
B-6	2	B500NC	6	3920	00	3920				

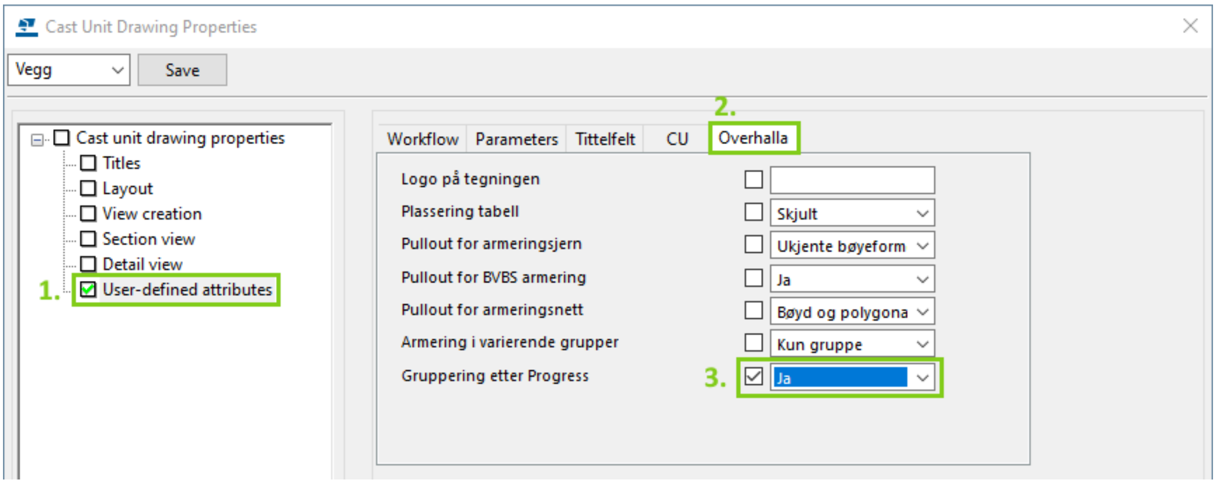
Bur 2, Søyle

Posisjon	Ant.	Kval.	Dia.	Len.	Kode	A	B	C	D	E
B-1	4	B500NC	10	5930	00	5930				
B-2	29	B500NC	8	918	31	250	150	250	150	

Nett bunn

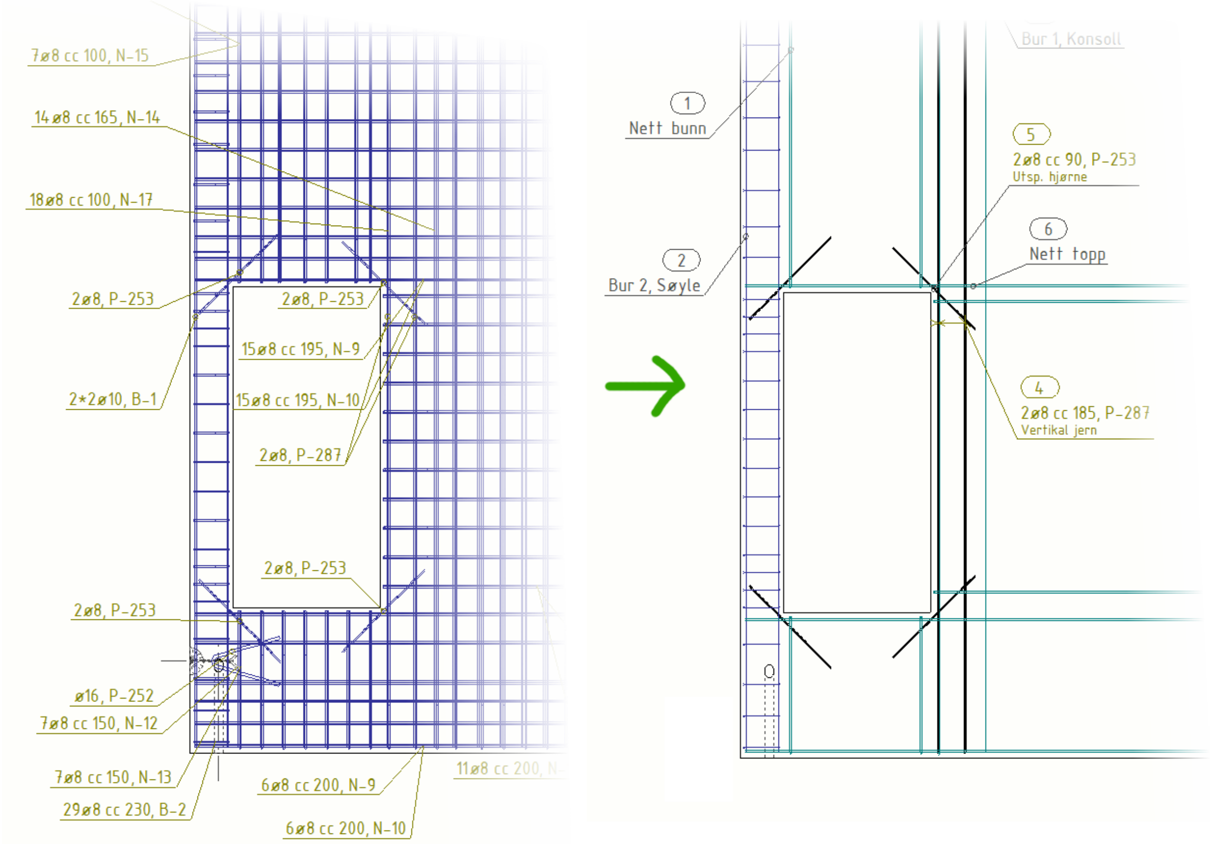
Posisjon	Ant.	Kval.	Dia.	Len.	Kode	A	B	C	D	E
N-8	7	B500NC	8	3534	41	300	130	2740	130	300
N-9	21	B500NC	8	4357	21	3940	150	300		
N-13	7	B500NC	8	1744	41	300	130	950	130	300

