Erasmus+ Project 2022-1-NO01-KA220-HED-000087893

This Erasmus+ Project has been funded with support from the European Commission. This publication reflects the views only of the authors, and the European Commission and Erasmus+ National Agencies cannot be held responsible for any use which may be made of the information contained therein

BIM-LCA Construction Project

Title: Tekla software tutorials. CS: Industrial Building

Tekla tutorials provide valuable guidance and instruction for learning how to efficiently utilize Tekla Structures, a powerful software solution for structural engineering and detailing. These tutorials offer step-by-step demonstrations and explanations, helping users master essential skills and techniques for modeling, detailing, and managing construction projects with Tekla. By following Tekla tutorials, individuals can enhance their proficiency in creating accurate 3D models, generating construction drawings, and effectively communicating design intent within the industry-standard Tekla environment. Additionally, these tutorials enable users to stay updated with the latest features and functionalities of Tekla Structures, ensuring they can leverage the software to its full potential for successful project outcomes.

1 – Aims

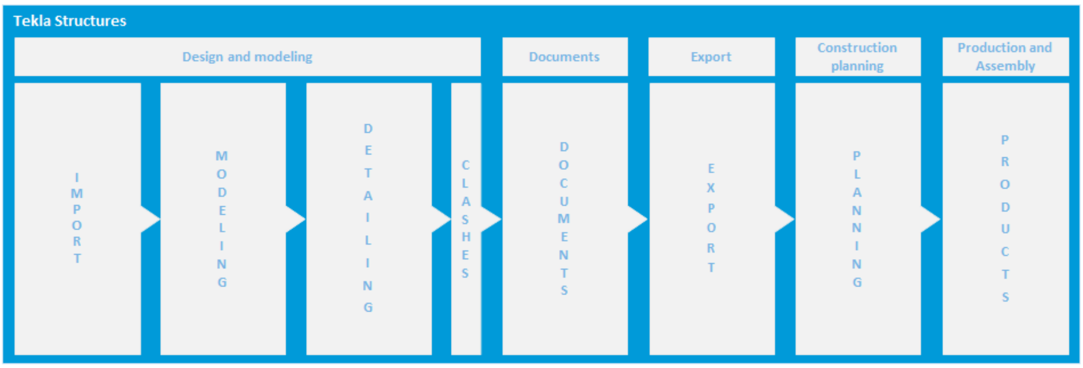
The objectives of this tutorial are as follows:

The objective of BIM is to leverage digital technologies to create a collaborative, data-driven approach to building design, construction, and management. By using BIM, stakeholders can achieve improved project outcomes, including cost savings, schedule adherence, better quality, and sustainable building practices throughout the building's lifecycle.

Interoperability is crucial for BIM (Building Information Modeling) structures when using software like Tekla because it ensures seamless collaboration, data consistency, and efficient workflows throughout the construction lifecycle.

Multi-disciplinary Collaboration: BIM projects involve various disciplines such as architecture, structural engineering, MEP (mechanical, electrical, plumbing), and construction management. Interoperability allows different teams using different software (like Tekla for structural modelling and other BIM tools for architecture or MEP) to exchange data accurately and efficiently. This enables better coordination and integration of design and construction elements.

Interoperability enhances workflow efficiency by enabling automated data exchange and synchronization between software applications. For instance, Tekla's interoperability with other BIM tools allows for seamless import and export of models, drawings, and schedules, streamlining tasks like clash detection, quantity take-off, and construction sequencing.



1.2 – Tekla Structures

Tekla software is an advanced structural BIM software for construction.

Structural engineers, designers, detailers, fabricators, contractors and project managers can rise beyond traditional limits on every stage of construction. With Tekla Structures, they can create, combine, manage and share information with remarkable efficiency.

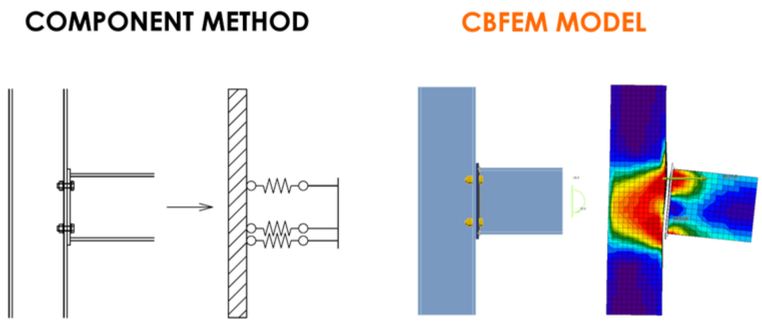
Tekla software offers everything that is needed to improve BIM accuracy, utilize data, and reduce costly surprises. It will enhance the profitability with the highest level of development (LOD) and reduce the uncertainty of uncoordinated construction documents.

Is simple to import, export and link your model data with other project parties, software, digital construction tools, and machinery for smoother workflows.

1.3 – Idea StatiCa Steel Connections

Idea StatiCa is a patented software designed for the analysis and structural design of steel connections. It excels in handling various types of connections, including welded and bolted joints, plates, footings, and anchors. Furthermore, it allows for the evaluation of buckling effects on steel components.

The Component-based Finite Element Method (CBFEM) that is based on, effectively combines all the code equations and conditions with finite elements, overcoming the topology and loading limits of the old methods.

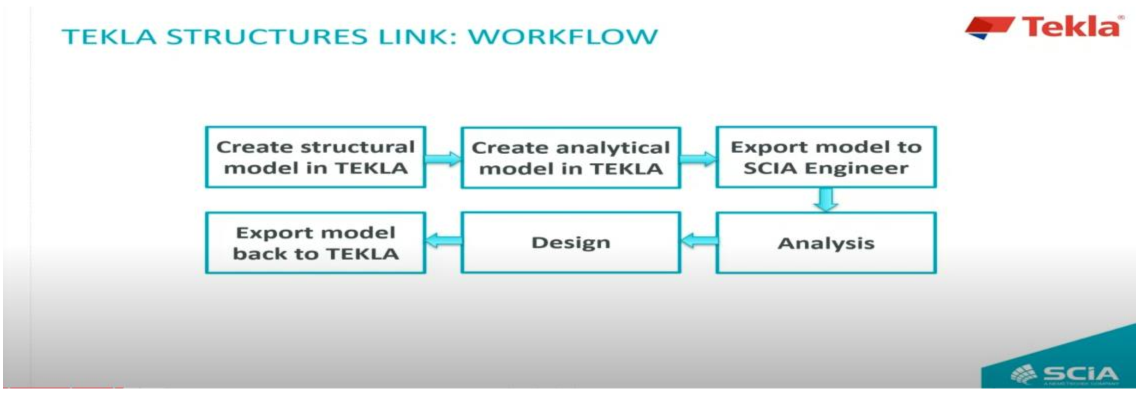


1.4 – Interoperability between SCIA Engineer, Tekla Structures and Idea StatiCa

1.5 – SCIA Engineer integration with Tekla Structures

SCIA and Tekla are both part of buildingSMART alliance's OpenBIM initiative and promote IFC as the preferable format for data exchange of 3D structural models. In addition, SCIA Engineer offers a bi-directional link that makes it easy exchange of steel models.

TEKLA TO SCIA ENGINEER:



SCIA Engineer offers a seamless workflow for modeling, analysing, and optimizing steel structures and components. It allows for easy integration with Tekla, enabling efficient final documentation and detailing. The software supports both Open BIM (based on IFC format) and Closed BIM (proprietary links) interoperability options. One such example is the Tekla Structures link, facilitating smooth model transfer between Tekla and SCIA Engineer.

This bi-directional link is compatible with the last two versions of major releases for both platforms, allowing for simultaneous updates from either side. Users have the choice of direct transfer for real-time collaboration or file export for sharing with colleagues. The link also provides the flexibility to transfer the entire model or specific parts, such as steel or concrete components. Progress can be monitored through a dialogue window, and a comprehensive transfer report can be generated and saved.

Additionally, users can customize national standards for materials and cross-sections in SCIA Engineer, with the chosen settings preserved during the transfer process. The link supports mapping of materials and cross-sections between projects in both applications. It also accommodates parametric steel profiles, ensuring accurate representation. The mapping tables created are project-specific and stored for future use.

The current link capabilities include transferring 1D and 2D elements, constraints/support, hinges, rigid links, as well as exporting and importing reinforcement details for beams and columns between SCIA Engineer and Tekla. This integration significantly streamlines the workflow for structural engineers and designers

The teacher will give an explanation about modelling of a steel structure by using tekla in about 2h.

Students will read this tutorial and follow the steps shown in the tutorial, namely:

- Knowing the interface and operation of Tekla Structures.
- Introduction of structural elements.

In order to evaluate the success of the application, students will prepare a report on the steps taken in practice, difficulties faced and decisions taken.

3 - Tutorial duration

The implementation described in this tutorial will be carried out through the Tekla software.

4 – Necessary teaching recourses

Computer room with PCs/laptop with internet access.

Required software: Tekla

5 - Contents & tutorial

5.1 Tutorial

In this tutorial, I will present the basic steps for creating a steel frame plan using Tekla Structures software. A steel frame refers to steel structures used in construction, such as columns, beams, and supporting elements. Tekla Structures is a powerful tool for designing and detailing these structures. We will explore how to model and generate plans for a simple steel frame using Tekla.
While Tekla BIM is emphasized, it is necessary to use other programs for building design prior to drawing. Therefore, we provide a brief tutorial in SCIA for the design aspect, along with an extended tutorial in Tekla Structures.

To enhance our understanding of BIM, we will engage in a learning activity that simulates a scenario within a structural engineer's office at a local level. This exercise will involve the seamless exchange of information between several software applications: Scia Engineer, a tool for structural analysis and design; Tekla Structures, used to create comprehensive 3D models of steel structures such as portal frames; and Idea StatiCa, which provides precise assessments including strength, stiffness, and buckling analyses of steel joints.

Our objective is to perform structural calculations for an industrial hall. The construction system comprises steel portal frames interconnected longitudinally with metal beams and braced in both the walls and roof planes. The perimeter enclosures will feature 10cm vertical thermal insulation panels. The foundation consists of elastic isolated foundations with reinforced concrete blocks, while the superstructure includes flat metal frames with an opening span of 22.00m, provided at 5.0m intervals.

Participants are expected to have basic familiarity with the functions of these programs, having completed beginner tutorials available on the respective producers' websites. This exercise will enable us to explore the practical application of BIM tools in a real-world structural engineering context.

5.1.1. Model design

In many portal frames, enhancing the resistance of the rafter at the eaves is achieved by incorporating haunches, which are tapered sections of the rafter. The inclusion of haunches not only augments the overall stiffness of the frame but also has the potential to reduce displacements.

Portal frames are two-dimensional rigid structures characterized by a fixed joint between the column and beam. The main objective of this form of design is to reduce bending moment in the beam, which allows the frame to act as one structural unit.

The elastic theory serves as the prevalent foundation for analyzing general structures. Under the application of load, these structures maintain their elasticity, ensuring that load paths remain consistent regardless of load magnitude, and deflections are directly proportional to the load.

In this model, beam elements are represented by lines, which denote the axes of the members. It's crucial that these lines pass through the centroid of the cross-sections of the beams and columns. Consequently, the effective span length of the portal frame is determined by the distance between the centroid axes of the columns.

In many portal frames, enhancing the resistance of the rafter at the eaves is achieved by incorporating haunches, which are tapered sections of the rafter. The inclusion of haunches not only augments the overall stiffness of the frame but also has the potential to reduce displacements.

The following dimensions have been considered for structural design of portal frame:

- Span: L = 22 m

- Height: H = 7.4 m

- Bay: B = 5 m

- Roof slope: 6°

- Column: HEA 400

- Beam: IPE 400

Step 1. Starting a new project:

After opening the program in project settings, you'll define general data such as the name, type of structure, select your material, and specify the national code and annex.

Step 2. Input of the geometry

S.2.1. Cross-sections: When entering one or more 1D members, a cross-section is immediately assigned to each member. By default, the active cross-section is represented. You can open the profile library to activate another cross section.

S.2.2. Geometry: You can use single columns and beams to enter the frame,but SCIA Engineer offers as well multiple Catalog blocks, allowing for a smooth and simple input of the structure.

S.2.3. Additional data: The structure is completely set up. Now, we can finish the geometry input by adding end conditions, enter haunches, hinges and supports.

S.2.3.1. Haunches: In this SCIA Engineer project, each member is modelled as prismatic with a constant cross-section, unless a haunch is specified. Haunches have been incorporated into the design for the roof beams. These haunches are characterized by two key parameters: a cross-section with a variable height and a specified length, over which the height can vary by up to 0 units. The selected cross-section combines elements of both an I-section and a variable section, denoted as I + I var.

S.2.3.2. Hinges: In SCIA Engineer, every node where two or more members connect is regarded as fixed, until a hinge is entered and some translations and/or rotations are released. The geometry input can be completed with supports. The column bases are modelled with pinned hinges that allow rotation without transmitting moments.

Step 3. Check structure

After input of the geometry, the structure is checked for duplicate nodes, zero beams, duplicate members, wrong references of hinges or supports.

Step 4. Load cases and load groups

Each load is attributed to a load case with properties which are determinant for the automat generation of combinations. The action type of a load case can be permanent or variable.

Each load case is associated with a load group. The load group contains information about the category of the load (service load, wind, snow) and its appearance (default, together, exclusive). In an exclusive load group, the different load cases attributed to this load group cannot act together in a single combination when using envelope combinations or code combinations.

