

SESAM TUTORIAL

GeniE

Modelling an Arch

Valid from program version 8.2





Sesam Tutorial

GeniE – Modelling an Arch

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved

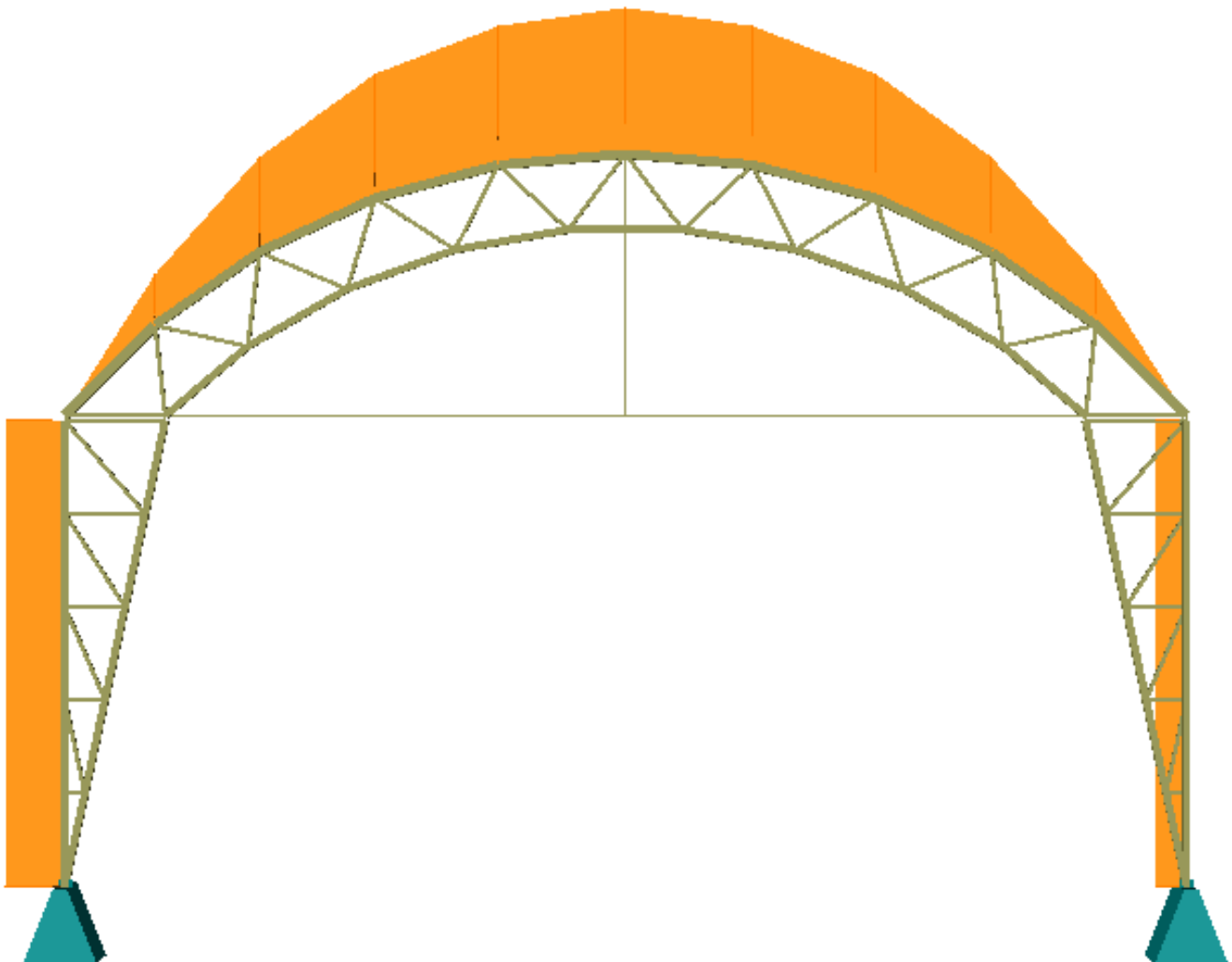
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. Open Workspace	Page 5
3. Material and Beam Cross Sections	Page 6
4. Guiding Geometry	Page 8
5. Upper and Lower Chord of the Arch	Page 10
6. Column Frame	Page 13
7. Complete Model	Page 16
8. Loads	Page 17
9. Structural Analysis	Page 19
10. View Results	Page 20
11. Redesign and Reanalysis	Page 21

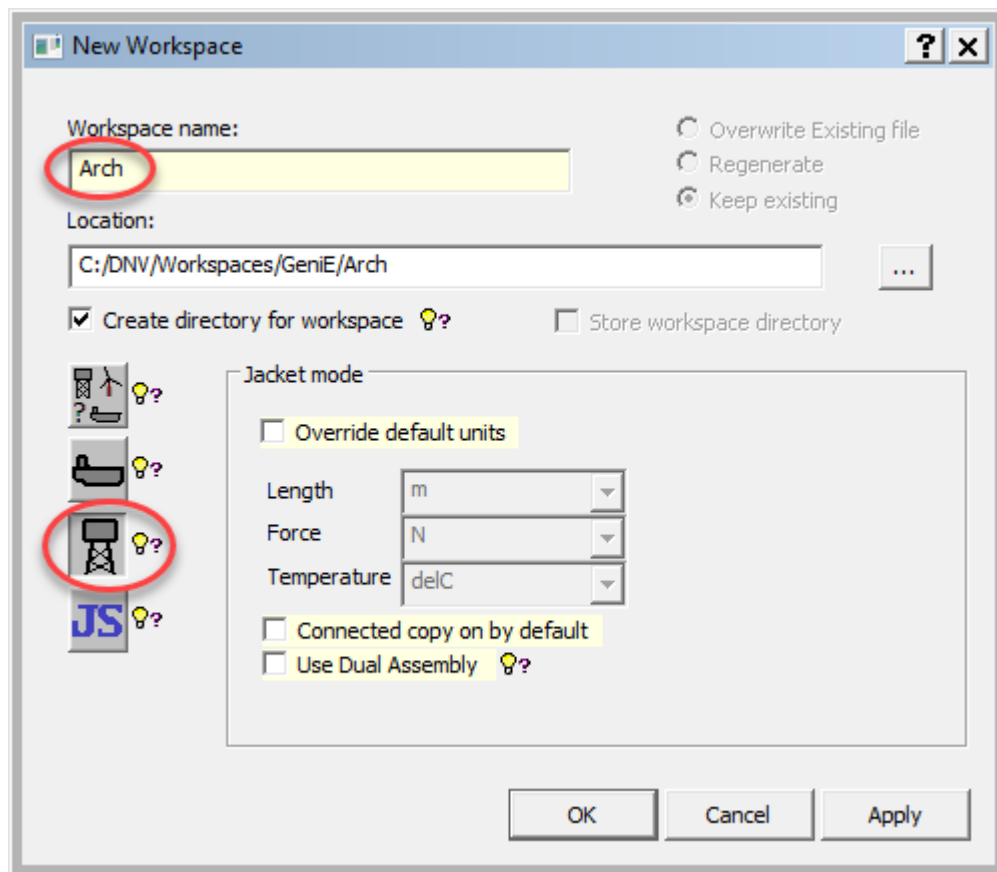
1 INTRODUCTION

- In this tutorial an arch is modelled and analysed by the following steps:
 - Define materials and beam cross sections
 - Define guiding geometry to facilitate the modelling
 - Model the arch
 - Specify boundary conditions
 - Add loading conditions
 - Perform linear elastic analysis
 - Evaluate results, redesign and reanalyse
- A GeniE input file for creating the complete model is provided.
- The model with snow and wind loads is displayed below.