To enable grouping, open the drawing properties and got to the `User-defined attributes > Overhalla` tab and change the `Gruppering etter Progress` setting to `Ja`. Re-open drawing to see the change in the bending schedule.

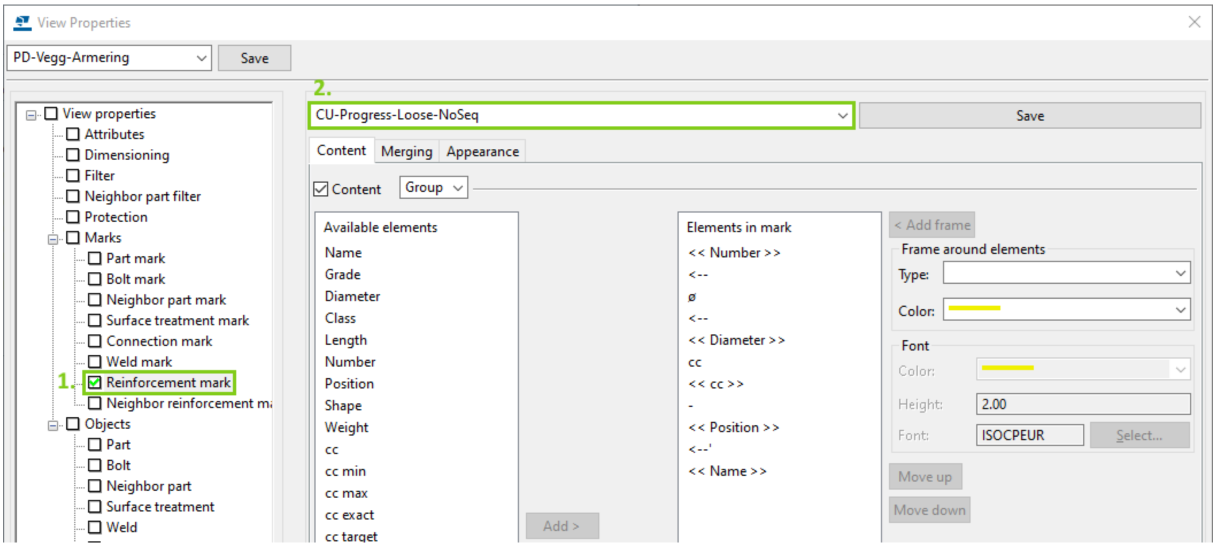


Configure marks for reinforcement groups

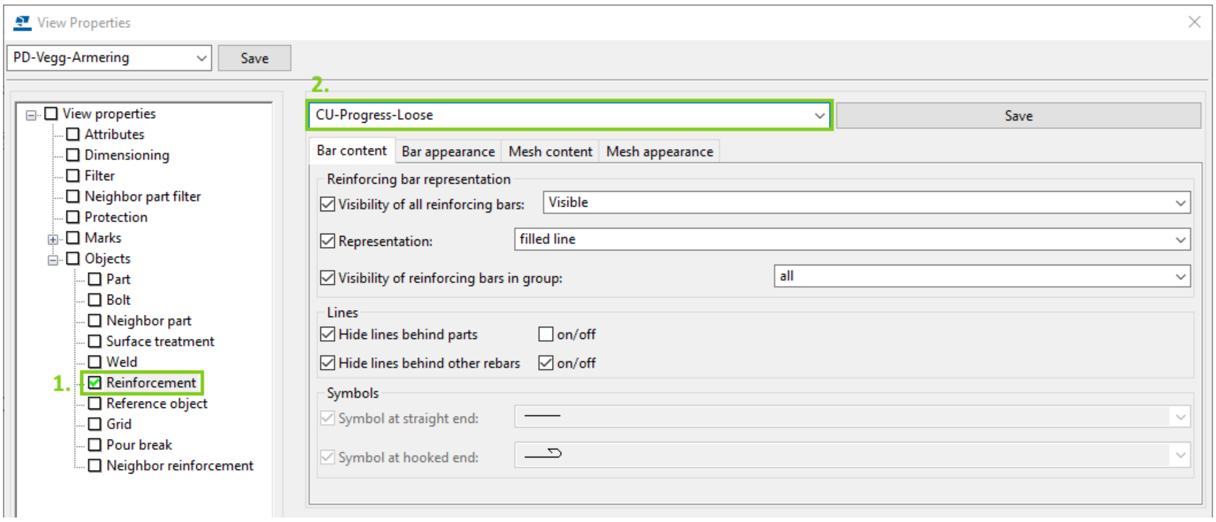