Load cases:

GROUP	NAME
Dead group	LC1-Self weight

LC2-Permanent: 0.8kN/m

Snow group LC3-Snow: 1.2kN/m

Wind group LC4-Wind

Seismic group LC5-Seism

Seismic spectrum:

INFO DRAWING

Type code – Romanian standard

Spectrum type -Horizontal

City – Cluj-Napoca

Gamma – importance factor - 1

coeff accel. ag - 0.1

ag–nominal acceleration – 0.981

TB - 0.14 / TC - 0.7 / TD - 3

beta0 - 2.5

q behaviour factor – 2.5

Mass groups:

NAME	LOAD CASE	DESCRIPTION
MG1	LC1 - Self weight	Self-weight mass
MG2	LC2 - Dead	Dead mass
MG3	LC3 - Snow	Snow mass

Wind load: While Scia Engineer offers an integrated 3D wind function, for our 2D structural analysis, we derived the wind forces and applied them as linear forces on the respective elements.

Combinations: Two automatic code combinations are created, one for the Ultimate Limit State and one for the Ultimate Serviceability State.

Step 5. Linear analysis:

Once the calculation model is fully prepared, proceed to initiate the calculation process. Ensure that all entities are properly interconnected and that the mesh setup is activated. After the analysis, a notification window will confirm the completion of the calculation, providing maximum deformation and rotation values for the normative load case.

Step 6. Results

S.6.1. Reactions

Linear calculation, Extreme: Global

Selection: All

Class: ULS class

SUPPORT	CASE	Rx[kN]	Rz[kN]	My[kNm]
Sn2/N1	ULS-Set B(auto)/1	81.64	177.98	0.00
Sn2/N1	ULS-Set B(auto)/2	-7	22.14	0.00
Sn1/N5	ULS-Set B(auto)/1	-81.64	177.98	0.00

S.6.2. Internal forces on member

Linear calculation, Extreme: Global

Selection: All

Class: ULS class

MEMBERS	CASE	N [kN]	Vz [kN]	My [kNm]	
B1	ULS-Set B (auto)/1	-15,91	-13,54	-21,32	
B1	ULS-Set B (auto)/2	-177,98		-81,64	0
B1	ULS-Set B (auto)/1	-22,14	7	0	
B1	ULS-Set B (auto)/2	-169,57		-81,64	-532,28
B1	ULS-Set B (auto)/1	-19,92	-0,34	7,76	
B2	ULS-Set B (auto)/1	-9,85	-6,37	39,55	
B2	ULS-Set B (auto)/3	-80,51	-11,82	262,98	
B2	ULS-Set B (auto)/2	-98,11	160,61	-516,01	
B2	ULS-Set B (auto)/2	-91,37	-2,03	302,86	

B3	ULS-Set B (auto)/1	-10,92	4,29	39,55
B3	ULS-Set B (auto)/2	-91,08	-9,11	300,28
B3	ULS-Set B (auto)/2	-98,11	160,61	-516,01
B3	ULS-Set B (auto)/2	-91,37	-2,03	302,86
B4	ULS-Set B (auto)/1	-26,17	13,58	118,86
B4	ULS-Set B (auto)/2	-177,98	81,64	0
B4	ULS-Set B (auto)/2	-169,57	81,64	532,28

Axial force N:

Shear forces Vz

Moment My:

S.6.3. 3D deformations

Linear calculation, Extreme: Global

Selection: All

Class: SLS class

Step 7. Code checks

The steel modules include a number of tools to perform steel calculations in accordance with the chosen design code. The possibilities are as following:

• input of steel data per member;

• input and manipulation of buckling data;

• input of stiffeners, lateral-torsional buckling restraints, steel sheeting, …;

• performing a ULS unity check;

• optimization of the cross-section;

• performing a SLS unity check;

• performing a fire-resistance check;

• input, calculation and creation of drawings for connections;

For additional details on advanced steel calculations, such as 2nd order analysis and fire-resistance checks, you can refer to the Advanced Steel Training provided by the program's producer.

After performing an elastic analysis on a single-storey structure, it's imperative to verify the frame members, considering both cross-sectional resistance and member buckling resistance, commonly referred to as member stability. The steel member design process should strictly adhere to the guidelines outlined in SR EN 1990 [ref] and SR EN 1993-1-1 [ref].

S.7.1 Buckling parameters

The columns and rafters of portal frames are subject to combined axial force and bending moments. Consequently, the member verifications involve in/out of plane flexural buckling resistance, the lateral-torsional buckling resistance and the member resistance under combined axial force and bending. The secondary components (purlins and rails, flying braces, longitudinal beams) are used to provide intermediate restraints, to reduce the length of segments, increasing both the flexural and lateral-torsional buckling resistance.

Prior to performing steel code checks, it is essential to assign the buckling parameters for the rafter in relation to the position of purlins.

S.7.2 Steel code check

The ULS (Ultimate Limit State) unity check includes both a section and a stability check.

The detailed report following the Ultimate Limit State (ULS) checks revealed that the column does not meet the Combined Bending, Axial Force, and Shear Force check in accordance with EN 1993-1-1. SCIA Engineer allows for a simple and smooth optimization of the steel section, whether it does not satisfy or whether it is too "heavy" and overdesigned. The program automatically suggests a cross-section that satisfies the unity check; in our scenario, a HEA320 section was recommended.

After performing both Ultimate Limit State (ULS) and Serviceability Limit State (SLS) checks, which include the comparison of relative deformations with predefined deflection limits set either in the steel settings or via the system lengths and buckling settings, an IPE400 beam with a haunch with a height of 365mm and a length of 2.7m was chosen.

Step 8. Export to Tekla

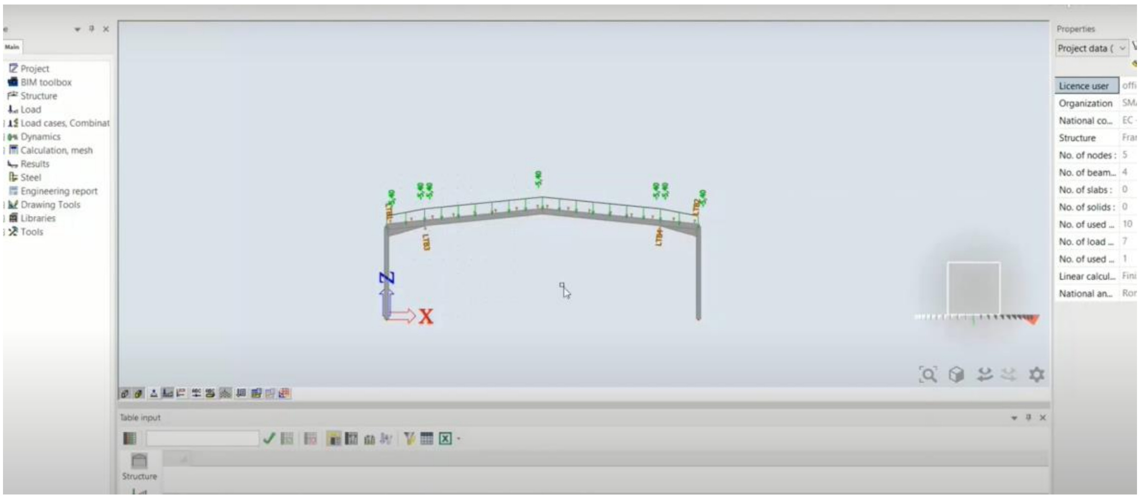
- Open the ESA file in Scia Engineer.
- Follow the steps indicated in the provided picture.
- Save the file in s2t (Scia to Tekla) format for export.
- Import the S2t in Tekla Structures

When exporting files from Scia Engineer to Tekla Structures, the haunches are not recognized. In Tekla Structures, haunches are considered components of joints.

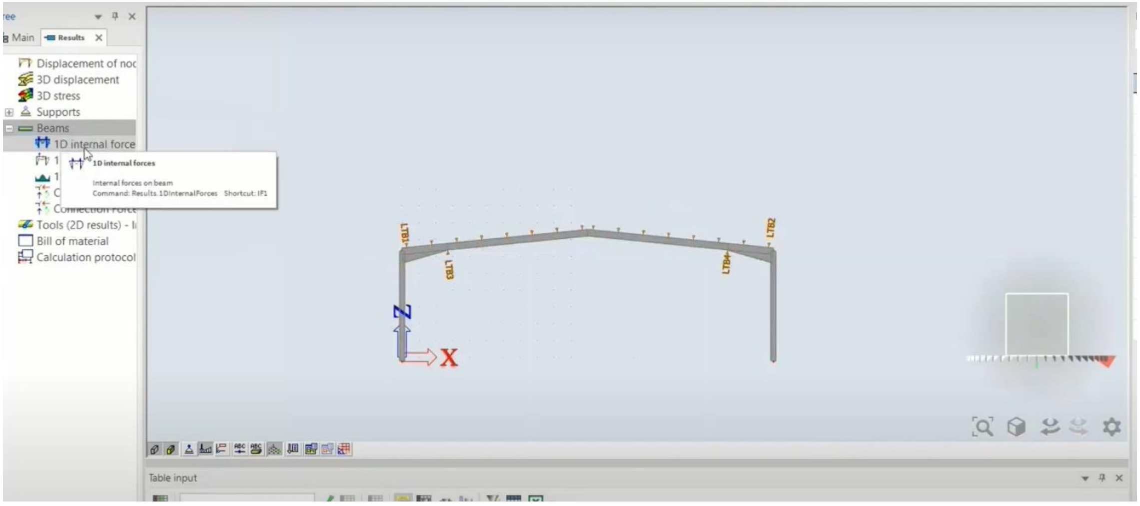
5.1.2 SCIA ENGINEERING for the design part

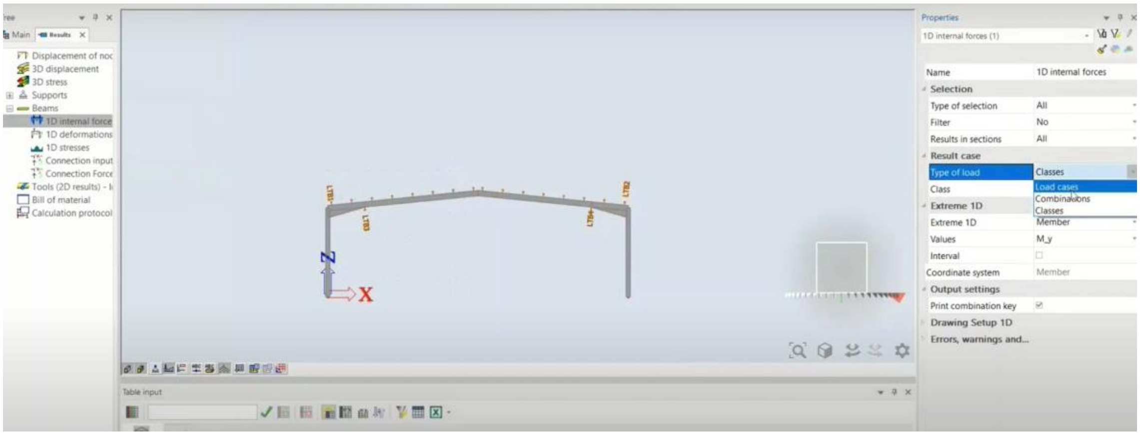
The beams, walls and slabs of the second floor are also drawn similarly

Open the ESA file in Scia Engineer: defining the structure

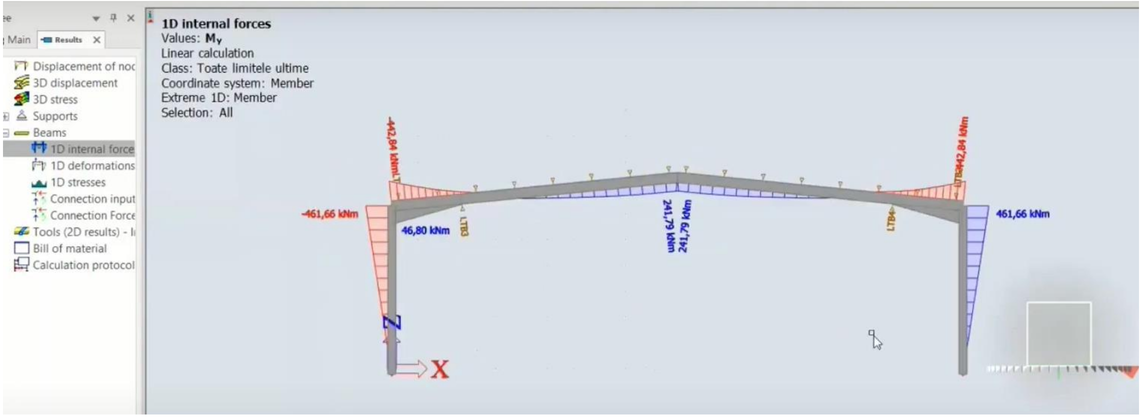


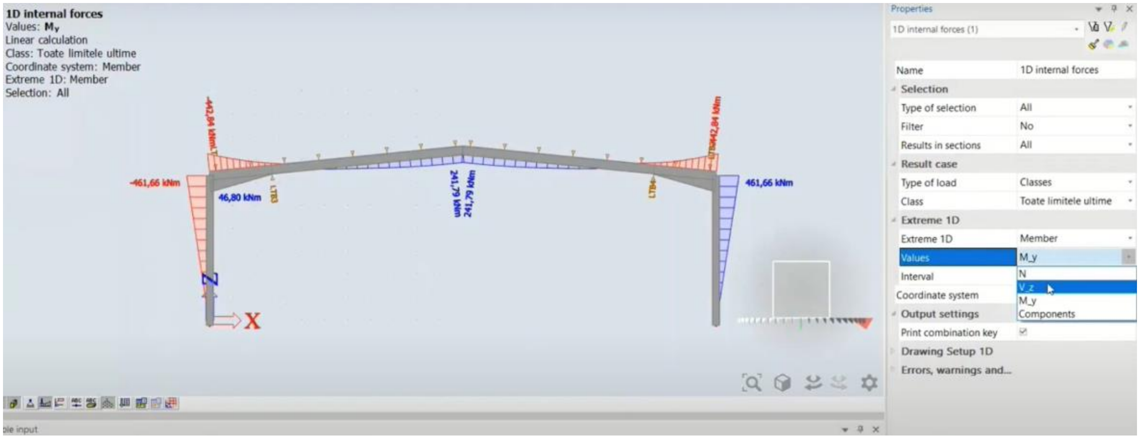

Main
Results
Displacement of noc
3D displacement
3D stress
Calculation protocol
3D stress
Command: Results.Stress Shortcut: sss