2 OPEN WORKSPACE

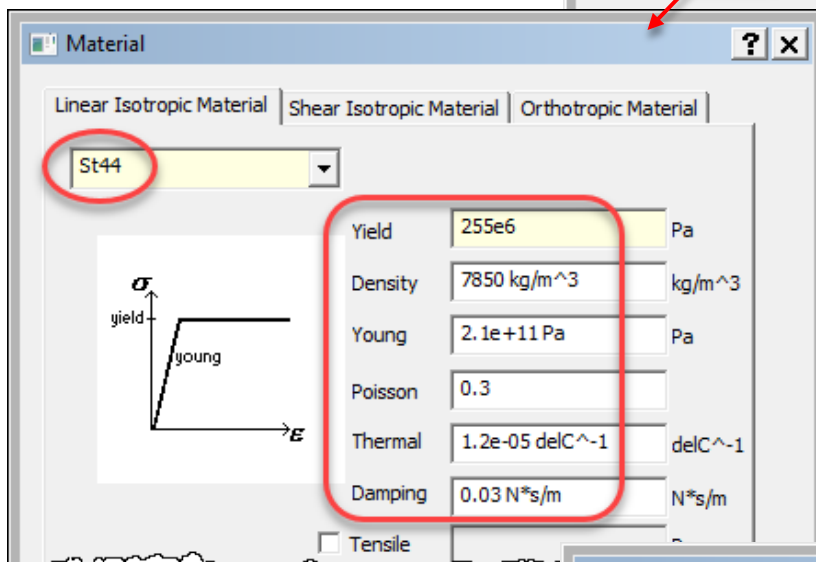
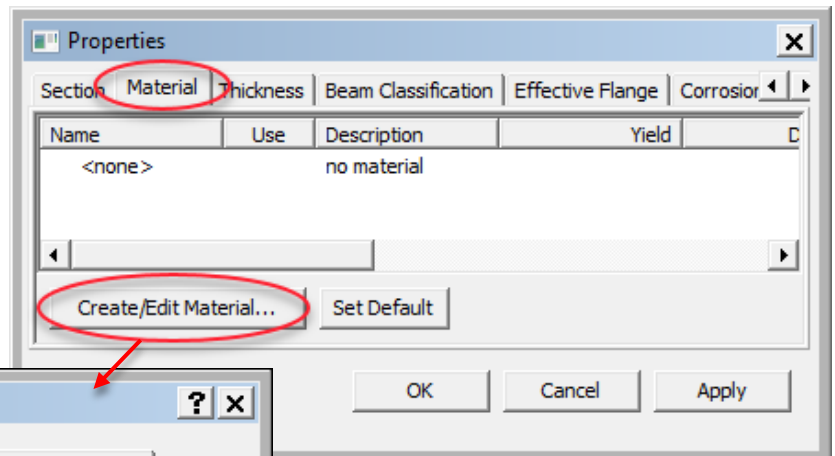
- Start GeniE and open a new workspace.
 - Give a *Workspace name*.
 - Click the *Jacket mode* button to customise for jacket (spaceframe) modelling, i.e. limit menus and buttons to those relevant for frame modelling.
 - Accept output units m and N and click *OK*.



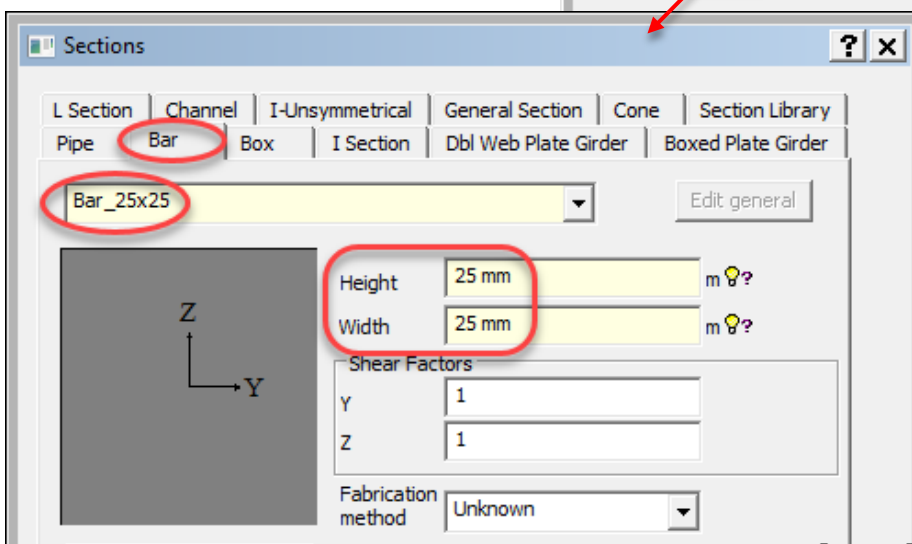
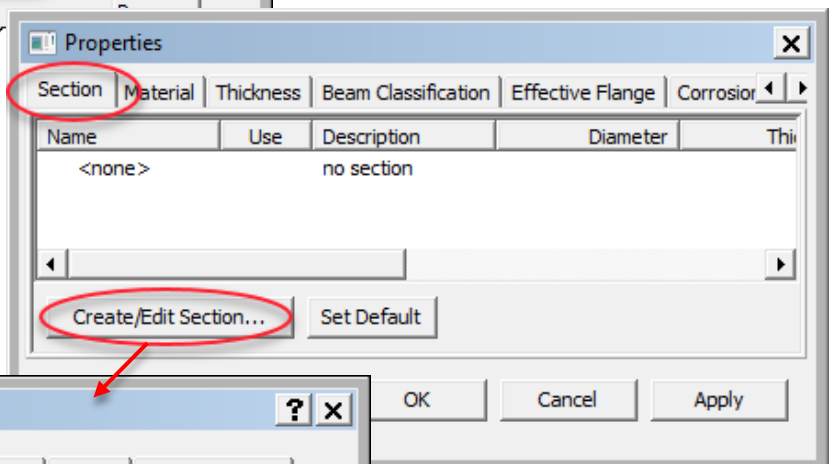
- If the workspace name exists select *Overwrite Existing file* or give another name.

3 MATERIAL AND BEAM CROSS SECTIONS

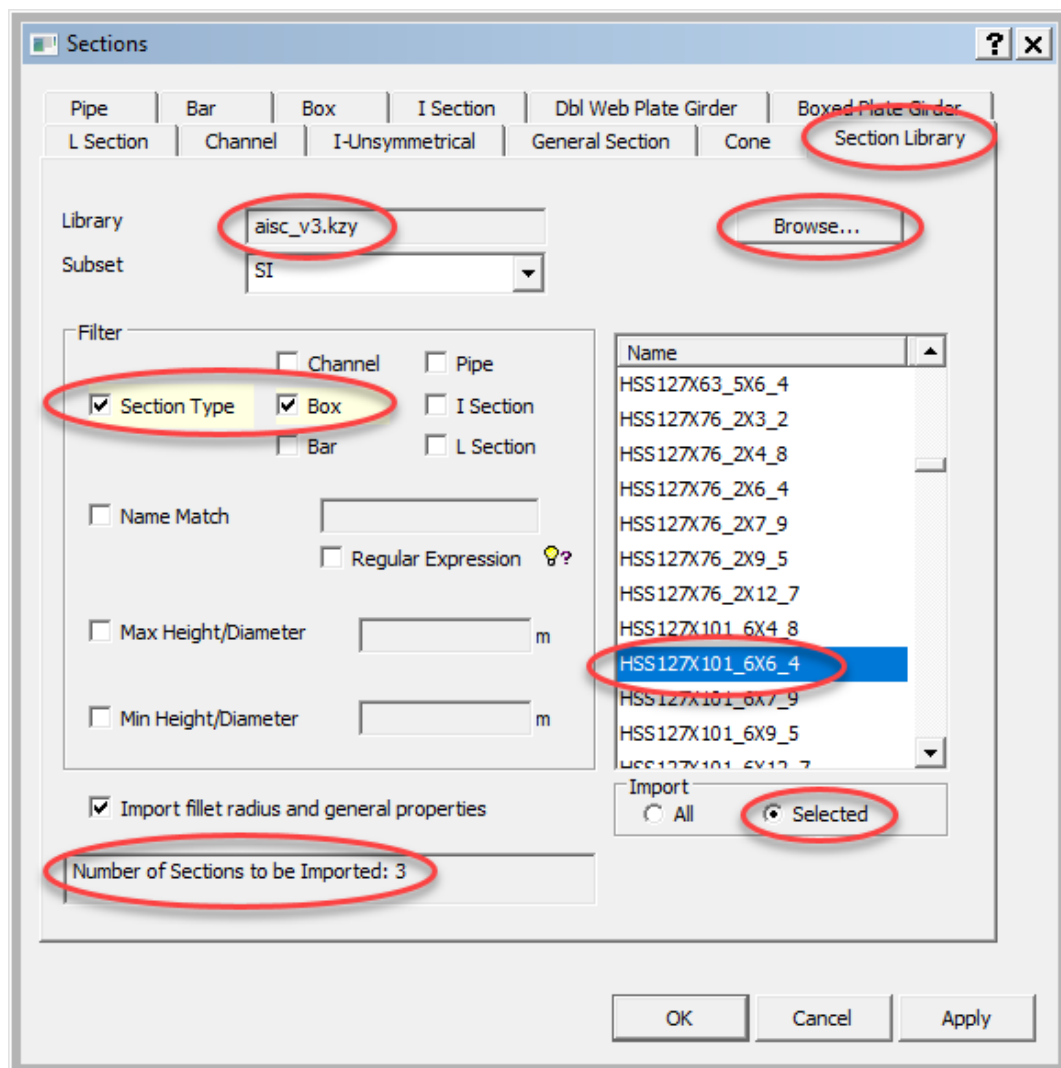
- Use *Edit | Properties* to open the *Properties* dialog. Go to the *Material* tab, click *Create/Edit Material* and give data as shown below.



- Go to the *Section* tab of the *Properties* dialog and click *Create/Edit Section* to create a *Bar* type beam cross section. Let the name be *Bar_25x25* and height and width as shown.