The automatic reinforcement marks in drawings can be changed to use the PXML grouping. Rules for identifying meshes, cages and loose rebars are the same as for grouping in the bending schedule.



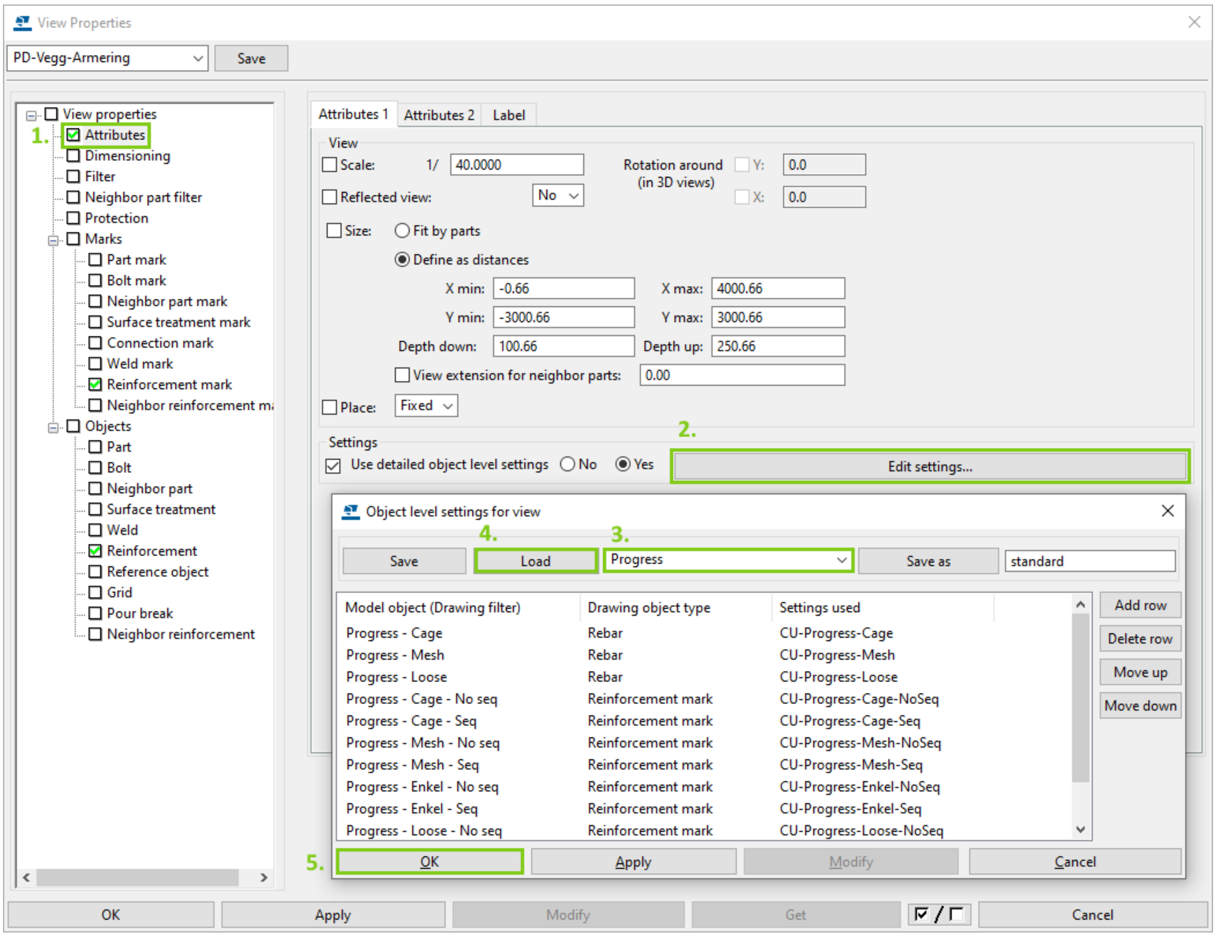
Open properties of the reinforcement view or views, go to the `Reinforcement mark` tab and load the `CU-Progress-Loose-NoSeq` settings.