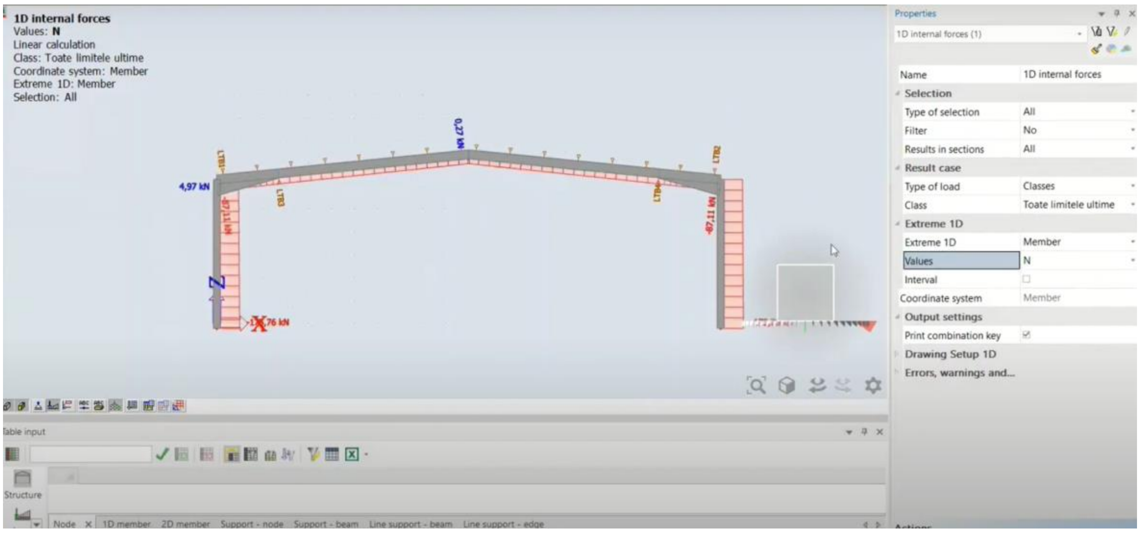

Main
Results
Displacement of noc
3D displacement
3D stress
Supports
Beams
1D internal force
1D internal forces
Internal forces on beam
Command: Results.1DInternalForces Shortcut: IF1
Tools (2D results)
Bill of material
Calculation protocol
Table input


Main
Results
Displacement of noc
3D displacement
3D stress
Supports
Beams
1D internal force
1D deformations
1D stresses
Connection input
Connection Force
Tools (2D results)
Bill of material
Calculation protocol
Properties
1D internal forces (1)
Name
1D internal forces
Selection
Type of selection
All
Filter
No
Results in sections
All
Result case
Type of load
Classes
Class
Load cases
Combinations
Classes
Extreme 1D
Member
Values
M_y
Interval
Coordinate system
Member
Output settings
Print combination key
Drawing Setup 1D
Errors, warnings and...
Table input

Tekla software tutorials. CS: Industrial Building.

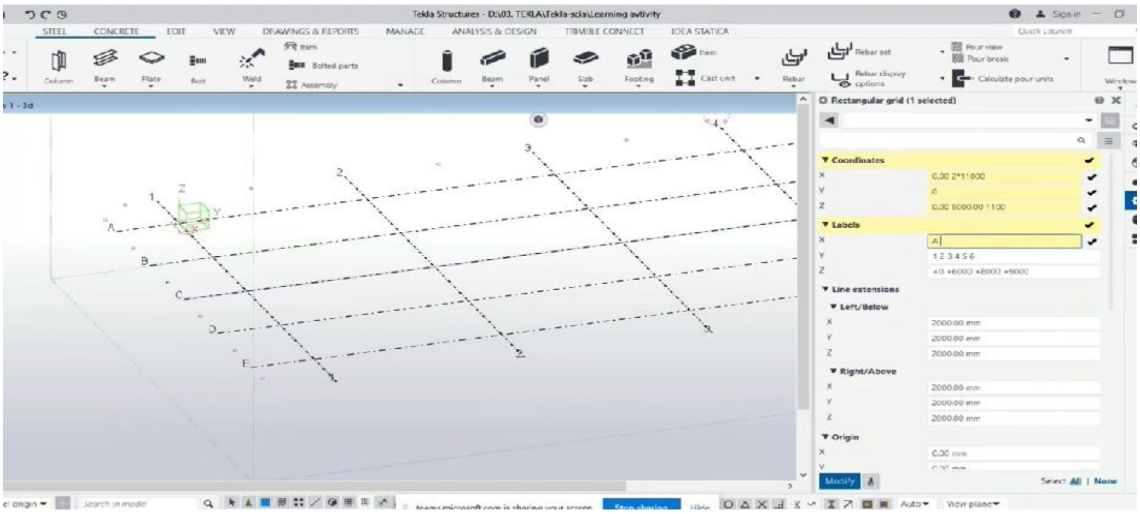


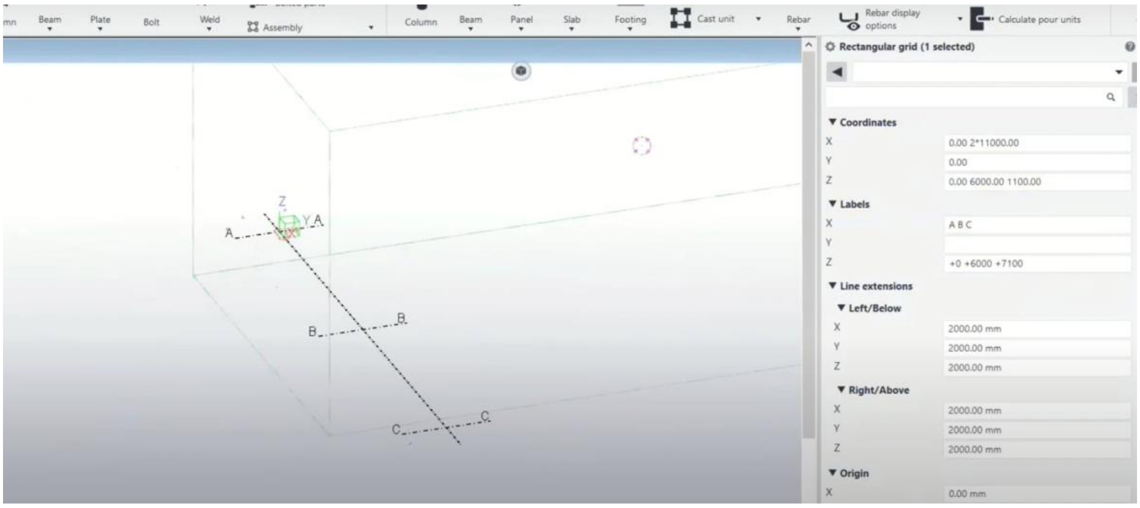


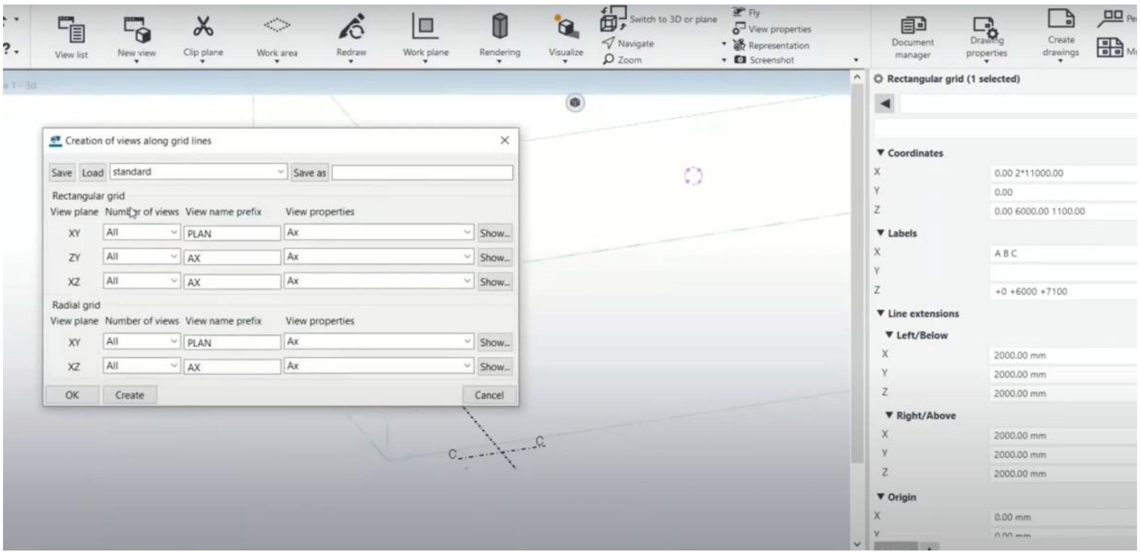


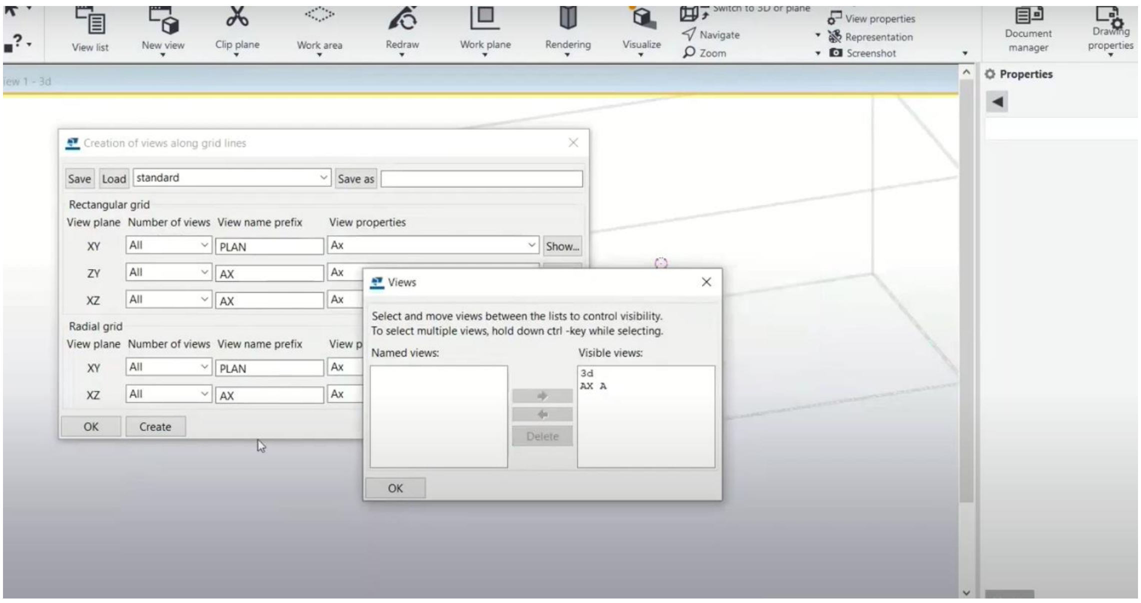
5.1.3 TEKLA STRUCTURES TUTORIAL for the drawing part

