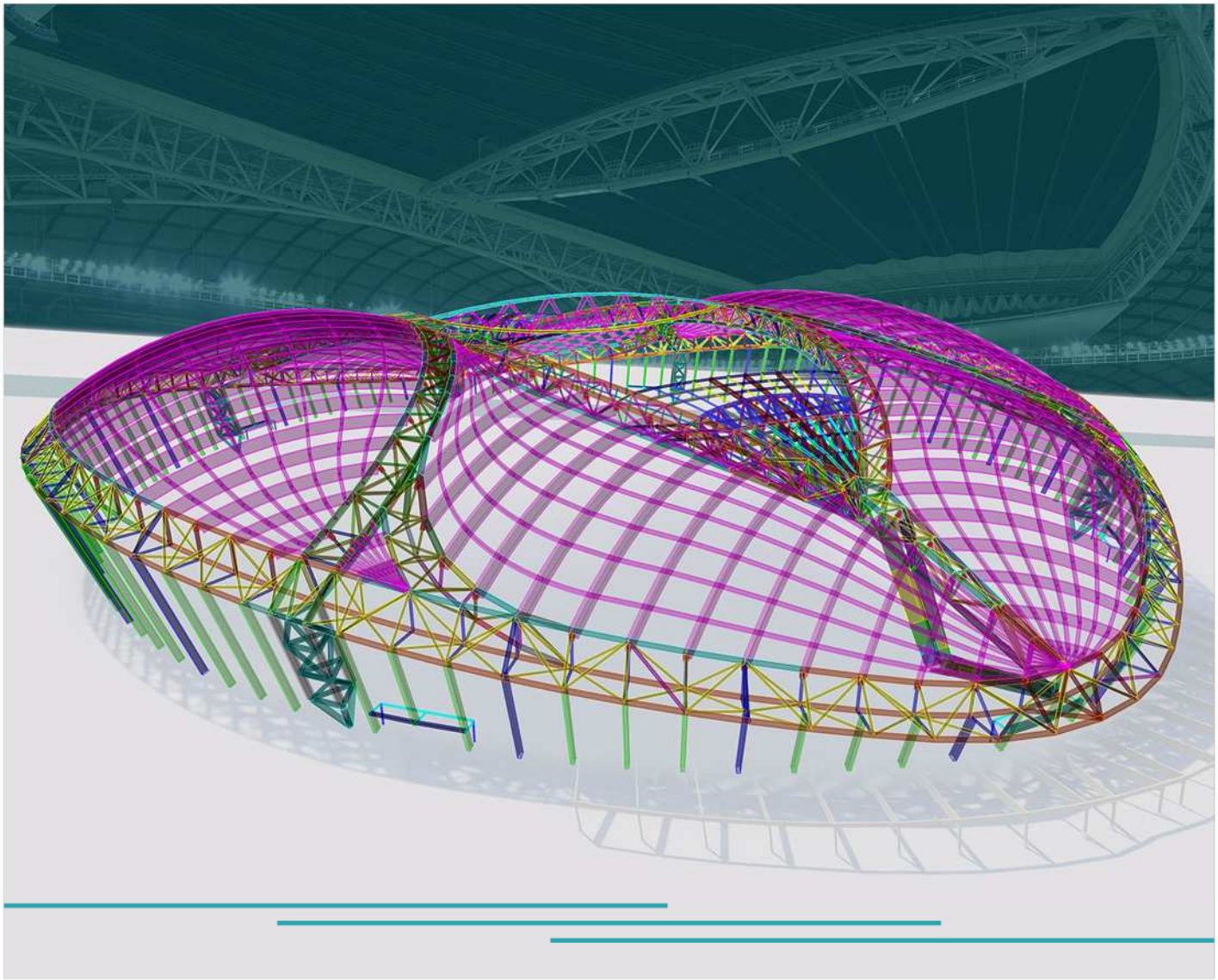
SCIA ENGINEER



TUTORIAL

CALCULATION OF TIMBER TRUSS JOINTS WITH SCIA & FRILO

Table of Contents

Introduction

Since FRILO release R2023-2 and SCIA Engineer version 22.1 it is now possible to transfer Timber Truss Joints from a 3D model of SCIA Engineer to the FRILO program “Timber Truss Joint - HO13+”.