Now go to the `Reinforcement` tab and load the `CU-Progress-Loose` settings.

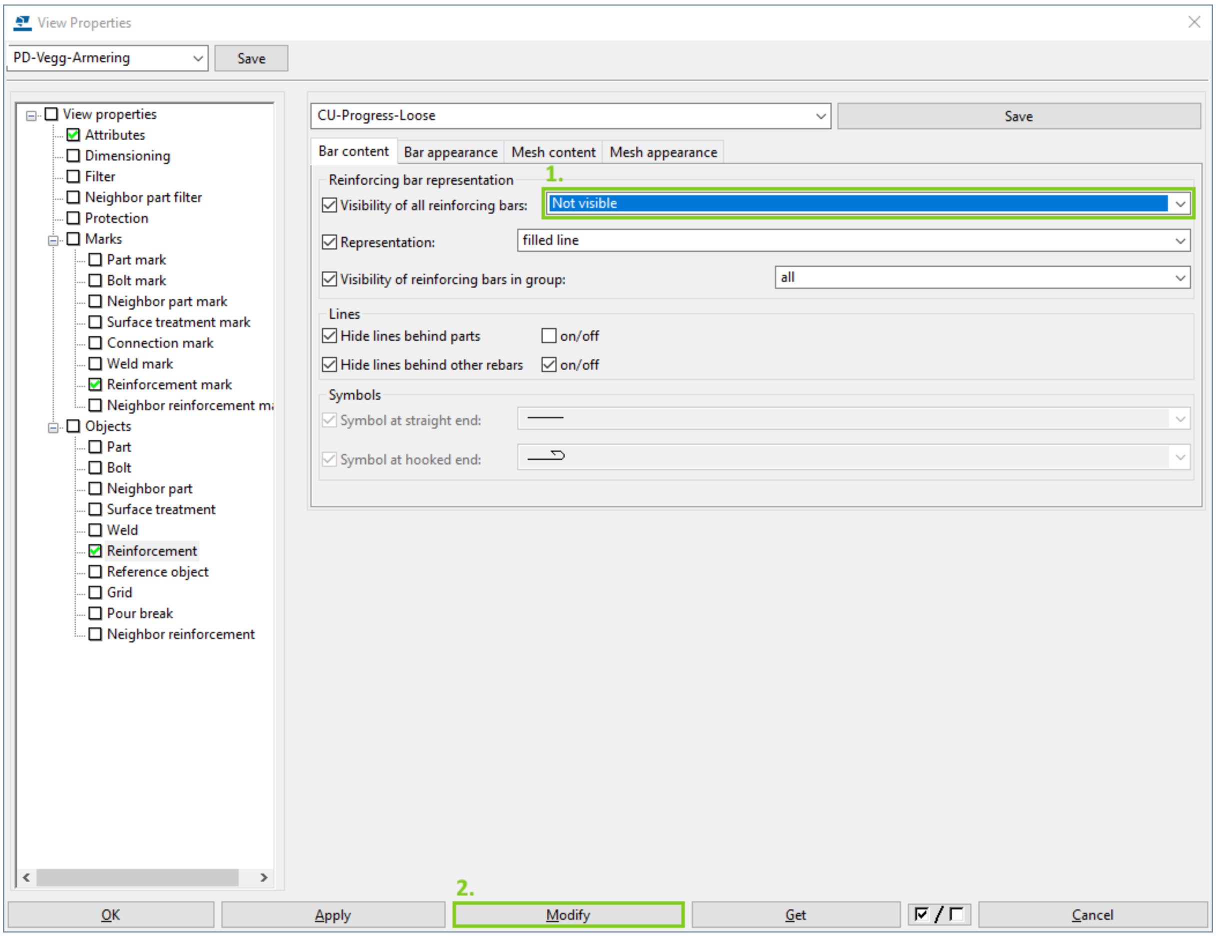


Then go to the `Attributes` tab, click `Edit settings…` button, load `Progress` and click `OK`.



Finally click the `Modify` button to put the changes in effect.

If some marks are not updated correctly, open back the view settings and go to the `Reinforcement` tab. Change the option `Visibility of all reinforcing bars` to `Not visible` and click `Modify`. Then change it back to `Visible` and click `Modify` again.