Drawing coordinate axes



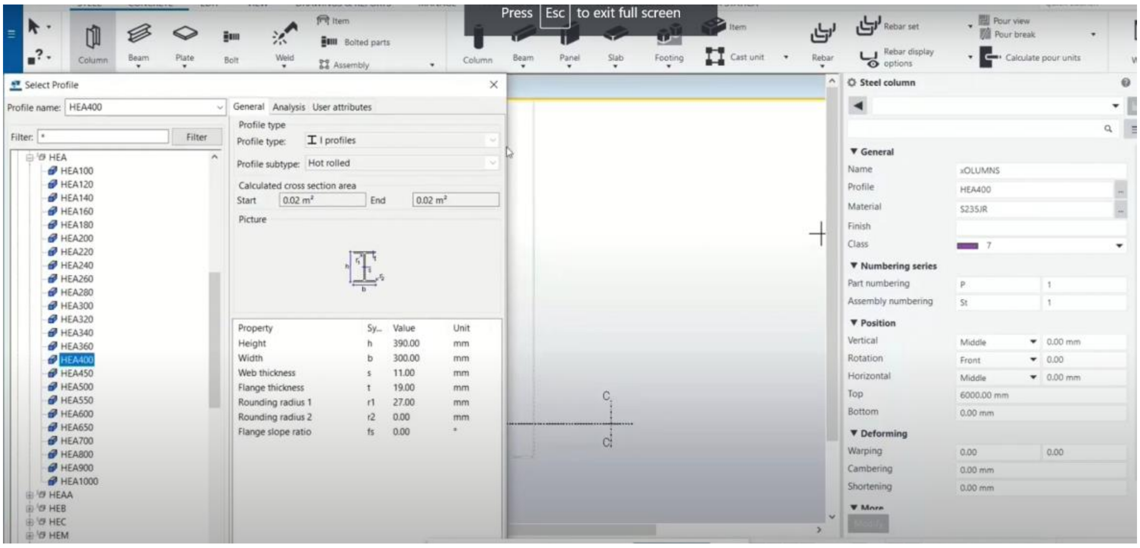


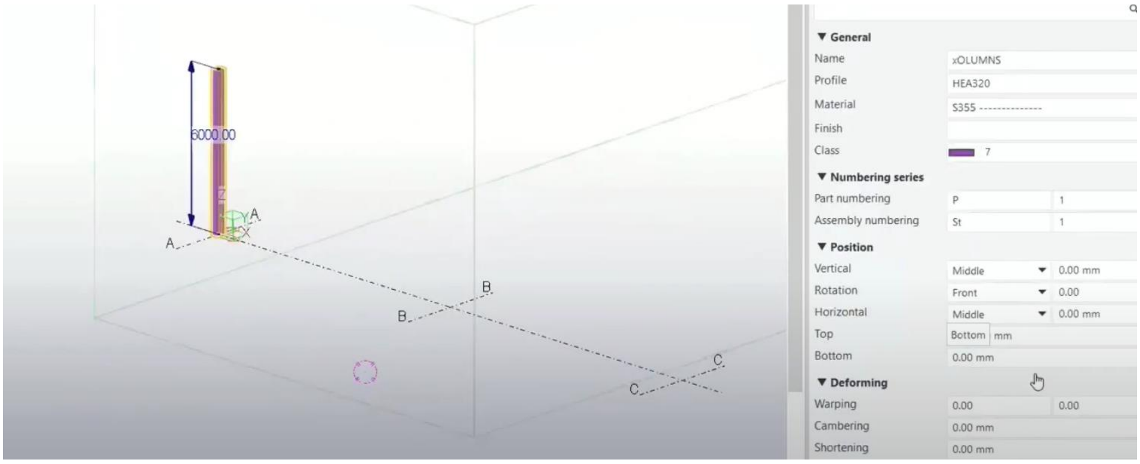




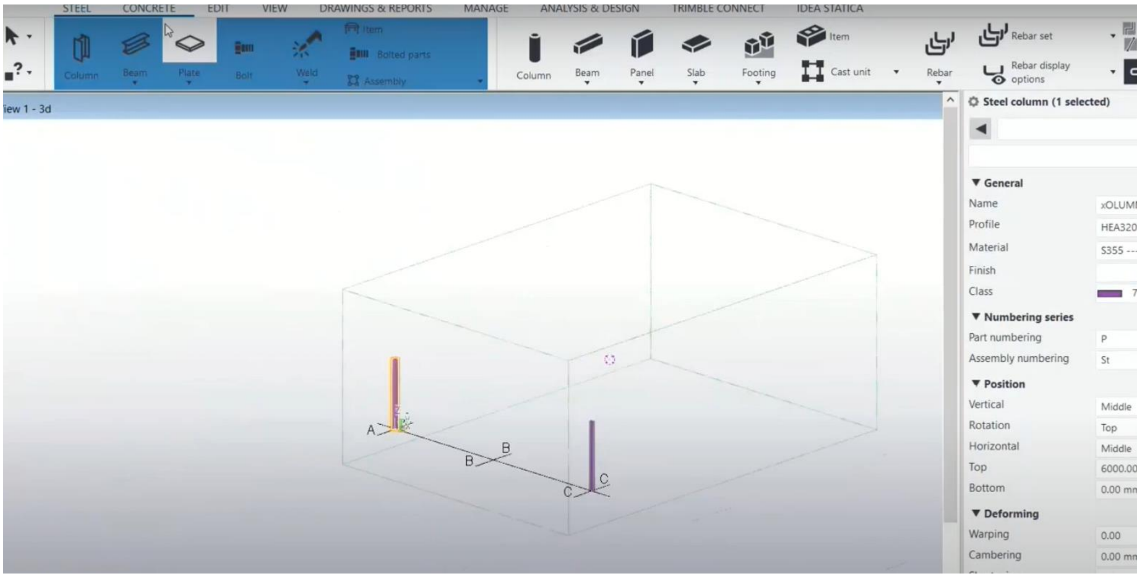
Designing the columns and beams

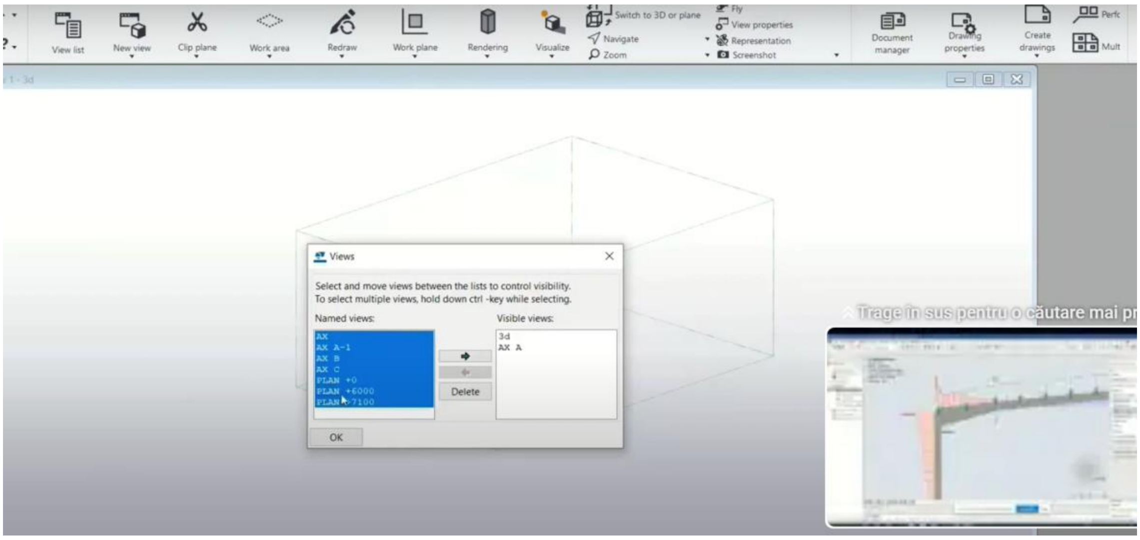
Designing the columns and beams involves a comprehensive process of structural analysis and engineering to ensure the elements can safely support the loads and forces imposed on them within a building or structure.

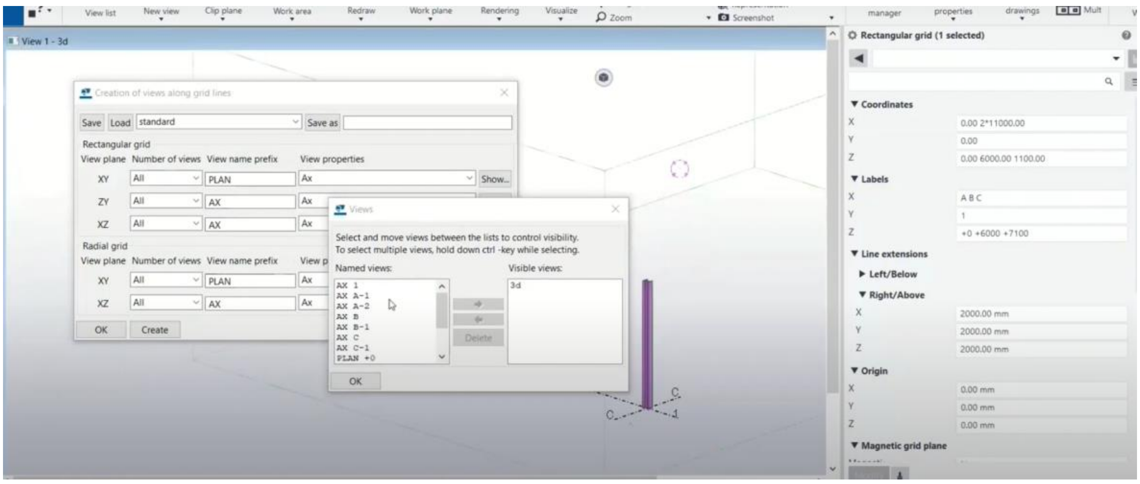


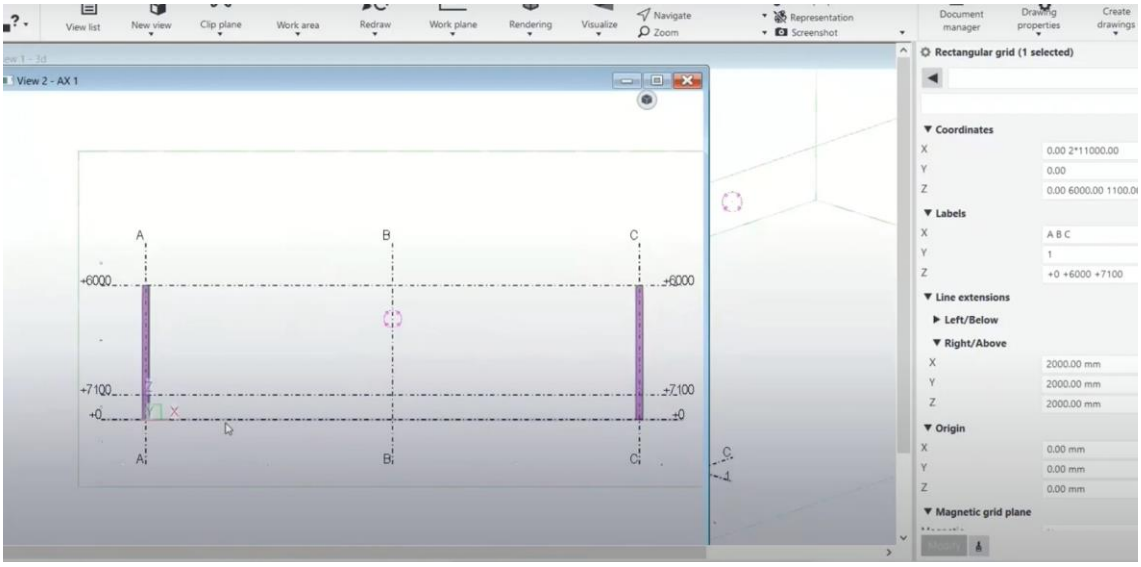


Tekla software tutorials. CS: Industrial Building.