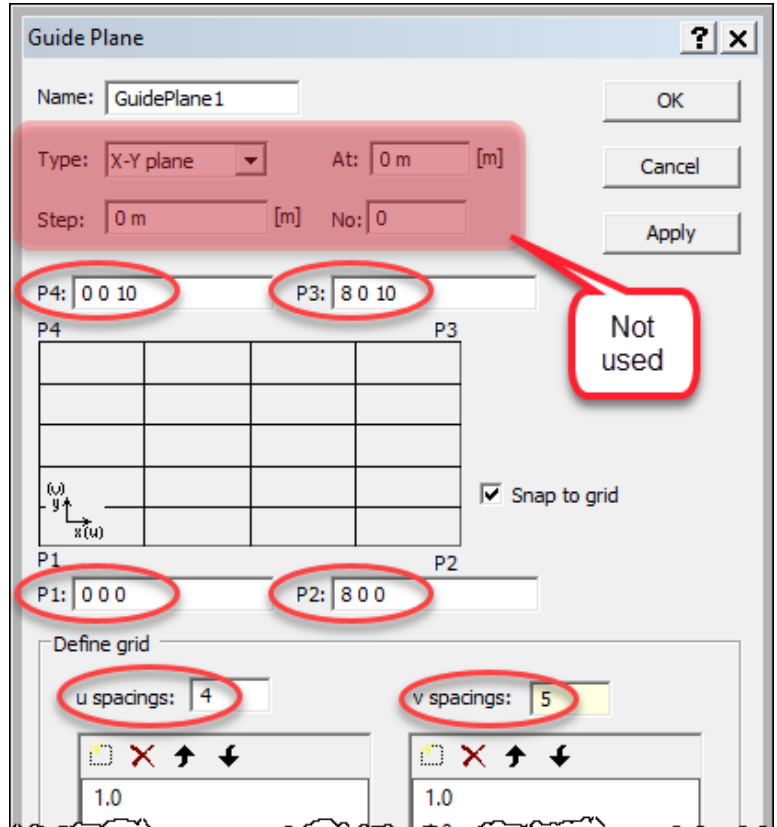


- Go to the *Section Library* tab of the *Sections* dialog to import beam cross sections.
 - Click *Browse* to find the library named aisc_v3.kzy.
 - Check *Section Type* and *Box* to filter the list of sections.
 - Select the section names (Click and Ctrl+Click):
 - HSS57_2X57_2X6_4
 - HSS101_6X101_6X6_4
 - HSS127X101_6X6_4
 - Make sure number of sections to be imported is 3 and click *OK*.

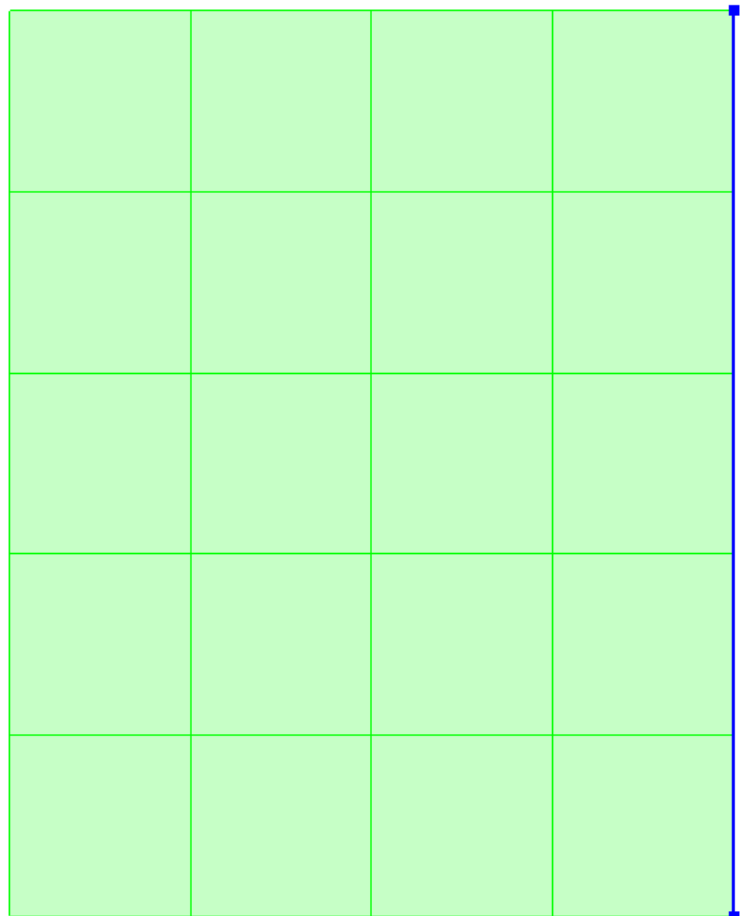


4 GUIDING GEOMETRY

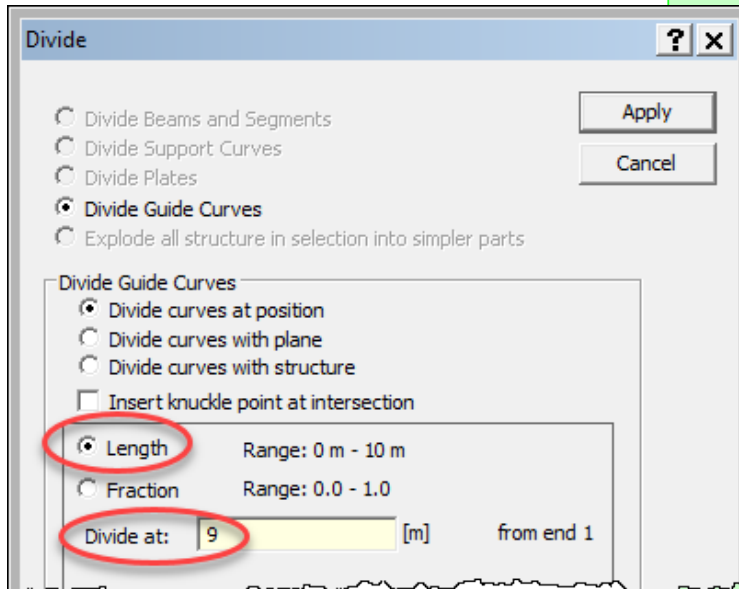
- Use *Guiding Geometry | Planes | Guide Plane Dialog* to open the dialog to the right. Enter data as shown to create a 8 by 10 m grid with 4 horizontal and 5 vertical even spacings.
 - The shaded area is for creating multiple guide planes and is not used in this case.



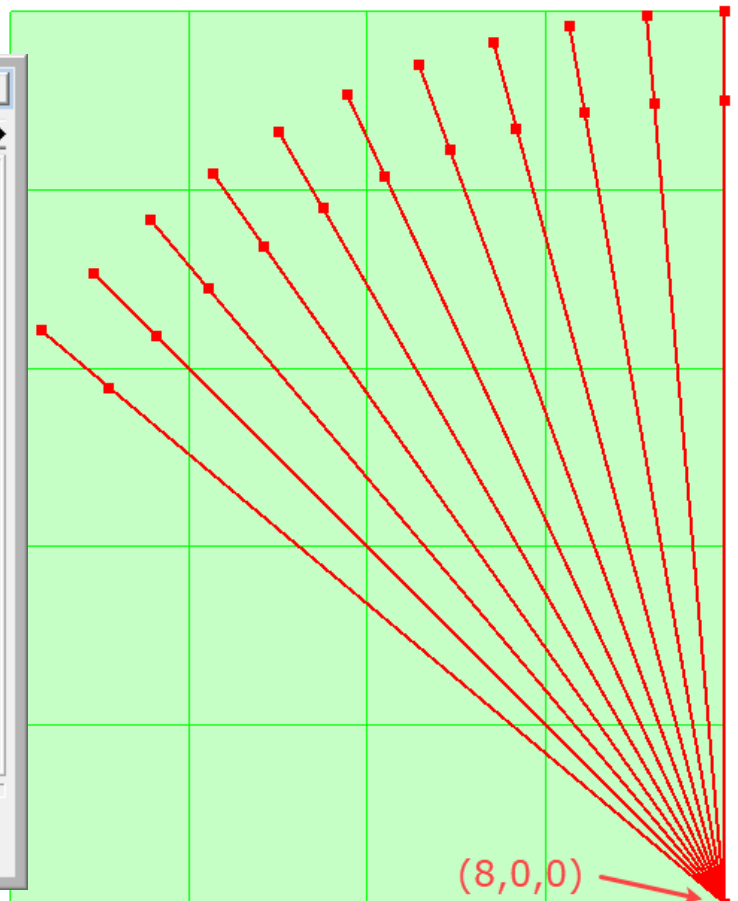
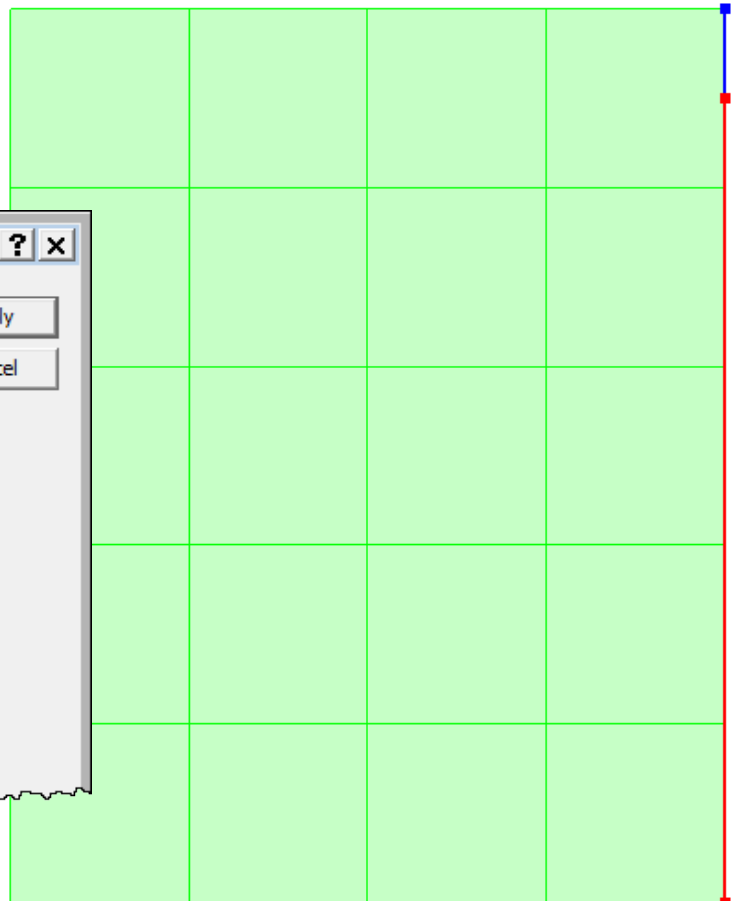
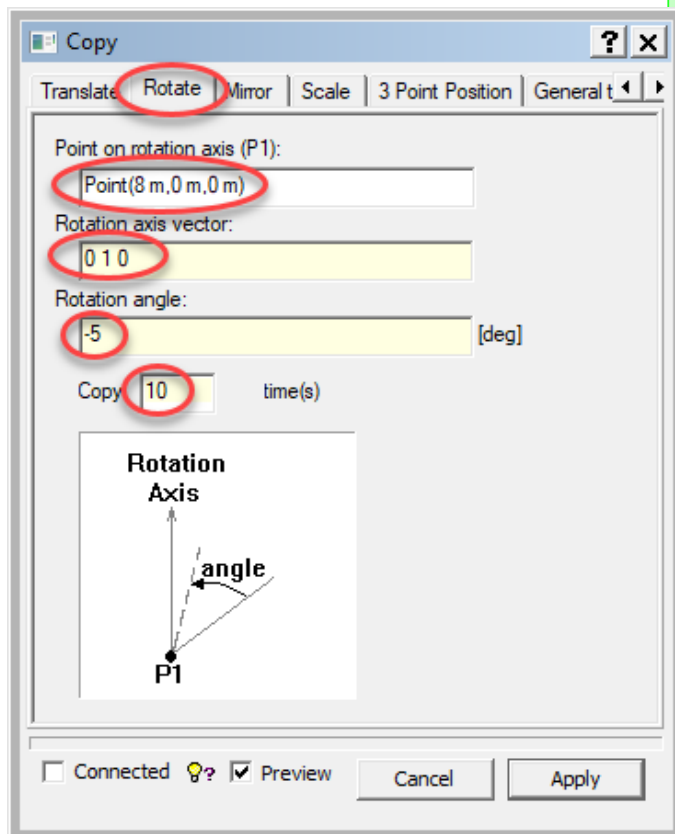
- Having created the guide plane press the F7 function key to view the guide plane in positive Y direction.
 - Pressing F7 repeatedly toggles between viewing in positive and negative Y direction.
- Use *Guiding Geometry | Lines | From Two Points* to create a vertical guide line as shown.