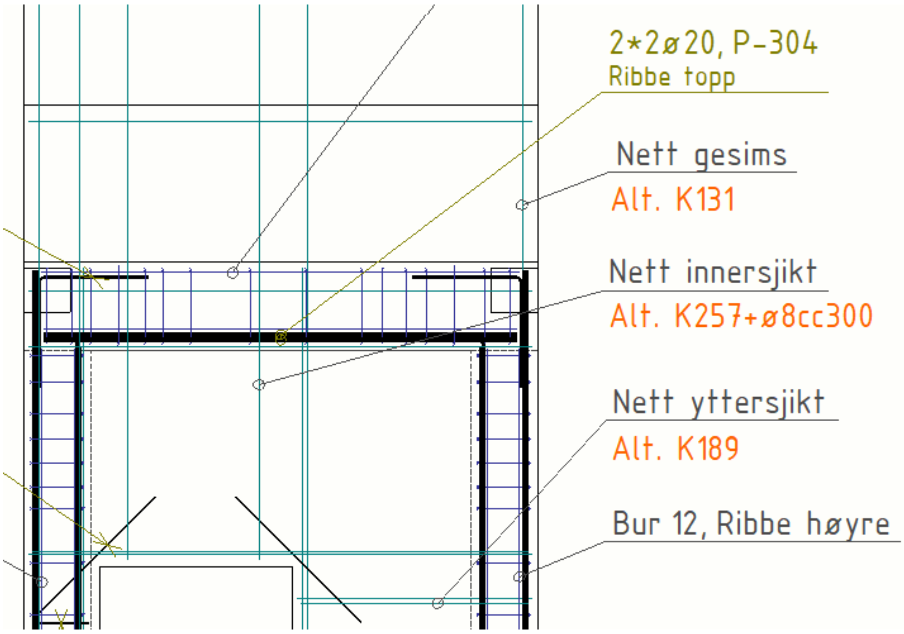
Backup descriptions for manual production

At times the machine may not operate due to a maintenance, software problem, or a broken part. When the issue is difficult to fix it may take a few days before the production is restored. In principle this kind of delays cannot be afforded in element production and designers should take additional precautions to simplify reinforcing the elements manually when such a situation occurs.