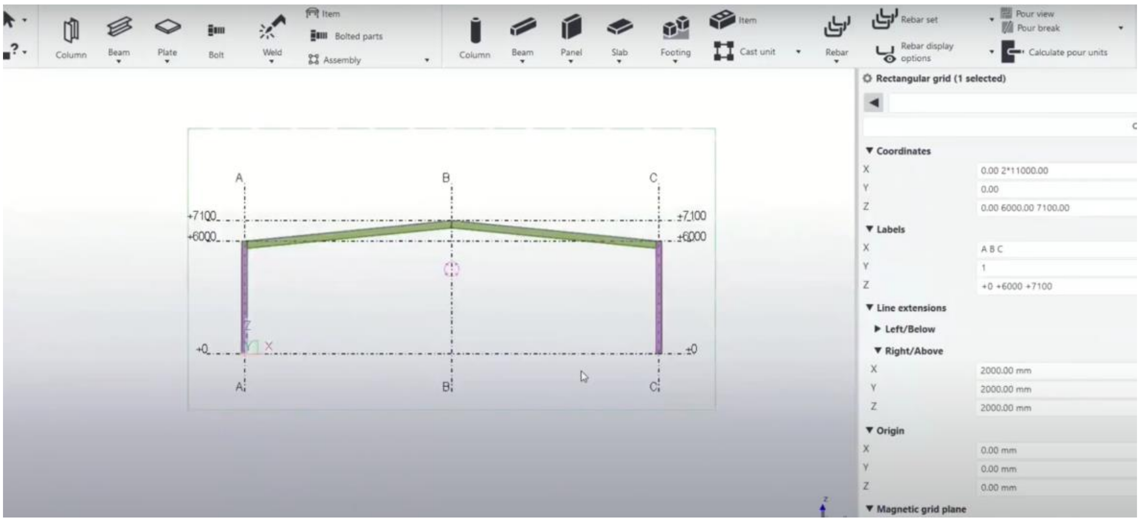


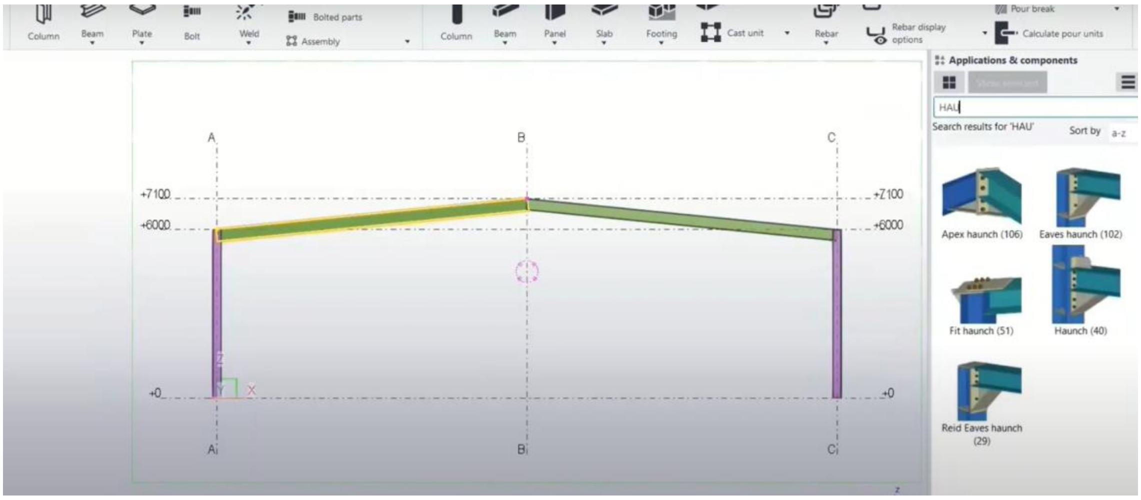


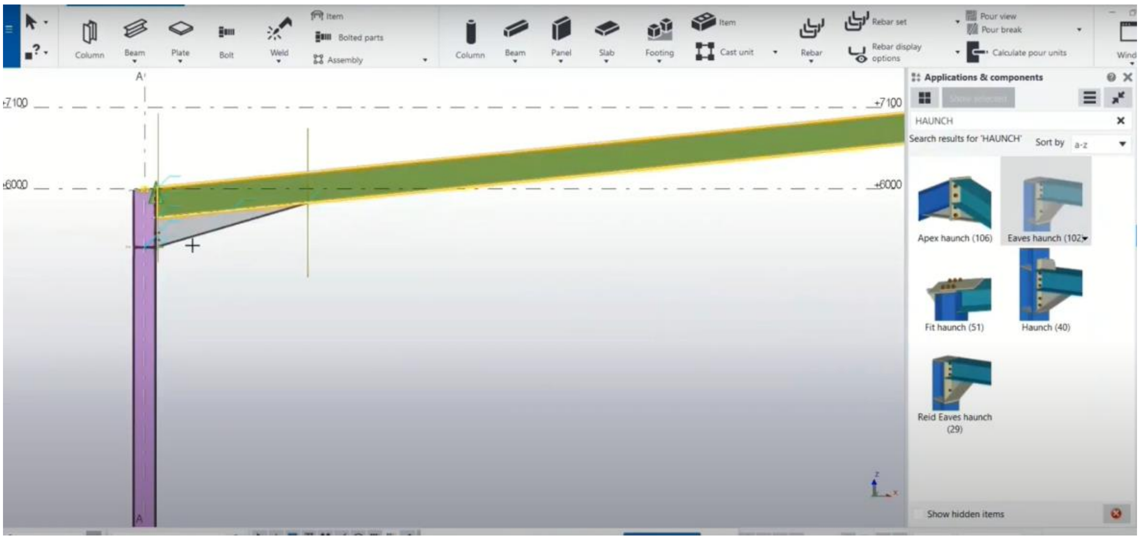


Assembly of the portal frame

This process begins with the preparation of the foundation and anchor bolts, followed by the positioning and alignment of the base plates to receive the column sections. Subsequently, columns are carefully lifted and secured into place, often using cranes or other lifting equipment, and connected to the base plates.

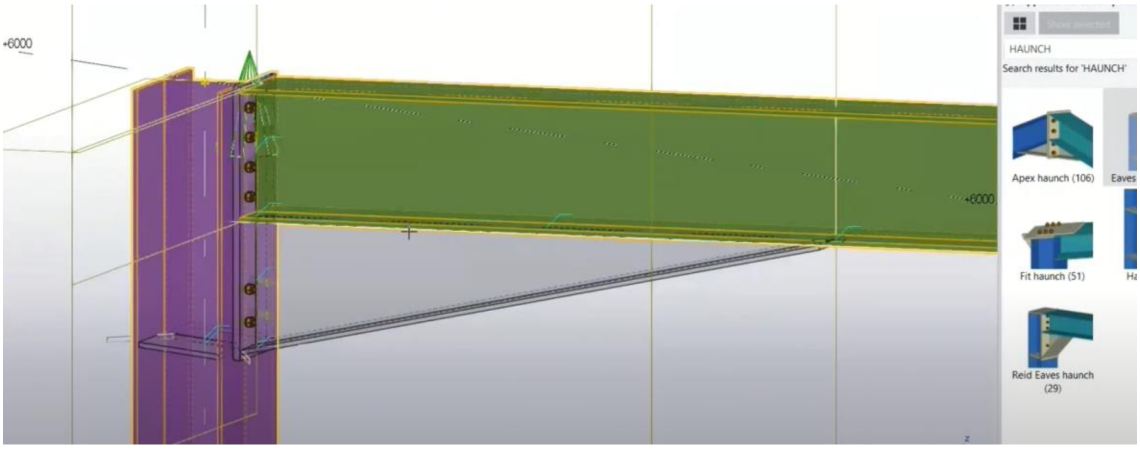


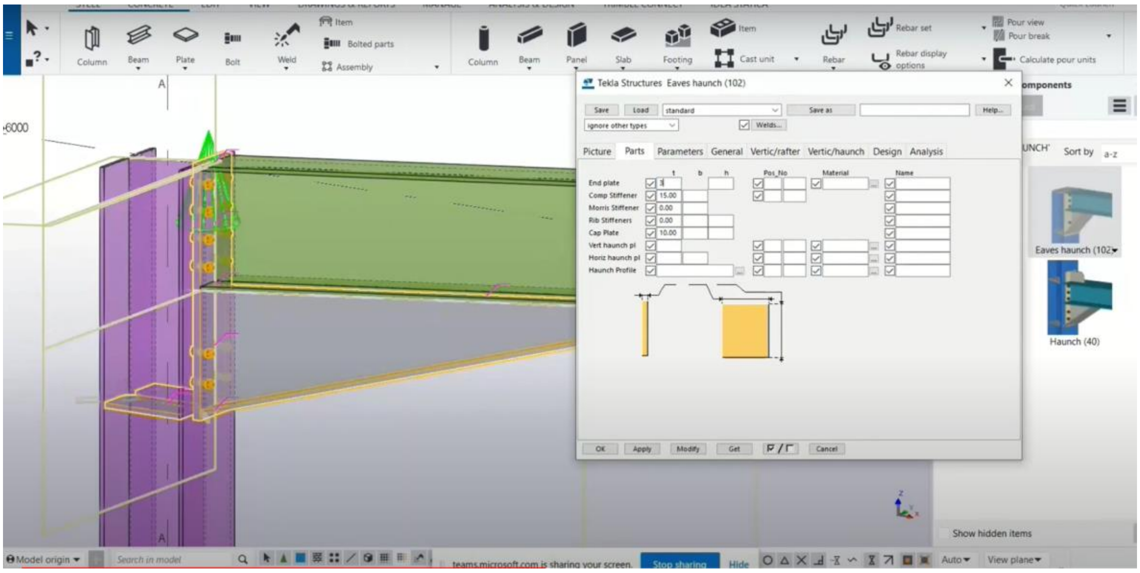


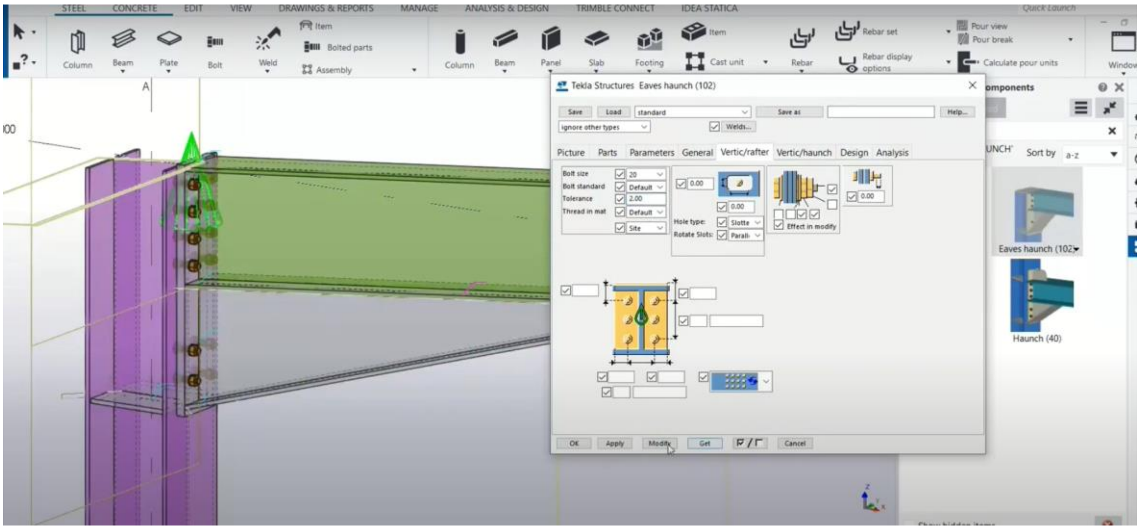


Connection design

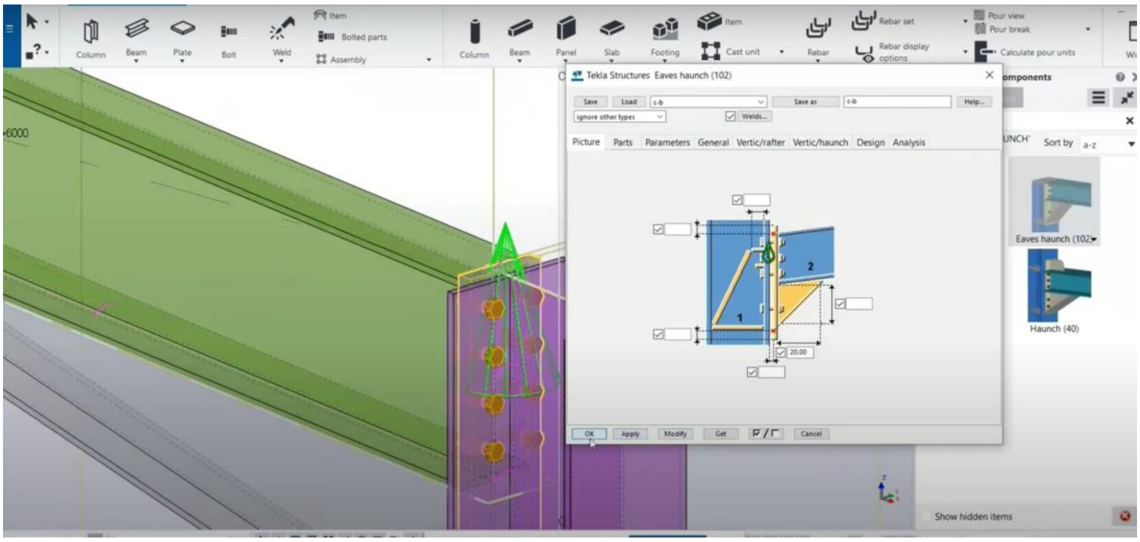
Connection design encompasses the critical process of determining the appropriate connections between structural elements in a building or infrastructure project. This involves analysing the loads and forces acting on the structure to ensure that connections are designed to safely transfer these loads while meeting structural integrity and performance requirements.

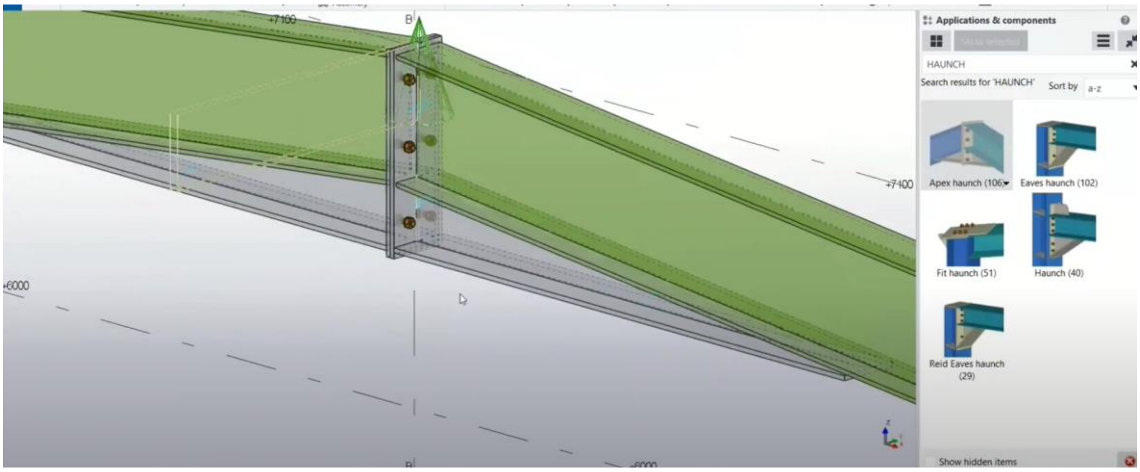

+6000
HAUNCH
Search results for 'HAUNCH'
Apex haunch (106)
Fit haunch (51)
Reid Eaves haunch (29)