- Divide the guide line 1 m from the top. In the *Divide* dialog give 1 or 9 m depending on in which direction the guide line was created.

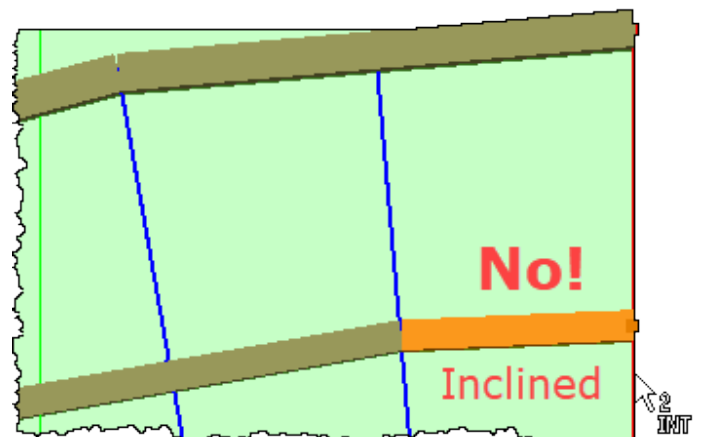
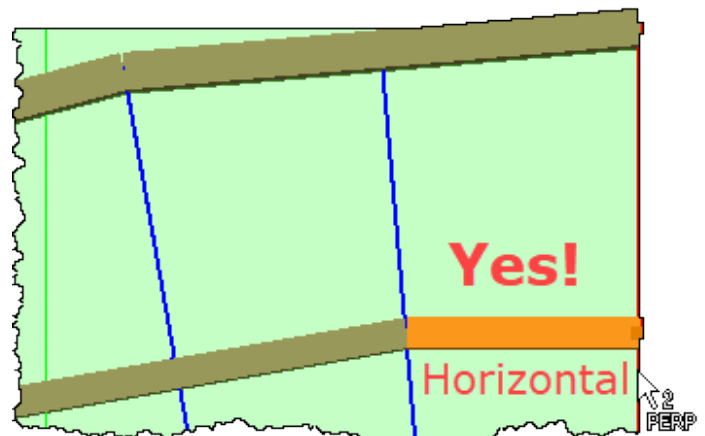
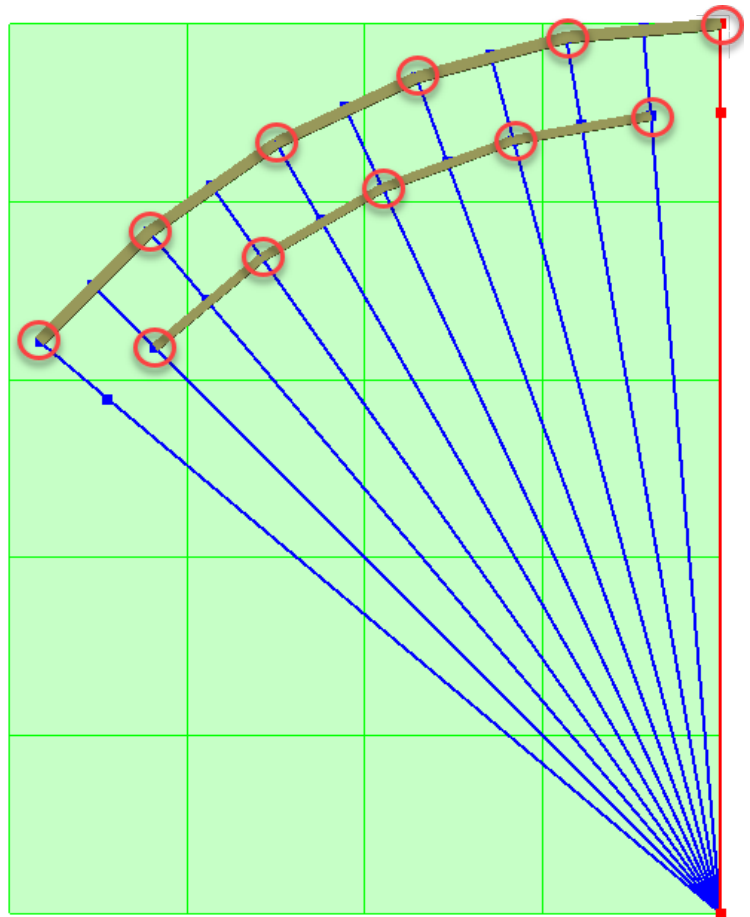



- Copy the guide line split in two by a rotation 10 times as shown.