Application options of FRILO HO13+

HO13+ is suitable for the calculation of typical truss nodes in timber construction. In such a joint, up to five outgoing bars are connected in a centre point. It is important that the bars lie in one plane.

The currently available fasteners are dowel pins/fit bolts/bolts as well as nails. For timber-to-timber connections and steel-to-timber connections with surface-mounted plates, special dowels are additionally available. In timber-to-timber connections, combined arrangements of dowel pins and fit bolts are definable.

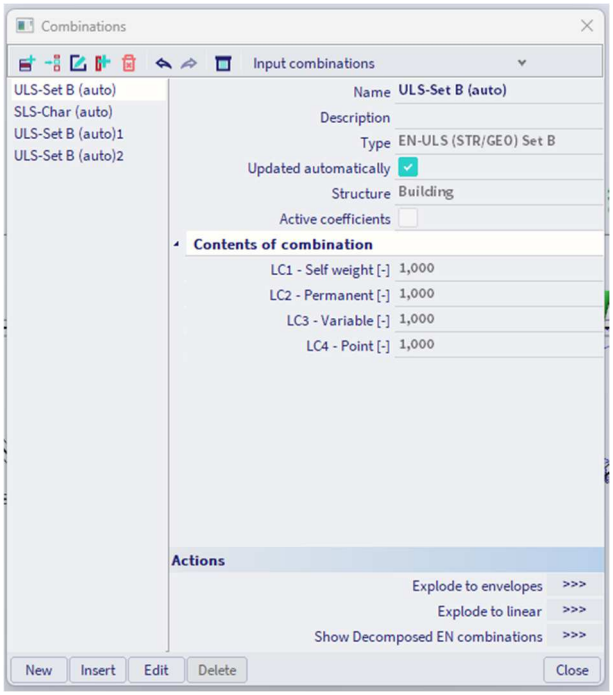
The connection strength verifications of the fasteners are performed in accordance with Johansen's theory (verification method in accordance with Annex G of DIN 1052: 2004/2008). Suspension effects can be taken into account, if applicable. On the basis of the connection strength verifications, the software calculates the required number of fasteners, checks the minimum spacing to be complied with and performs the necessary verifications on the connectedcomponents in the area of the joint.

You can find further information to HO13+ via the following link:
https://www.frilo.eu/wp-content/uploads/EN/Manuals/fl_ho13_ho3_ho14_eng.pdf