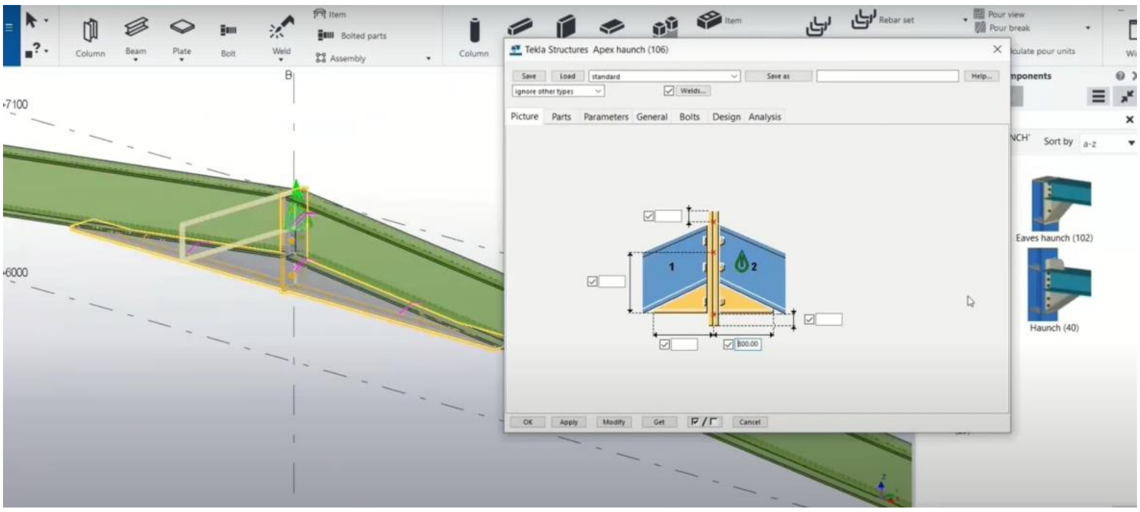

Column
Beam
Plate
Bolt
Weld
Bolted parts
Assembly
Panel
Slab
Footing
Cast unit
Rebar
Rebar set
Pour view
Pour break
Calculate pour units
Tekla Structures Eaves haunch (102)
Picture
Parts
Parameters
General
Vertic/rafter
Vertic/haunch
Design
Analysis
Eaves haunch (102)
Haunch (40)
Show hidden items

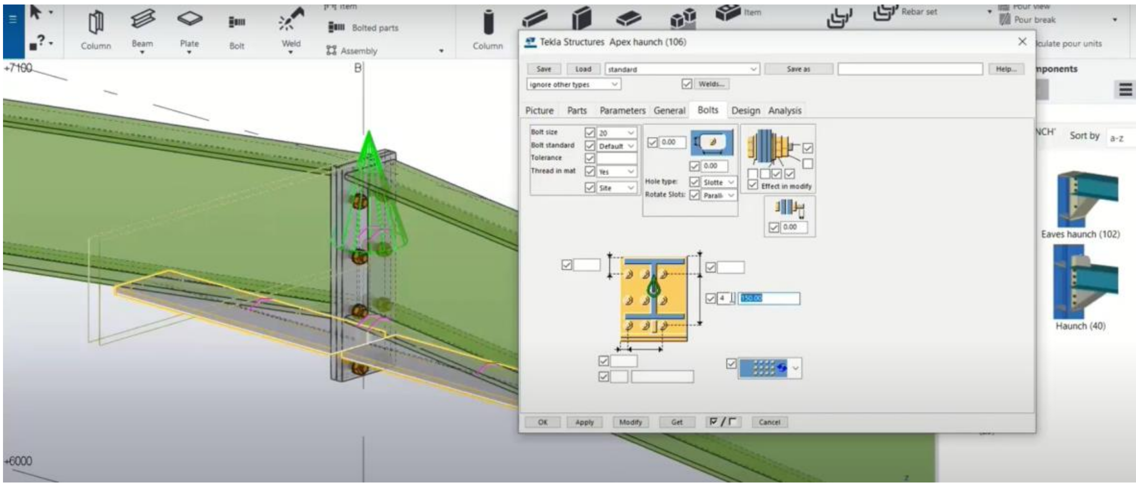

STEEL
CONCRETE
EDIT
VIEW
DRAWINGS & REPORTS
MANAGE
ANALYSIS & DESIGN
TRIMBLE CONNECT
IDEA STATICA
Tekla Structures Eaves haunch (102)
Picture
Parts
Parameters
General
Vertic/rafter
Vertic/haunch
Design
Analysis
Eaves haunch (102)
Haunch (40)

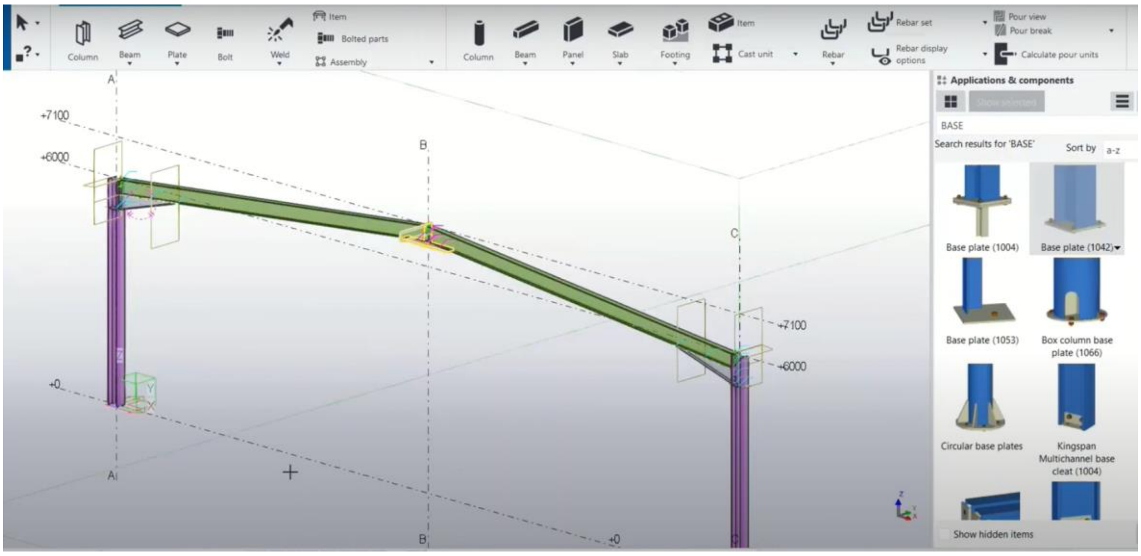
Tekla software tutorials. CS: Industrial Building.