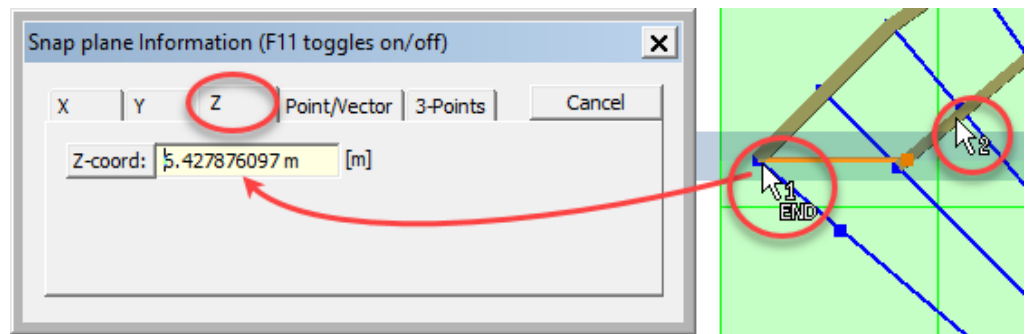


5 UPPER AND LOWER CHORD OF THE ARCH

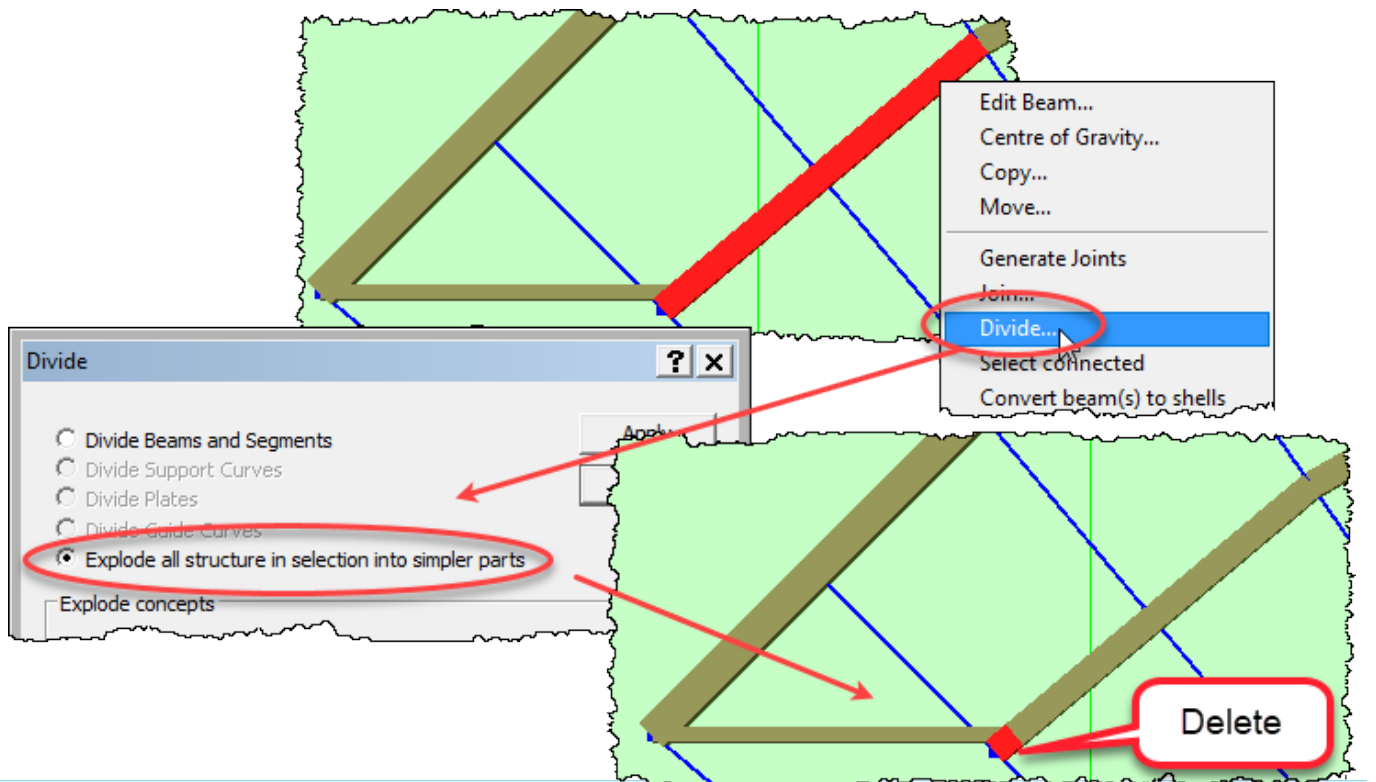
- Create the top chord of the arch.
 - Set material St44 and beam cross section to HSS127X101_6X6_4 as default.
 - Use *Structure | Beams and Piles | Straight Beam* or press  to create beams.
 - To create beams in a row switch from *Snap point* to *Snap point loop* .
 - The ends of the five beams are encircled in the figure to the right.
- Create the lower chord of the arch.
 - Switch to beam cross section HSS101_6X101_6X6_4.
 - To allow clicking the first end of the lower chord, i.e. not connect the last end of the upper chord with the first end of the lower chord press the *Undo snap point* button .
 - The ends of the four beams are encircled in the figure.
- The upper and rightmost piece of the lower chord which is half the length of the other lower chord pieces, should be horizontal. Click its left end and before clicking the other end press the *Snap perpendicular* button . Position the second end of the beam by clicking the vertical guide line as shown.
 - Without the *Snap perpendicular* button the beam would snap to the split between the two guide lines as shown to the right. This is not what we want.



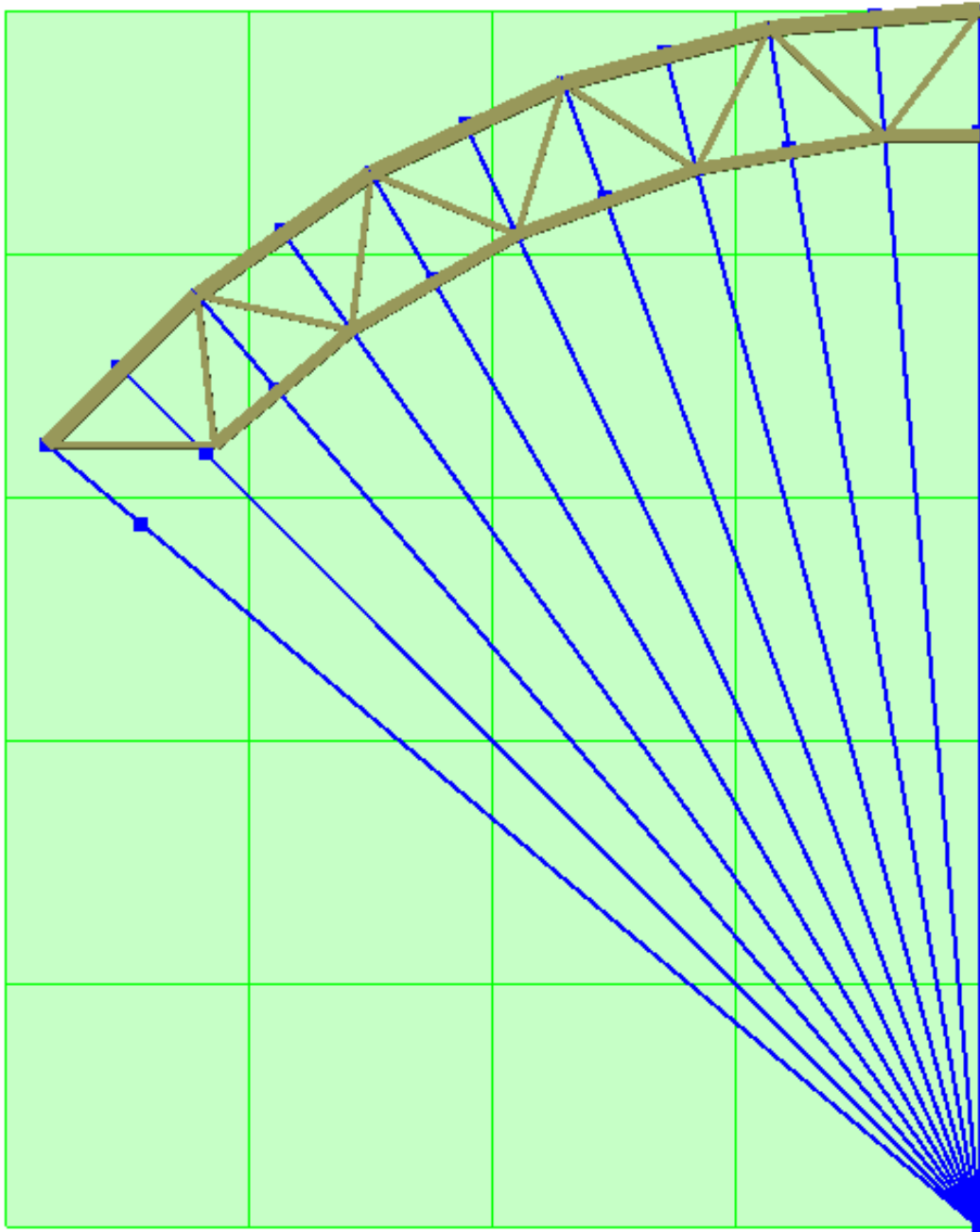
- The left and lower end of the lower chord should be level (same Z coordinate) with the left and lower end of the upper chord. (Currently the former is approximately at 6.36 m while the latter is at 6.43 m.) These ends should also be connected by a beam with cross section HSS57_2X57_2X6_4, so first set this section as default.
- Create a beam by clicking the left and lower end of the upper chord as the first end of the beam.
- Before clicking the second end press the *Restrict snapping to plane* button .
- With focus in the *Z-coord* field of the *Snap plane Information* dialog, click the left and lower end of the upper chord to put this Z coordinate into the dialog thereby defining a snap plane. This plane appears in the display.
- Hover anywhere on the leftmost piece of the lower chord and see that the point at which the snap plane intersects the lower chord is highlighted. Click to create the horizontal beam.



- Having been intersected by the new horizontal beam the leftmost piece of the lower chord is a bit too long. Divide it and delete the superfluous part as shown below.