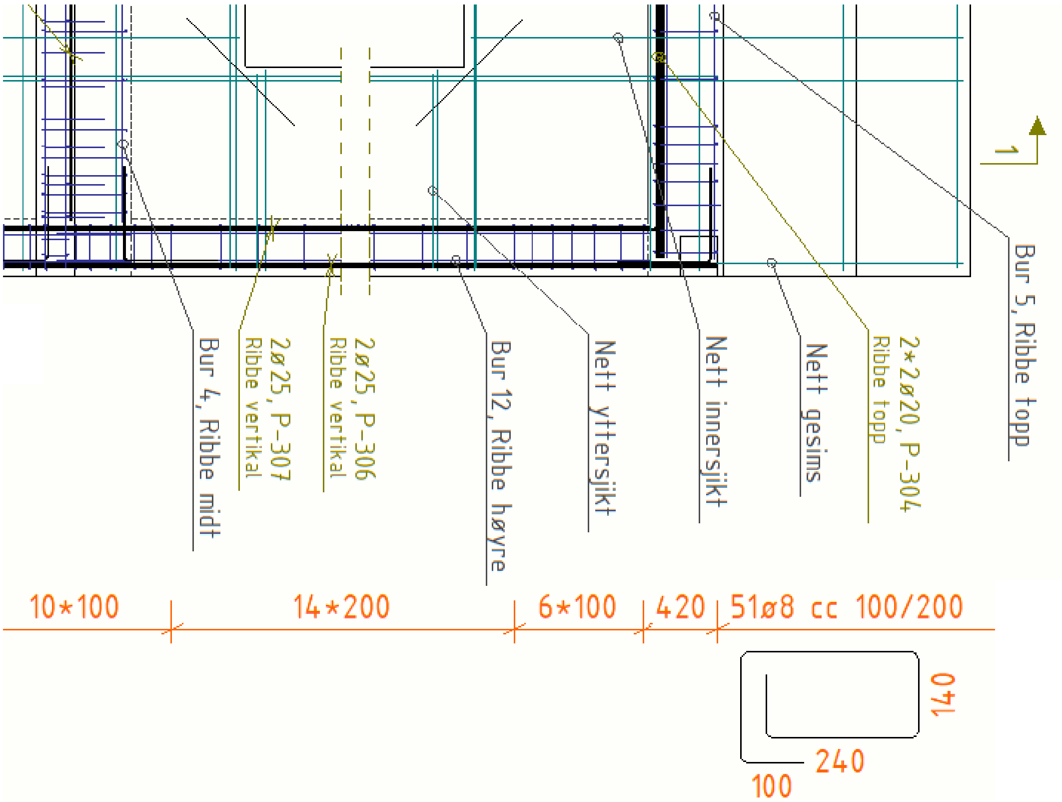
In general, the manual production will be very rare, and thus the time required to include the alternative descriptions should be reduced to minimum. The reinforcement should therefore be modelled first and foremost optimized for the machine production, and the descriptions of the manual variant should be simplified compared to the actually modelled reinforcement.

The backup descriptions should be clearly separated from the standard descriptions, for example by using a different font color.

Every unique mesh should have a description of catalog mesh and optional extra rebars that it can be replaced with. The dimensions can be inferred from the form drawing.



Every unique cage should have a description of main rebars, stirrup form (either in bending schedule or as a pullout) and stirrup spacings. Spacing can be shown on a manual, simplified dimension.