Phase 1: Model preparation In SCIA Engineer

Step 1: Preparation of the model in SCIA Engineer

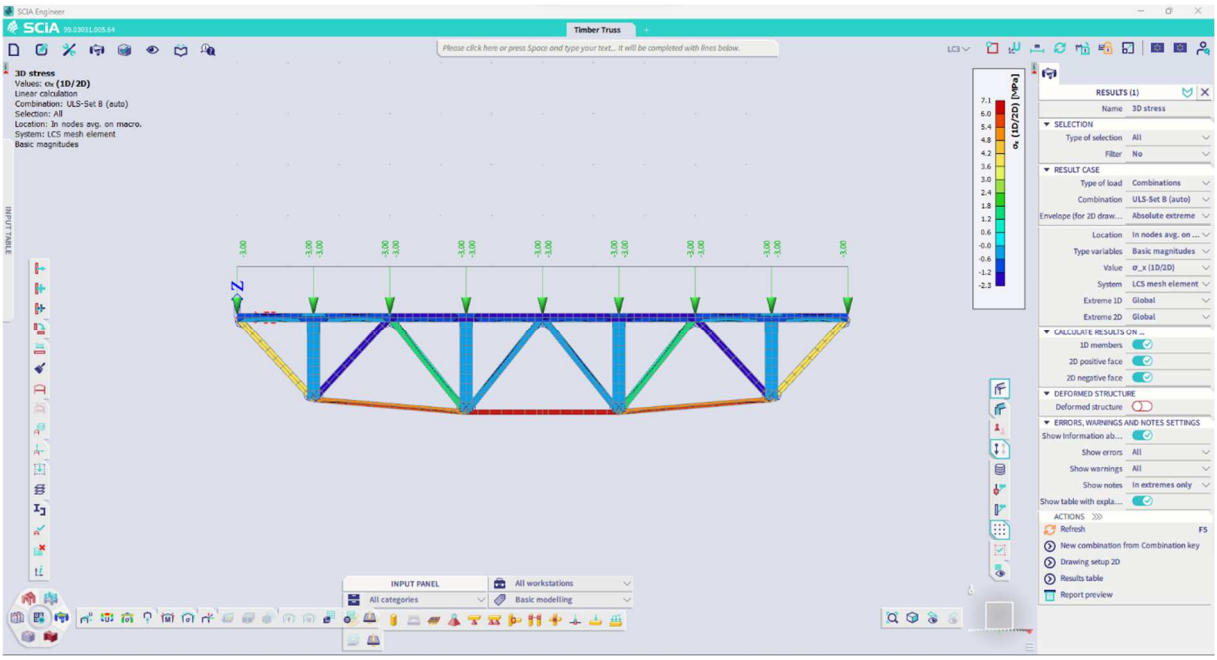
Since design internal forces are transferred between SCIA Engineer and HO13+, it is important that a load case combination is created in SCIA Engineer. This should be exploded to linear before the calculation.



Furthermore, it is important that the members are connected hinged in the generated truss, the chord may be continuous. The "truss bar" property does not currently affect the transfer.

Step 2: Localize design-relevant node

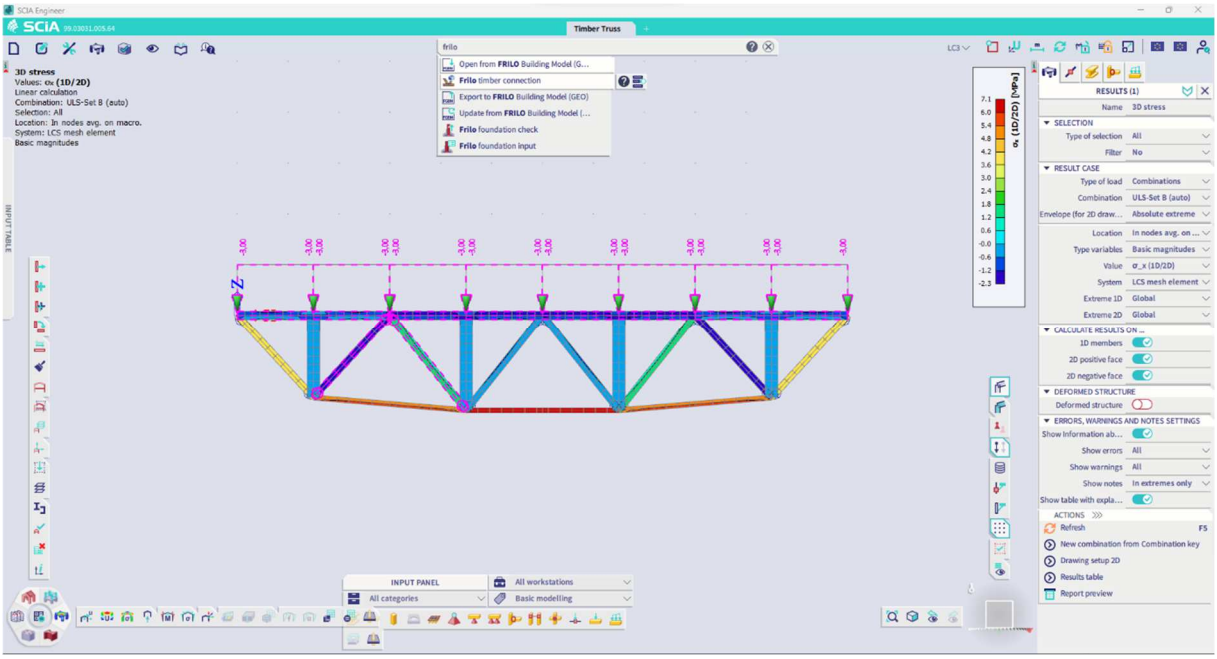
Once the model has been calculated, the most highly stressed node can be located by displaying the 1D-internal forces and the stresses. The node to be verified can then be identified and followed up.