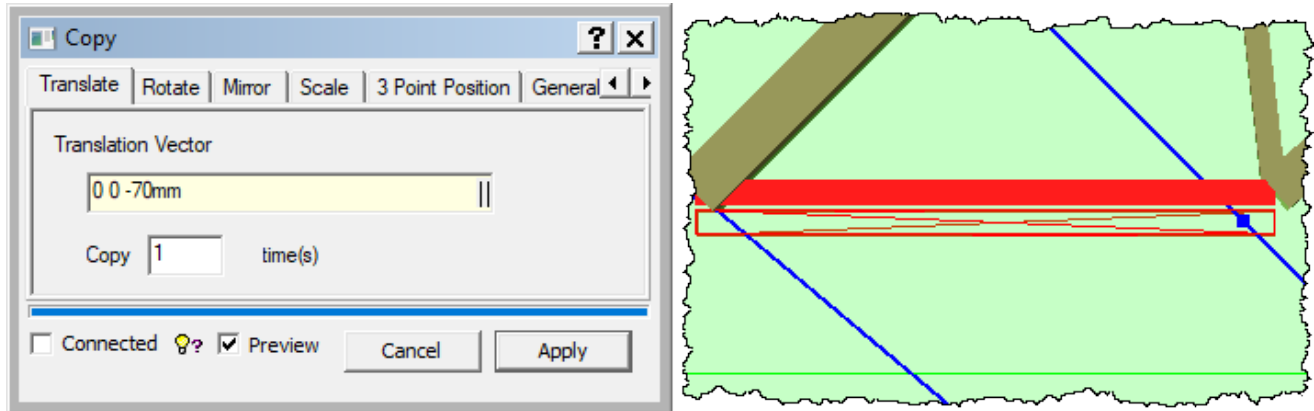


- Create the diagonal braces between the upper and lower chord, all with cross section HSS57_2X57_2X6_4. The model at this point should appear as shown below.

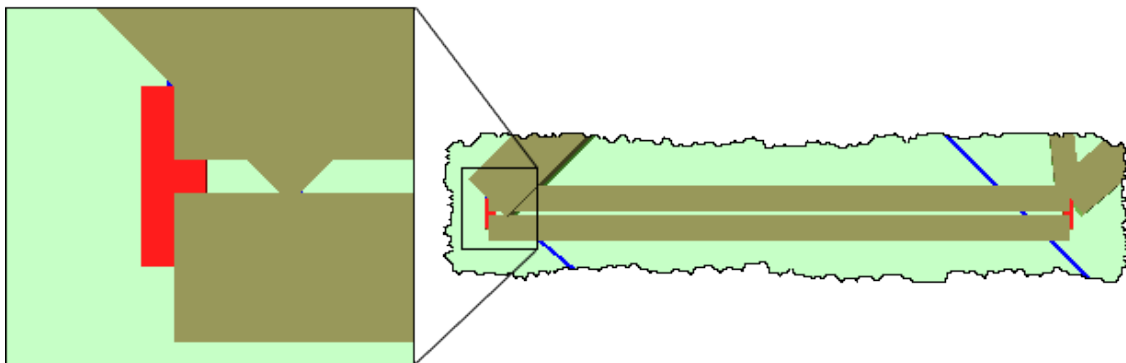


6 COLUMN FRAME

- The column is a separate frame connected to the arch with two bolts.
- The upper horizontal brace of the column frame is 70 mm below the horizontal brace of the arch. Select the brace of the arch and copy it 70 mm downwards.

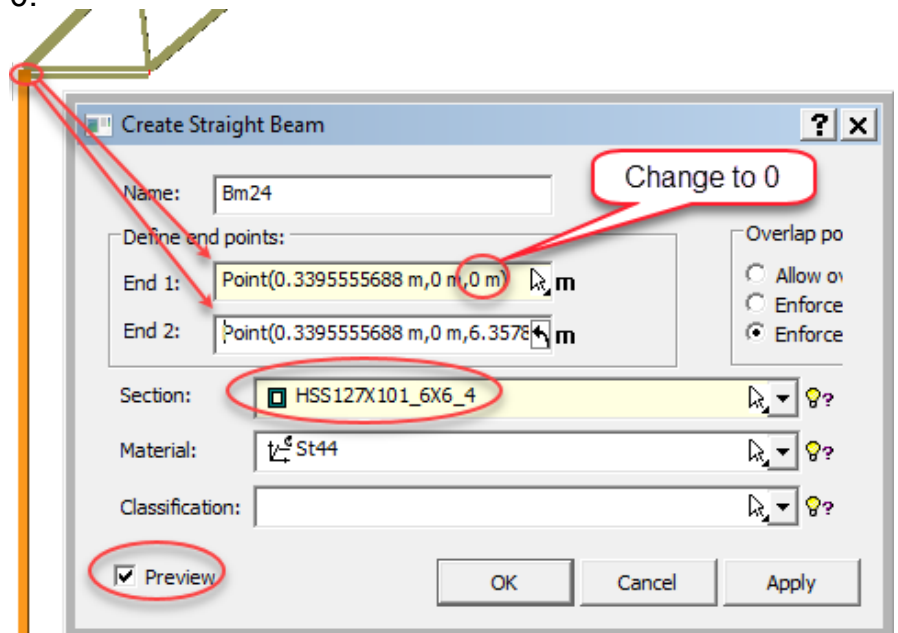


- Create the two bolts as vertical short beam stubs with cross section Bar_25x25 between the ends of the column brace and the arch brace as shown below.



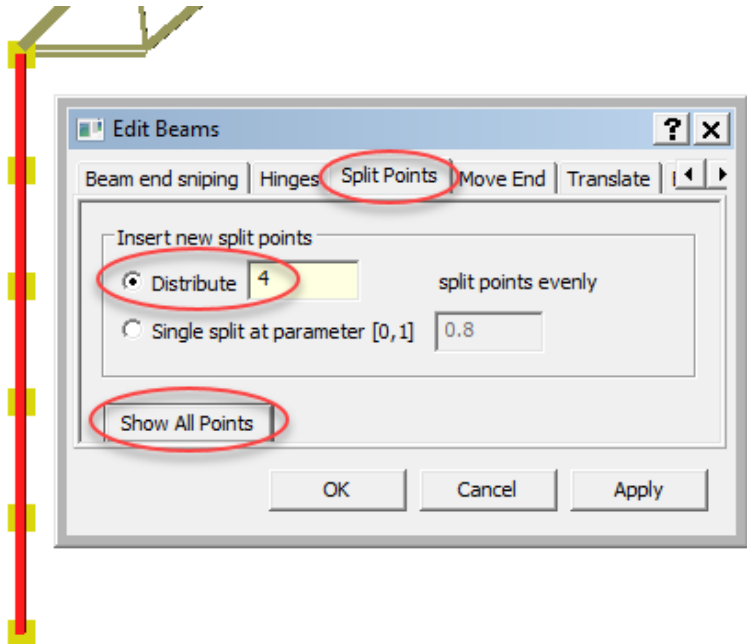
- Create a vertical beam from the left end of the upper horizontal brace of the column frame down to Z coordinate 0.

- Switch to *Modelling – Transparent* to limit snapping to beam ends.
- In the *Create Straight Beam* dialog fetch coordinates for both beam ends by clicking the left end of the upper horizontal brace of the column and changing the Z coordinate for *End 1* to 0.
- Select section HSS101_6X101_6X6_4.



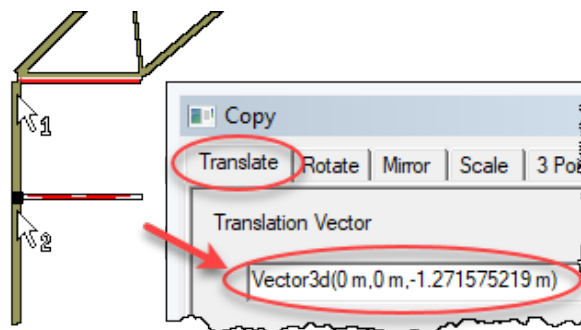
- The column frame has five equally spaced horizontal braces. Select the vertical beam, right-click and select *Edit Beam*. Go to the *Split Points* tab. Insert four equally spaced split points along the beam. Click *Apply*.

- Click *Show All points* to display the split points as yellow squares.

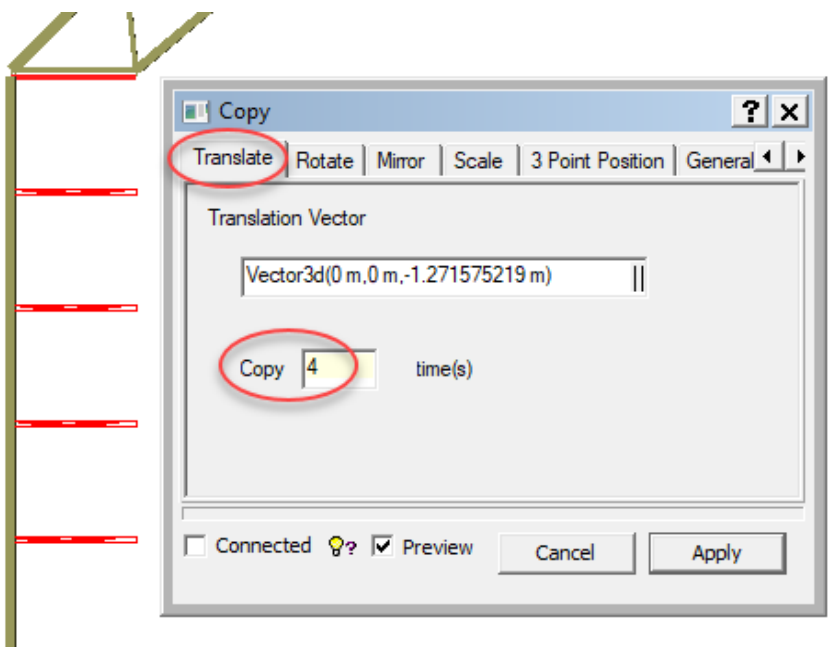


- Copy the upper horizontal brace of the column frame downwards to each of the four split points.