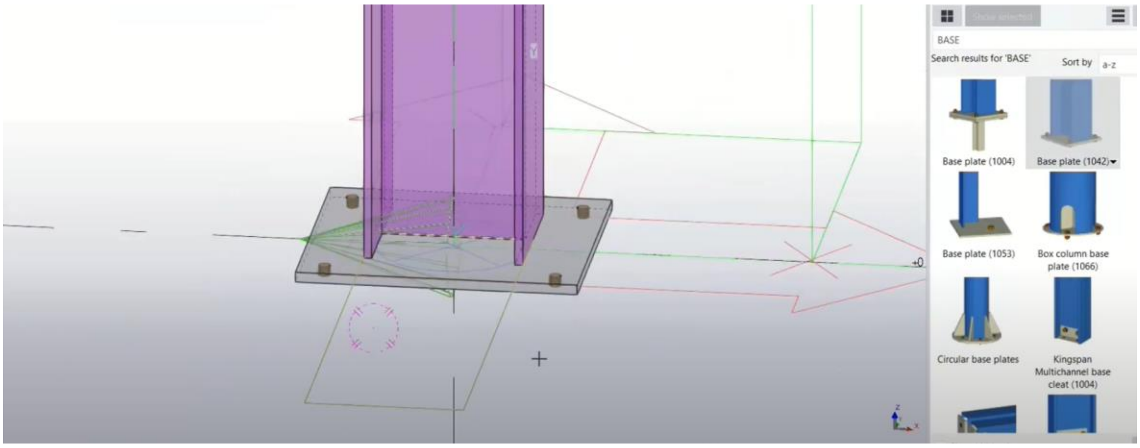


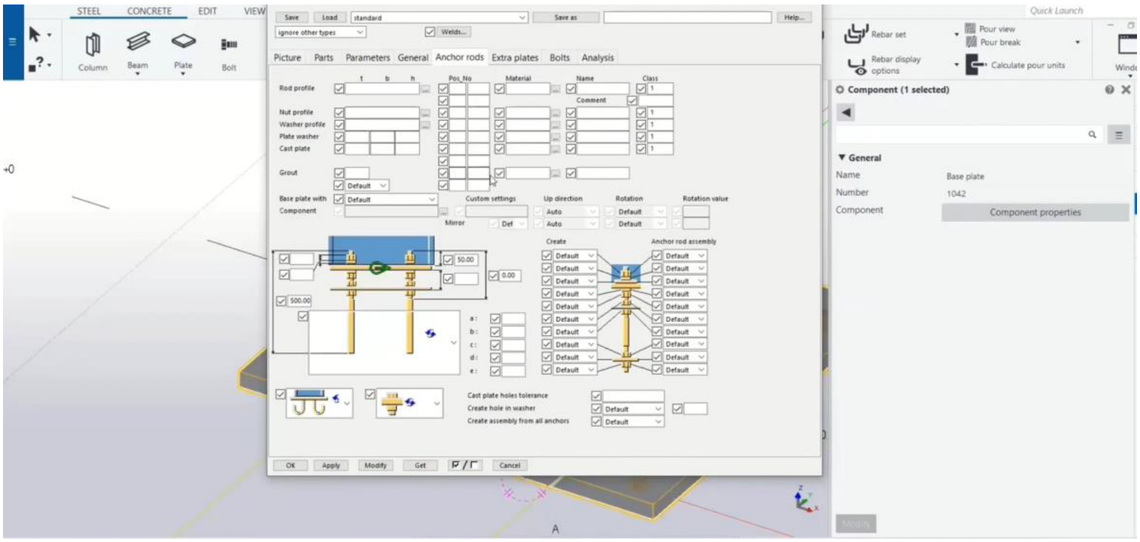


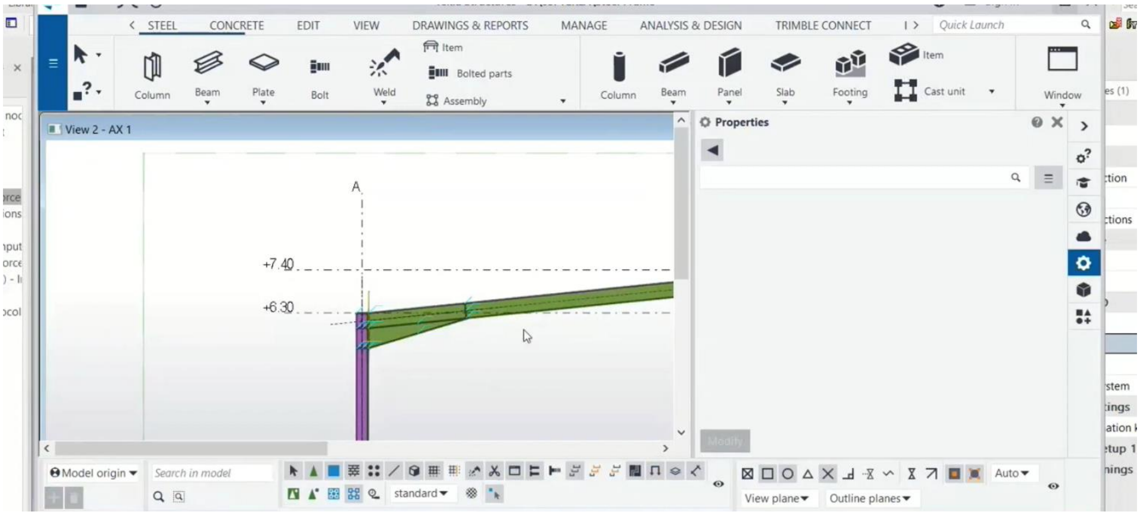


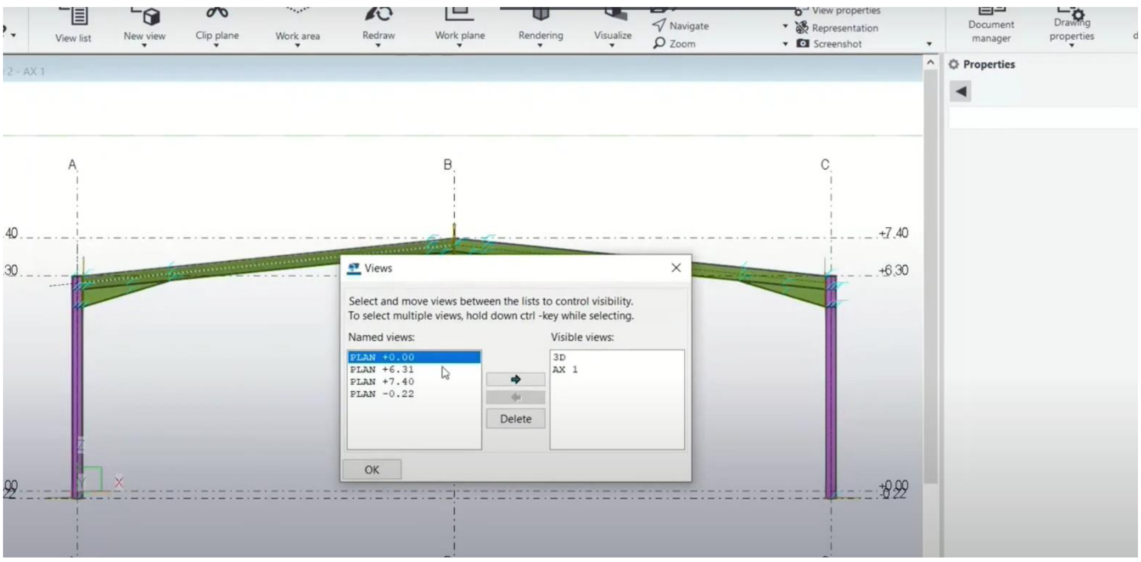


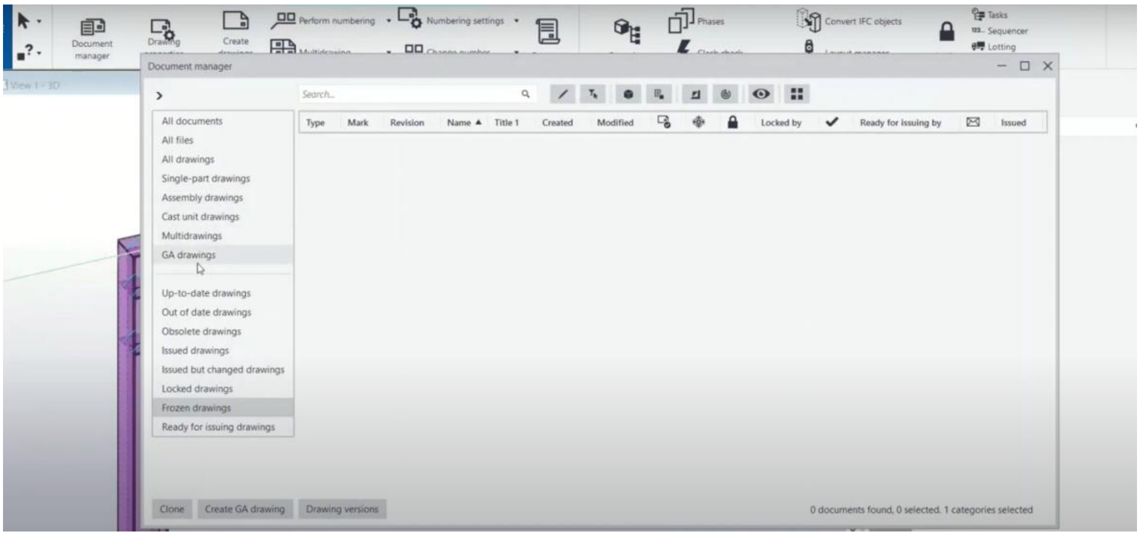
Base connection design



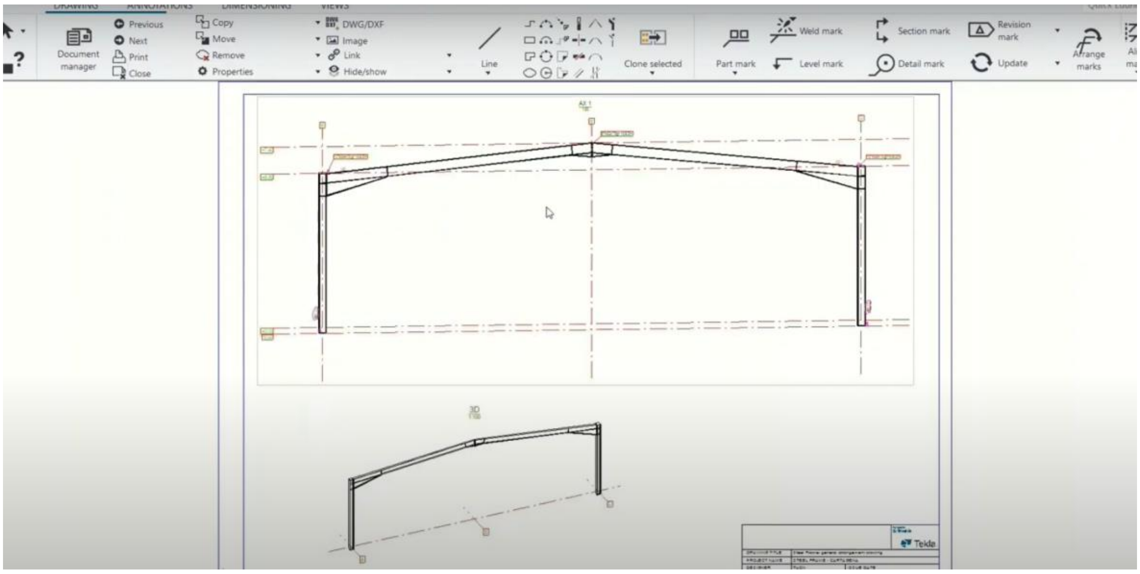

STEEL
CONCRETE
EDIT
Column
Beam
Plate
Bolt
Picture
Parts
Parameters
General
Anchor rods
Extra plates
Bolts
Analysis
Component (1 selected)
General
Name
Base plate
Number
1042
Component
Component properties
OK
Apply
Modify
Get
Cancel

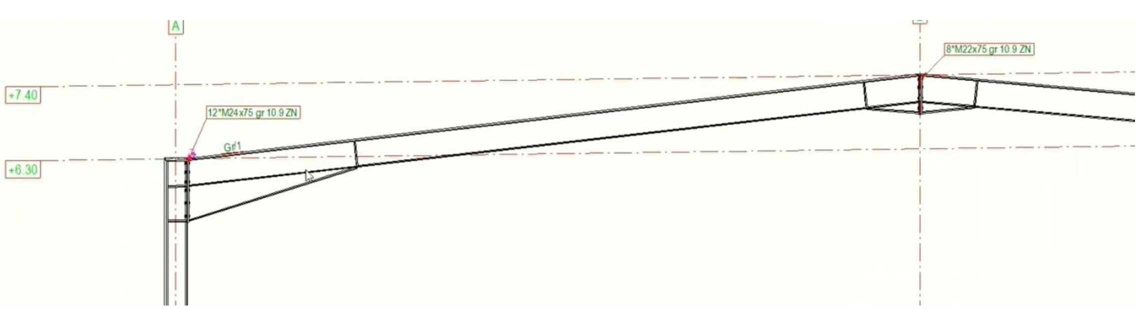

STEEL
CONCRETE
EDIT
VIEW
DRAWINGS & REPORTS
MANAGE
ANALYSIS & DESIGN
TRIMBLE CONNECT
Quick Launch
Column
Beam
Plate
Bolt
Weld
Item
Bolted parts
Assembly
Panel
Slab
Footing
Cast unit
Window
View 2 - AX 1
Properties
A
+7.40
+6.30
Model origin
Search in model
standard
View plane
Outline planes

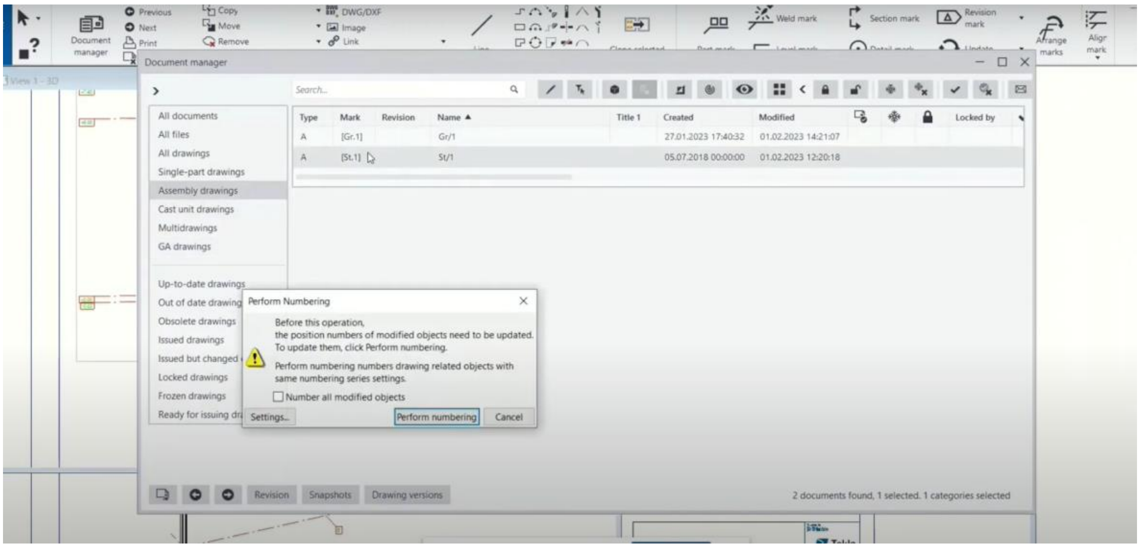

View list
New view
Clip plane
Work area
Redraw
Work plane
Rendering
Visualize
Navigate
Zoom
View properties
Representation
Screenshot
Document manager
Drawing properties
Properties
A
B
C
+7.40
+6.30
Views
Select and move views between the lists to control visibility.
To select multiple views, hold down ctrl -key while selecting.
Named views:
PLAN +0.00
PLAN +6.31
PLAN +7.40
PLAN -0.22
Visible views:
3D
AX 1
Delete
OK

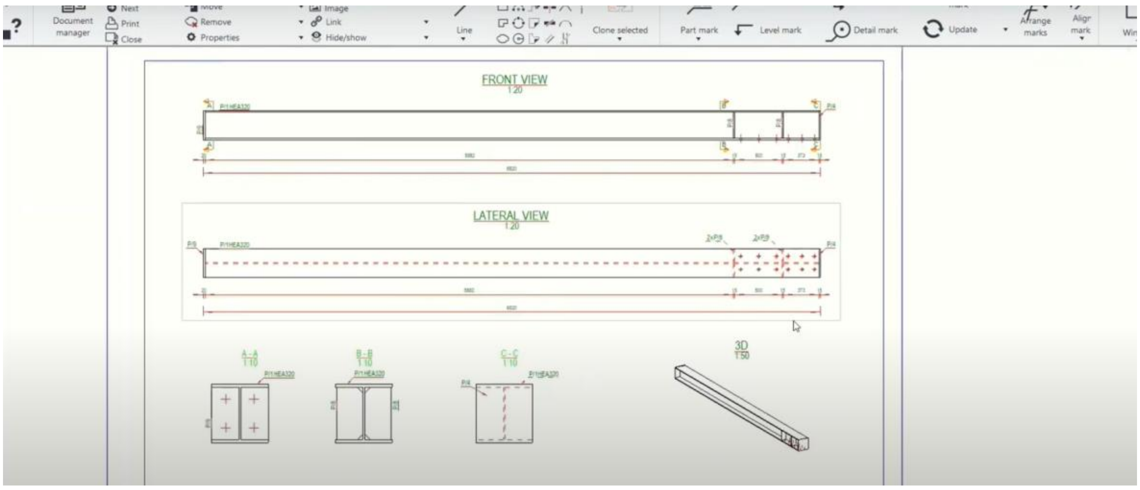


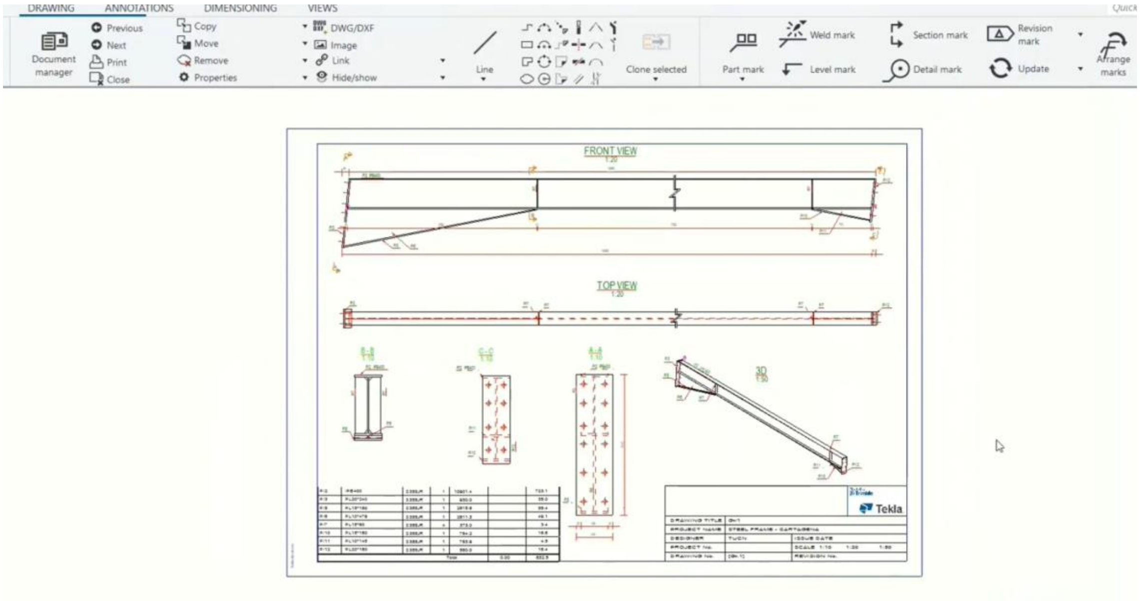
The plotting system for constructions enables precise visualization of geometric designs and structural layouts.