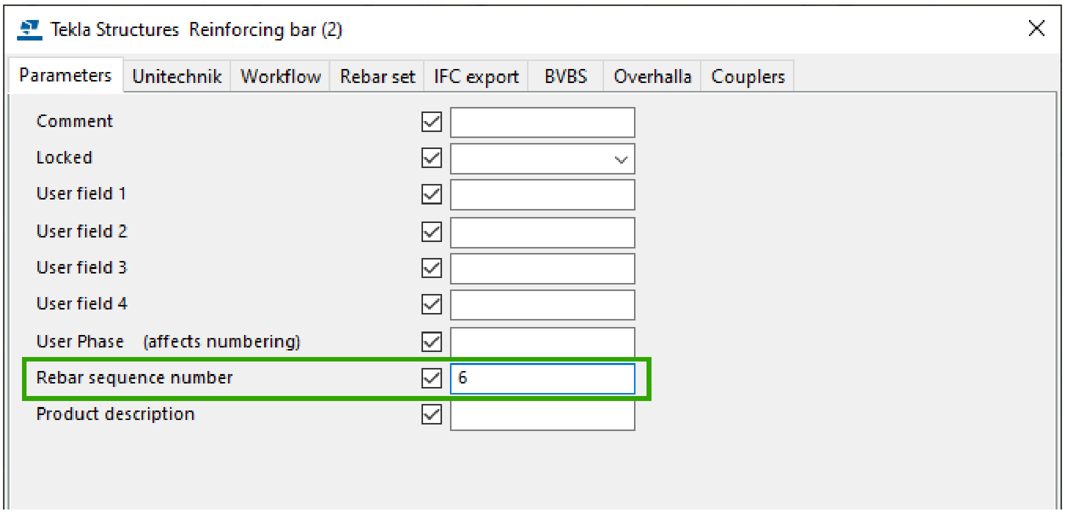


<u>Tip:</u> Dimensioning of the manual variant can be simplified by modelling the ribs first using even spacings between the stirrups and adjusting the positions of wires in the mesh instead.

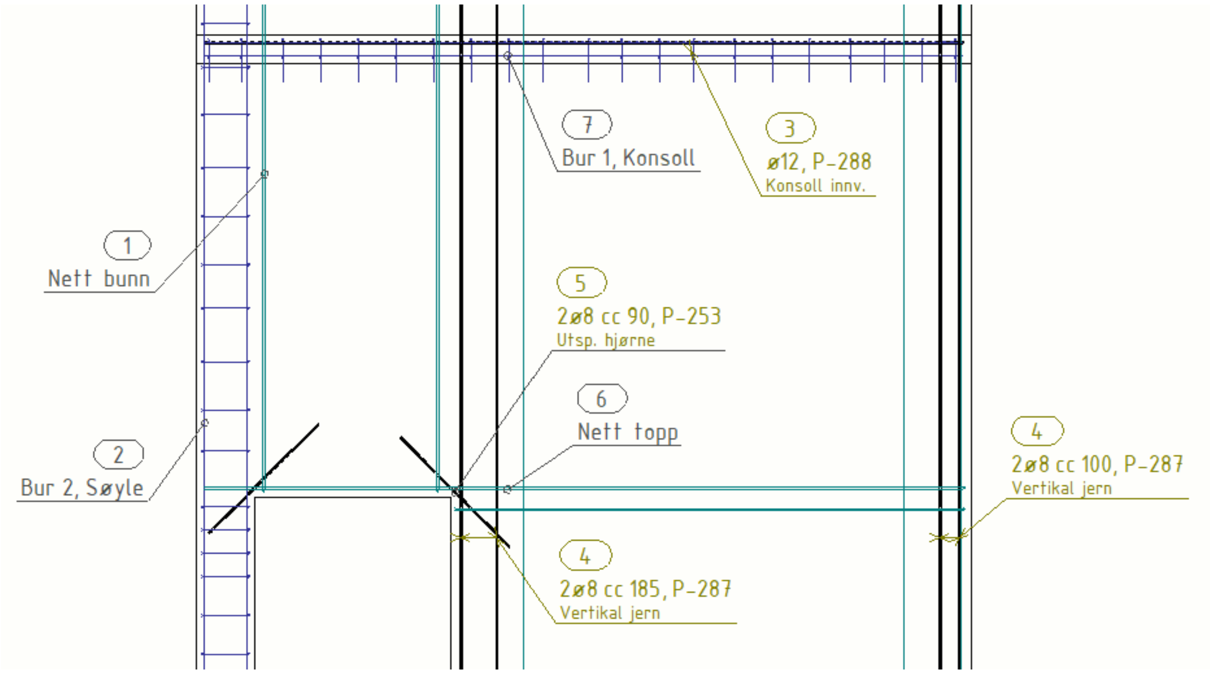
Loose reinforcements should already be described and require no additional information.

Showing the production sequence

The parts of reinforcement can be assigned `Rebar sequence number` to indicate order in which the reinforcement should be completed. The property can be defined for some or all rebars in the User-defined attributes dialog.



Once the `Progress` object-level settings are [applied for the drawing view](), the reinforcement marks will show the sequence number for rebars where it was set.