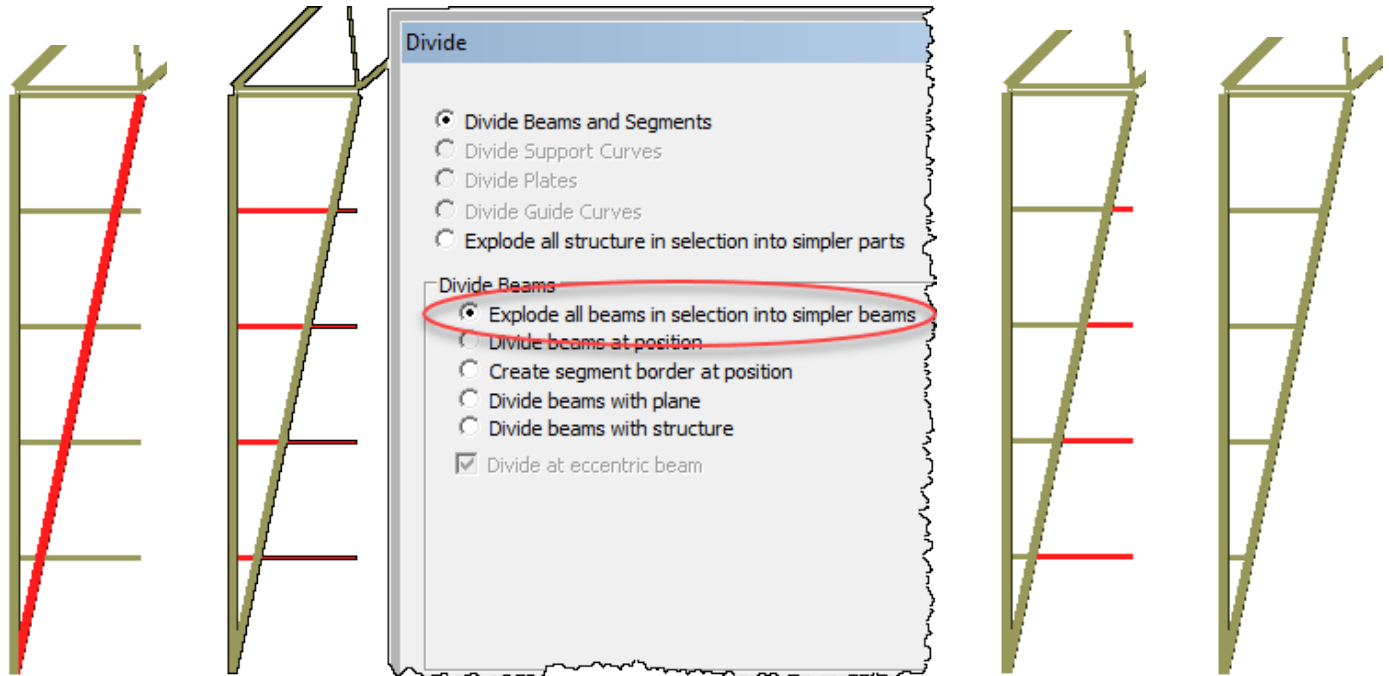
- Having selected the upper horizontal brace of the column, fetch the copy vector by clicking the upper end of the vertical beam and the split point just below as shown to the right.



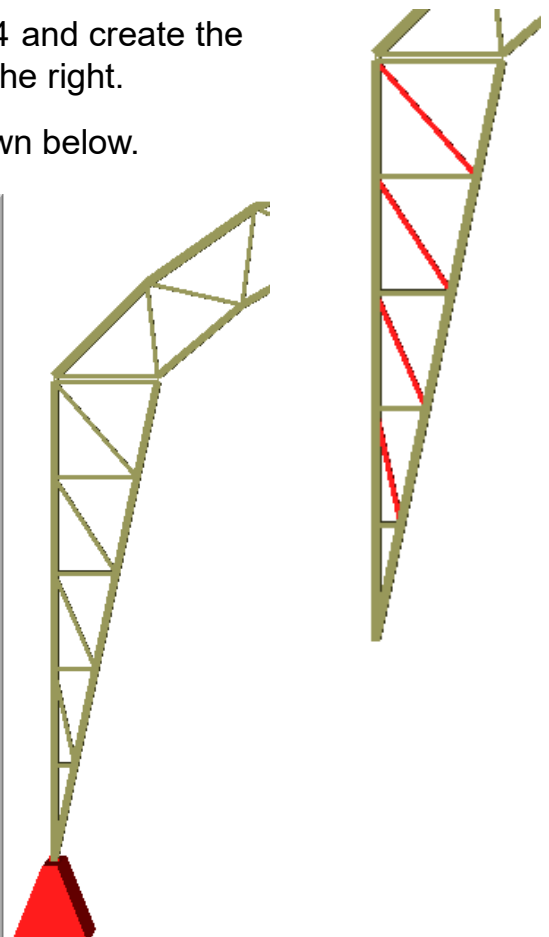
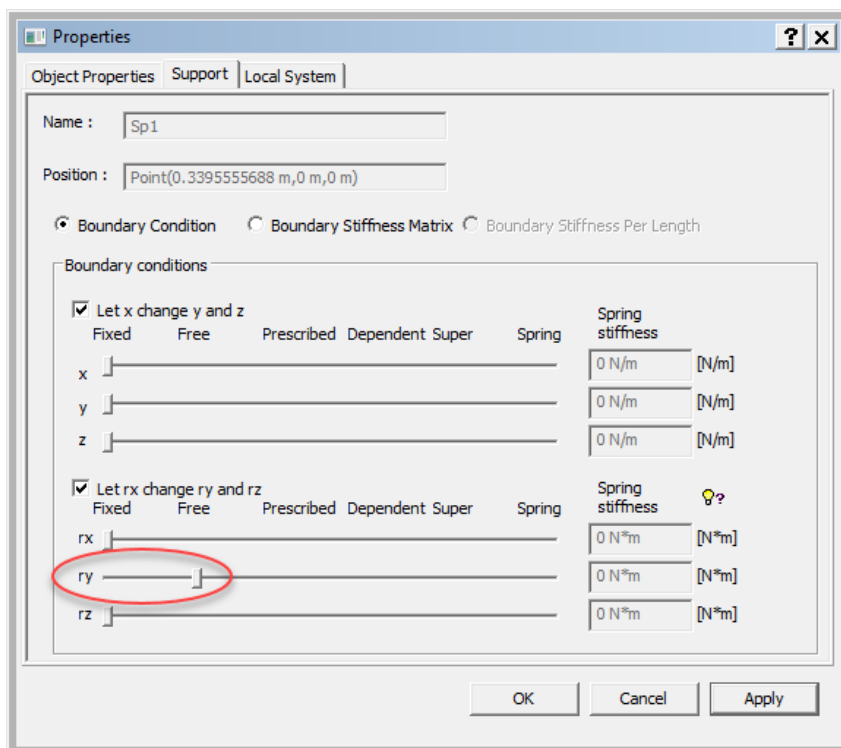
- Then change the number of copies to 4 and click *Apply*.



- Create the slanting beam shown below to the left.
- Split the horizontal braces and delete the protruding parts. The procedure is illustrated below.