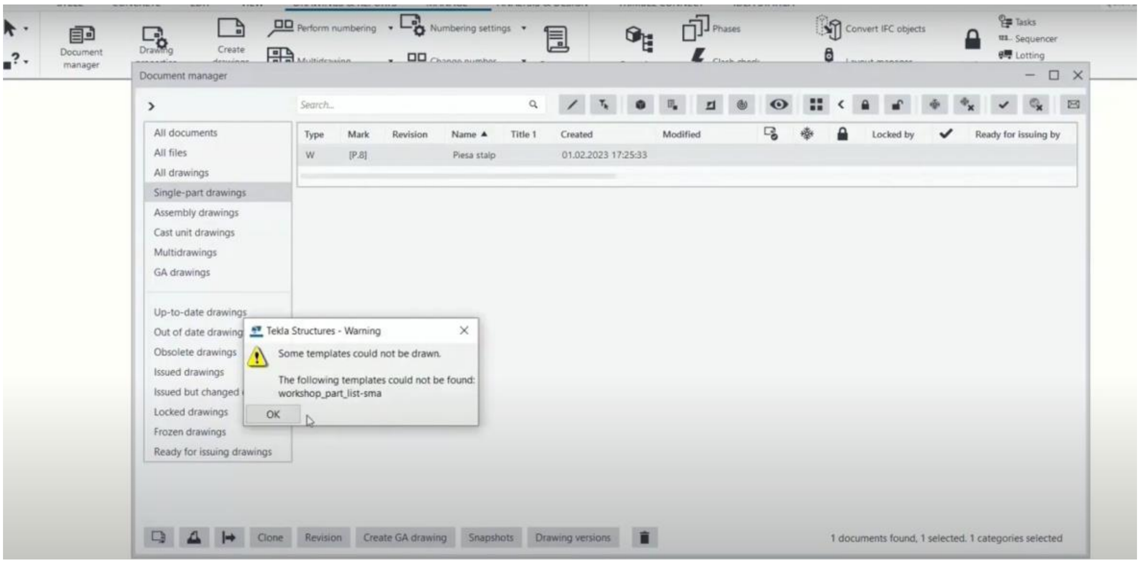




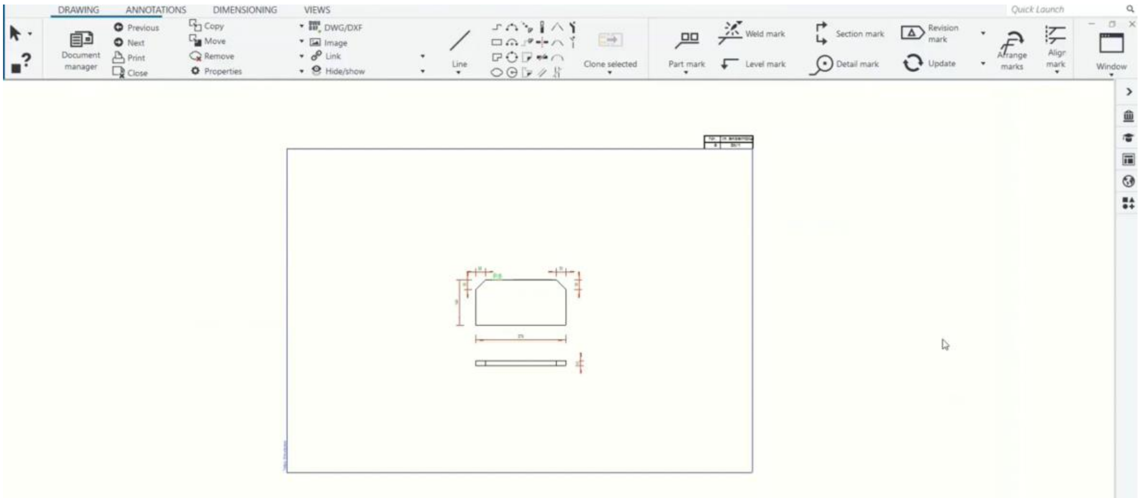

Document manager
All documents
All files
All drawings
Single-part drawings
Assembly drawings
Cast unit drawings
Multidrawings
GA drawings
Up-to-date drawings
Obsolete drawings
Issued drawings
Locked drawings
Frozen drawings
Type
Mark
Revision
Name
Title 1
Created
Modified
Locked by
[Gr.1]
Gr/1
27.01.2023 17:40:32
01.02.2023 14:21:07
[St.1]
St/1
05.07.2018 00:00:00
01.02.2023 12:20:18
Perform Numbering
Before this operation,
the position numbers of modified objects need to be updated.
To update them, click Perform numbering.
Perform numbering numbers drawing related objects with
same numbering series settings.
Number all modified objects
Settings...
Perform numbering
Cancel
Revision
Snapshots
Drawing versions
2 documents found, 1 selected, 1 categories selected

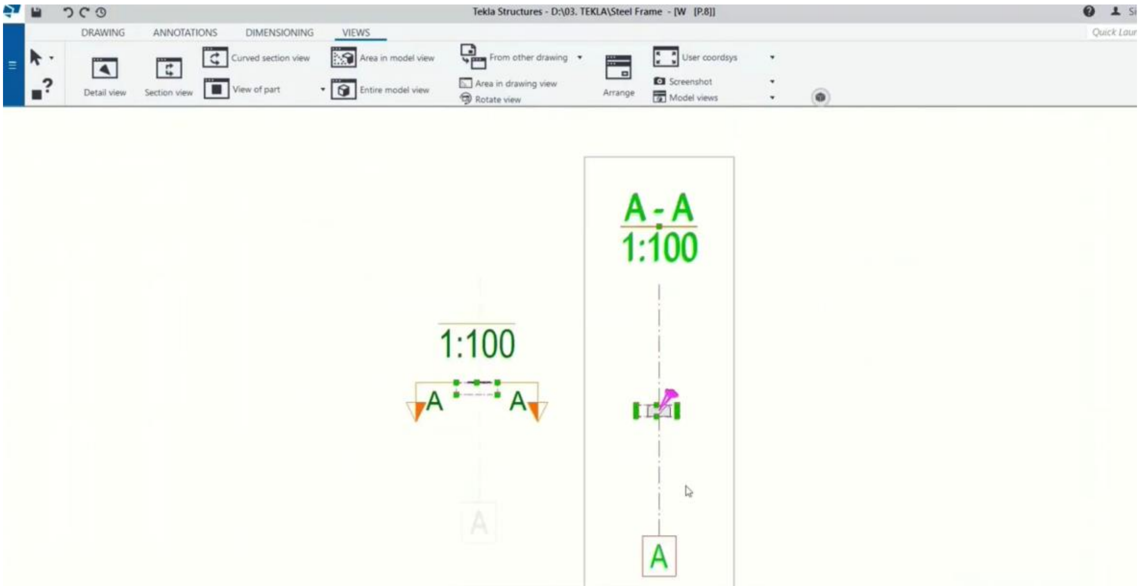

FRONT VIEW
LATERAL VIEW
3D

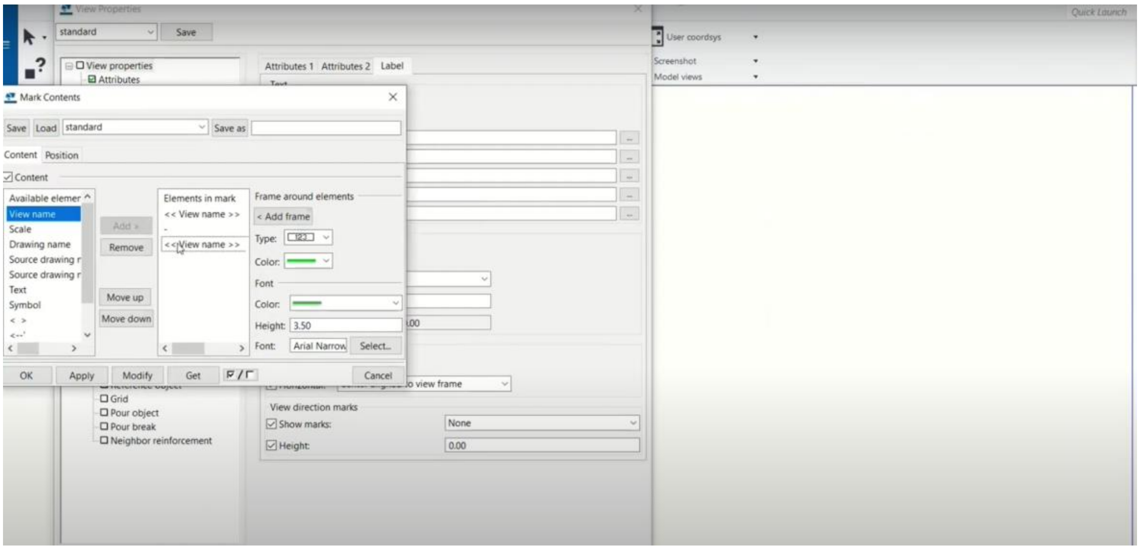

DRAWING
ANNOTATIONS
DIMENSIONING
VIEWS
Document manager
Previous
Next
Print
Close
Copy
Move
Remove
Properties
DWG/DXF
Image
Link
Hide/show
Line
Clone selected
Part mark
Weld mark
Level mark
Section mark
Detail mark
Revision mark
Update
Arrange marks
FRONT VIEW
TOP VIEW
Tekla

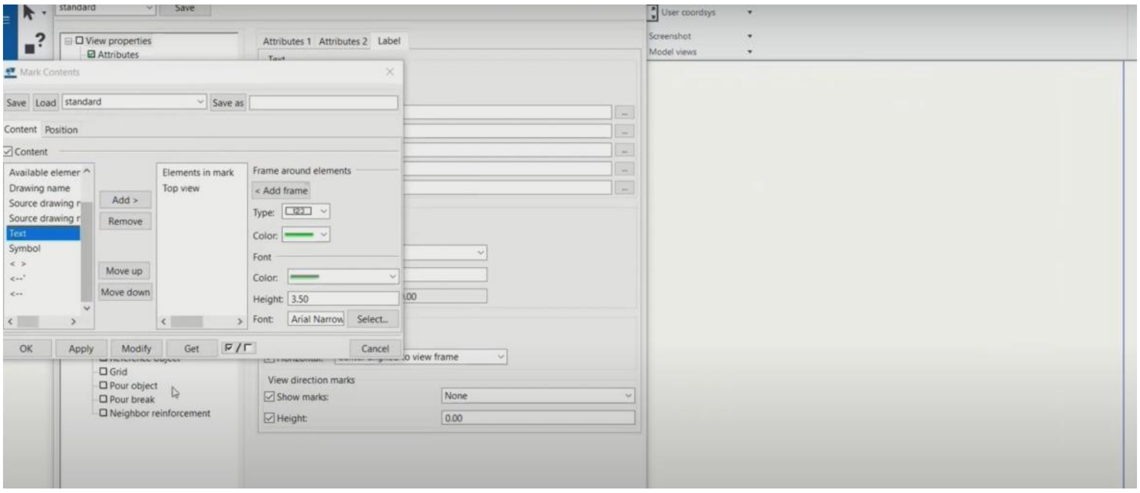

Document manager
Drawing
Create
Perform numbering
Numbering settings
Phases
Convert IFC objects
Tasks
Sequencer
Lotting
Search...
All documents
All files
All drawings
Single-part drawings
Assembly drawings
Cast unit drawings
Multidrawings
GA drawings
Up-to-date drawings
Obsolete drawings
Issued drawings
Locked drawings
Frozen drawings
Ready for issuing drawings
Type
Mark
Revision
Name
Title 1
Created
Modified
Locked by
Ready for issuing by
W
[P.8]
Piesa stalp
01.02.2023 17:25:33
Tekla Structures - Warning
Some templates could not be drawn.
The following templates could not be found:
workshop_part_list-sma
OK
Clone
Revision
Create GA drawing
Snapshots
Drawing versions
1 documents found, 1 selected. 1 categories selected

Tekla software tutorials. CS: Industrial Building.









The system is designed to ensure seamless compatibility and integration with IDEA Statica software, allowing for efficient data exchange and collaboration between platforms.

5.1.4 Conclusions

In conclusion, utilizing Tekla for the design and detailing of an industrial building offers significant advantages in terms of efficient modeling, precise connection design, accurate material quantification, and streamlined collaboration among project stakeholders. This comprehensive software solution enables engineers and designers to optimize the structural performance of the building, enhance constructability, and ensure compliance with industry standards, ultimately contributing to the successful execution of complex industrial projects.

References

[1] https://www.scia.net/en/innovations/integrated-design-solution

[2] https://www.tekla.com/resources

[3] https://www.ideastatica.com/support-center

[4] 1. Petran I., Senila M. – “DESIGN OF PITCHED ROOF STEEL PORTAL FRAME STRUCTURE”, Editura Mediamira, ISBN: 978-973-713-359-5, Cluj-Napoca, România, 2017

[5] EN 1991-1-1:2002. Eurocode 1: Actions on structures - Part 1-1: General actions - Densities, self-weight, imposed loads for buildings.

[6] EN 1991-1-3:2003. Eurocode 1: Actions on structures - Part 1-3: General actions - Snow loads and CR 1-1-3-2012: Design code. Assessing snow action on buildings.

[7] EN 1991-1-4:2005. Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions and CR 1-1-4-2012: Design code. Assessing wind action on buildings.

[8] P100-1/2013: Seismic Design Code. Part I: Design provisions for buildings.

[9] EN 1993-1-1:2005. Eurocode 3: Design of steel structures - Part 1-1: General rules

and rules for buildings

6 - Deliverables

To evaluate the success of the application, students will have to prepare a report on the steps taken in practice, difficulties faced and decisions taken.

7- What we have learned

To create a model of building using Tekla Structure.