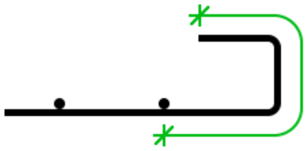
Technical data

BlueMesh mesh welding machine

Machine type	Progress M-System BlueMesh
Mesh width	500 mm – 4 600 mm
Mesh length	1 000 mm – 10 000 mm
Bar diameters	ø6, ø8, ø10, ø12, ø16
Max thickness	ø16 + ø12[1]
Bar length	350 mm – 10 000 mm
Overhang length[2]	25 mm – 600 mm
Bar horizontal angles	0° or 90°
Min welds per bar	2 weld points
Cross wire spacing[3]	Rastered[4]: min 32 mm (clearance) – recommended max 500 mm Gridless[5]: min 32 mm (clearance) – max 450 mm
Line wire spacing[3]	Rastered[4]: min 25 mm Gridless[5]: min 50 mm
Line wire grid[4]	25 mm
Bending angles	0 – 180°
Bending dies	D25, D32, D40, D48, D64
Bar length tolerance	± 5 mm
Bar placing accuracy	± 10 mm

[1] Welding ø16 on ø16 wires is also possible but with reduced speed because of the cooling capacity. The wear of electrodes is increased, and use of this combination should be limited.

[2] Length of the bar including all bendings past the weld point of the last perpendicular bar. The max does not apply to the trailing end of the mesh, where longer overhang is producible.



[3] Clearance is the distance between the edges of wires, while spacing is measured center to center. Unless noted otherwise, the specified distances are spacings.

[4] Production rastered (with grid) requires spacings of longitudinal wires to be a multiple of 25 mm. It is much more stable than gridless and should be used whenever possible. Recommended max clearance can be exceeded but requires special attention from the operator.

[5] Production gridless allows arbitrary longitudinal wire spacings in increments of 1 mm but requires bigger minimum of 50 mm. Line wire spacing below min or cross wire spacing above max forces rastered production and automatically shifts wires into the grid.

Hambi mesh bender

Machine type	Hambi MBH 1001 VS
Max mesh width[1]	4 600 mm
Max mesh length[1]	10 000 mm
Bent bar diameters	ø6, ø8, ø10, ø12
Corner bar diameters[2]	ø6, ø8, ø10, ø12, ø16
Min bend to bar distance[2]	60 mm
Bending dies	D40 cc 150 mm
Bending capacity	200ø6, 130ø8, 100ø10, 100ø12
Bent segment length	Middle segment[3] without corner bars: min 120 mm End segment[3] or middle segment with corner bars: min 110 mm

[1] The overall mesh weight should also be considered, but now a dedicated crane is available.

[2] Bendings with corner bars can be produced. Longitudinal bars not placed directly in the corners must be placed at the specified min distance from bent bar.

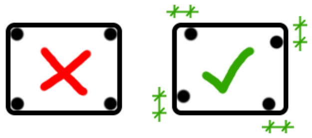
[3] Middle segment has bendings in both sides. End segment is either the first or the last bent part.

MEP mesh bender

Machine type[1]	MEP PRE 1226 A
Max mesh length	6 000 mm
Bent bar diameters	ø6, ø8, ø10
Corner bar diameters[2]	none
Min bend to bar distance[2]	40 mm
Bending dies	D17 cc 150 mm

[1] The MEP mesh bender is now replaced with Hambi for the majority of production.

[2] Bars cannot be placed directly in the corners and specified min distance from bent bar is required.



Pluristar stirrup bender

Machine type	Progress Pluristar 16: BaMD/3D + 2BK_MA
Bar diameters[1]	ø8, ø10, ø12, ø16
Bar length[1]	50 mm – 12 000 mm
Bending dies	BaMD/3D: D20, D25, D32, D50, D35 (circling) 2BK_MA: D20, D32, D60, D80, D100, D20 (cranking)

[1] Bars of bigger diameter or length are produced manually. Available diameters are ø20, ø25, ø32. Maximum length depends on diameter, order quantity and availability from supplier.

B26 bar bender

Machine type[1]	VM B26
Bar diameters	ø8, ø10, ø12, ø16, ø20, ø25
Bending dies	D60

[1] The manual bar bender is used for making simple bends of bars not available in Pluristar.