More information on interpreting the results can be found here:
https://help.scia.net/webhelplatest/en/#results/commands/1d_internal_forces.htm

Step 3: Call HO13+ interface

Before calling the interface, you can either select an entire 3D node, or you can already select only the members that lie in a plane. In HO13+, the necessary equilibrium forces are added and generated in this case. After that, the interface "FRILO Timer Connection " can be searched for via SCIA Spotlight in the upper edge of the screen. After calling the command, the node is transferred.

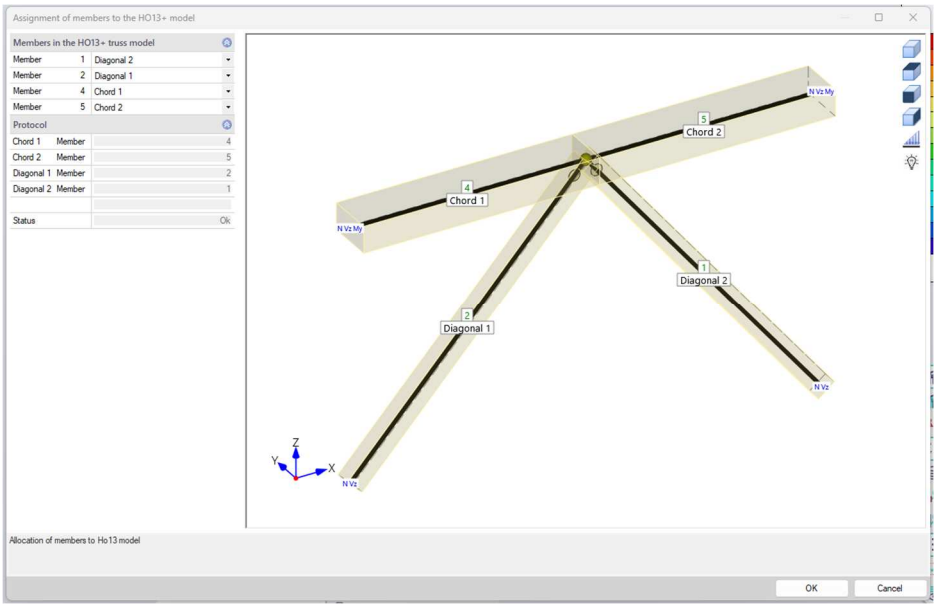


Phase 2: Design in FRILO's HO13+

Step 4: Intermediate dialog

Before the node is transferred to the HO13+ program, an intermediate dialog appears in which the members to be designed can be selected. In HO13+ it is not possible to design a 3D node. For this reason, a 3D node must be reduced to a 2D node in this intermediate dialog, in which the members lie in the same plane. The connection should then always be dimensioned separately for each plane.

Before the transfer, it is checked which type of member it is - diagonal, chord and post. If necessary, the preselection can be adjusted here again. Furthermore, it is possible to deselect any existing torsional moments or shear forces here. These cannot be taken into account in the HO13+ calculation.



Step 5: Dimensioning in HO13+

Afterwards, the node is automatically transferred to the FRILO program HO13+. In addition to the design standard, the material quality and the cross-section dimensions, the corresponding load combinations with the associated design internal forces are also transferred.

As a rule, the program already makes a proposal for the fastener after the transfer. However, this automatic arrangement cannot be guaranteed for every geometry. If this is the case, manual reworking is necessary.

The calculation can be started by clicking the "Calculation" button in the toolbar at the top of the screen.