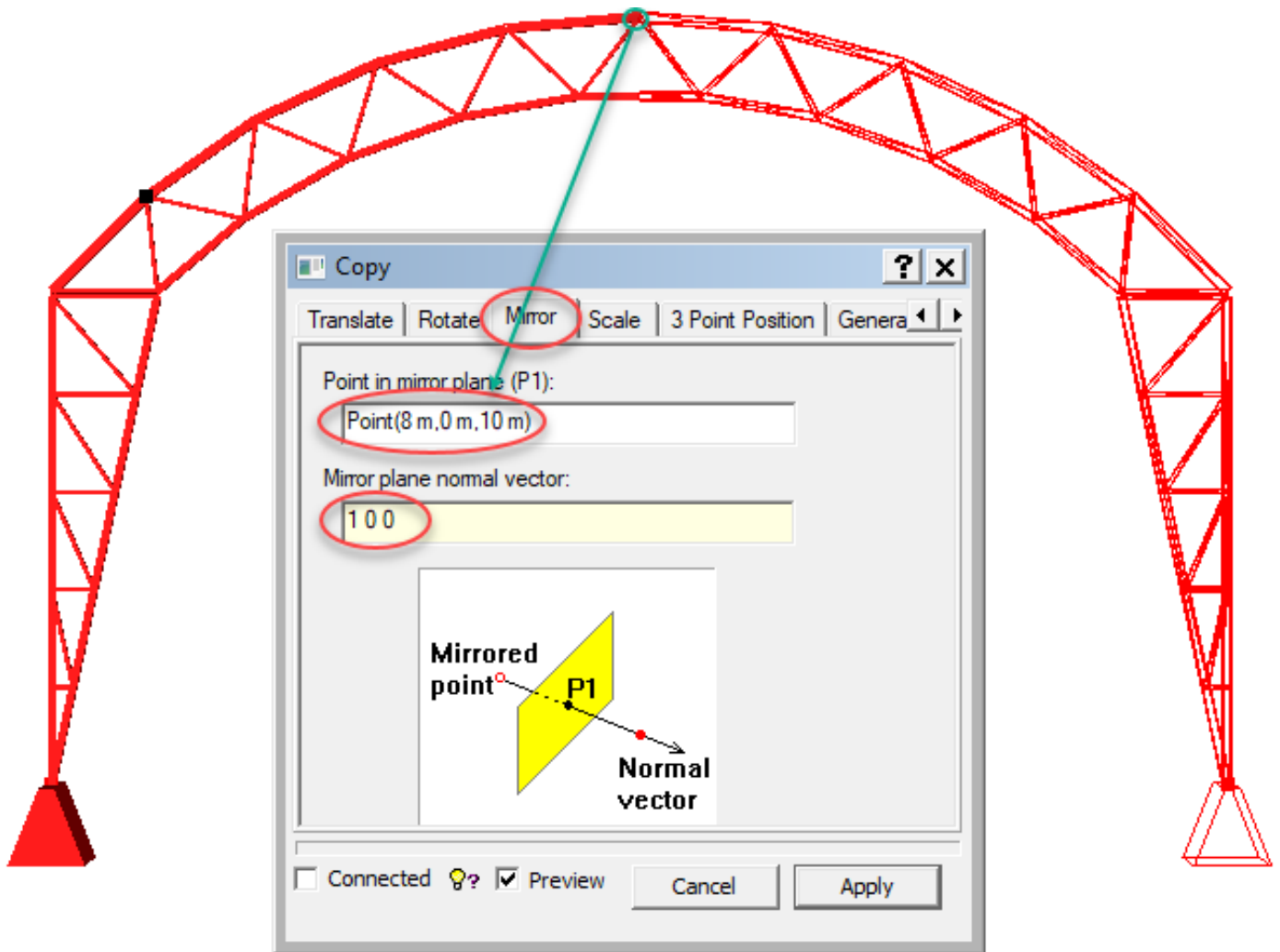


- Switch to beam cross section HSS57_2X57_2X6_4 and create the diagonal braces of the column frame as shown to the right.
- Create a support with rotation about Y free as shown below.

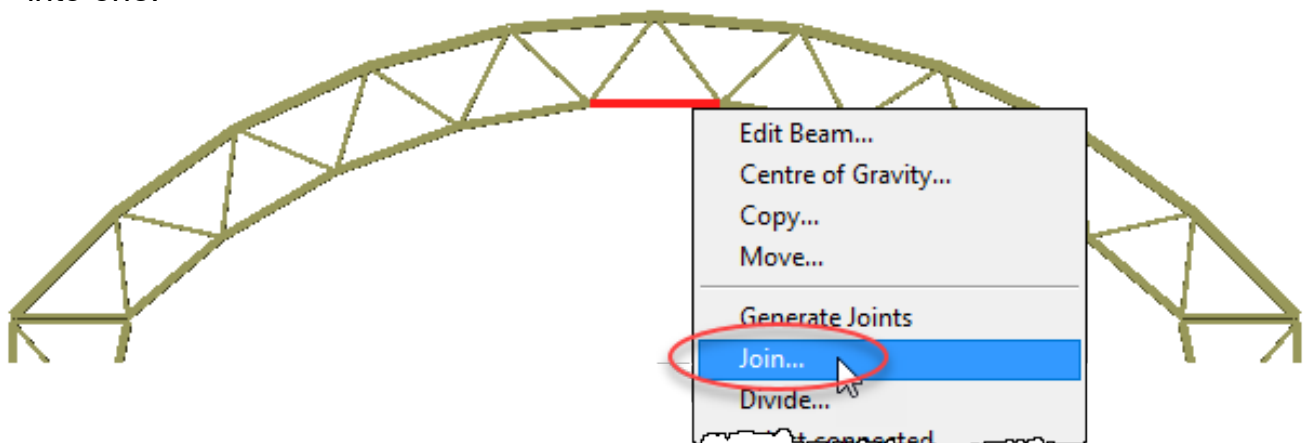


7 COMPLETE MODEL

- Copy by mirroring to create the complete model.



- The upper horizontal part of the lower chord is composed of two beams. Join these into one.



- Do a model verification by *Structure | Topology | Verify Model*.

8 LOADS

➤ Create three load cases:

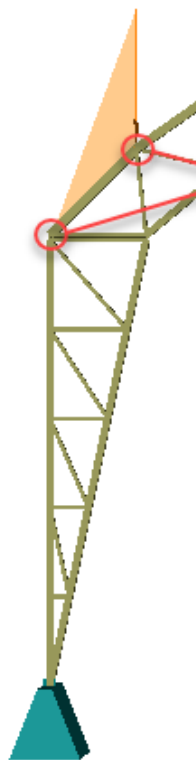
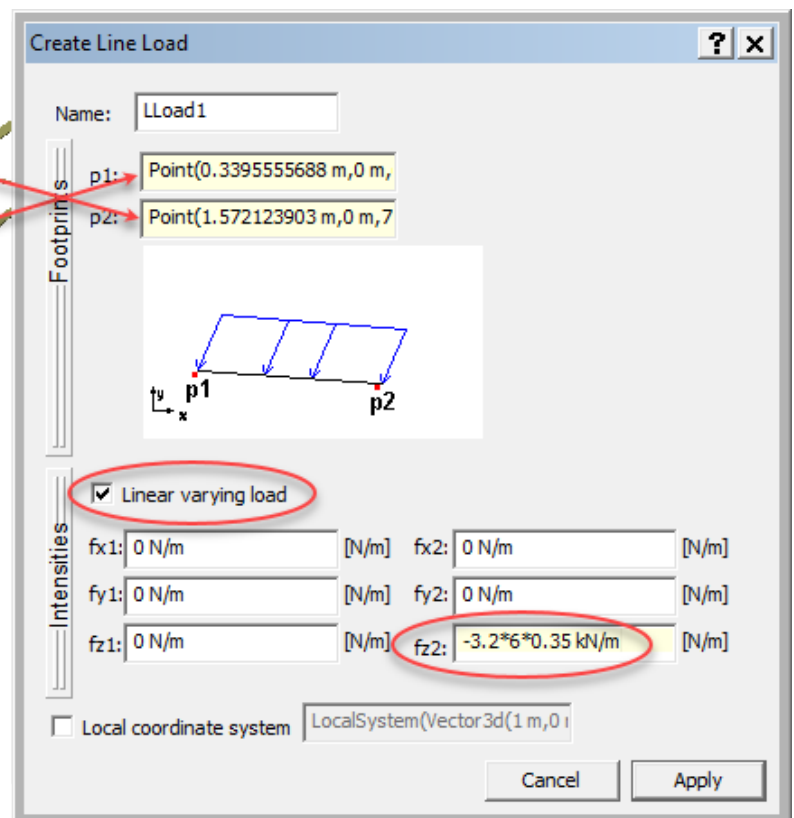
- SelfWeight
- Snow
- Wind

➤ Make SelfWeight a gravity load case by checking *Include structure self-weight in structural analysis* in the property dialog.

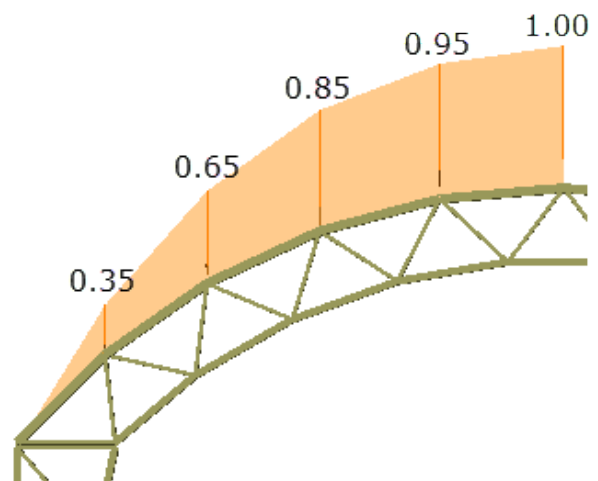
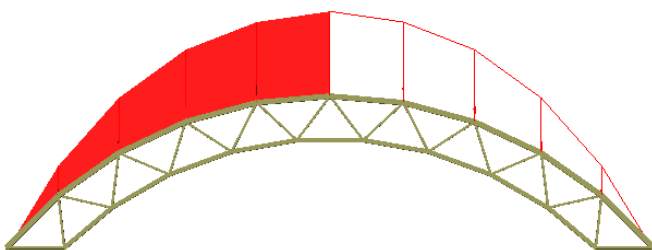
➤ A roof is supported by several arches with a span of 6 m between the arches.

Snow exerts a pressure of 3.2 kN/m^2 on top of the roof. The snow depth decreases gradually to 0 towards the edges of the roof. A shape factor is used to express the snow depth.

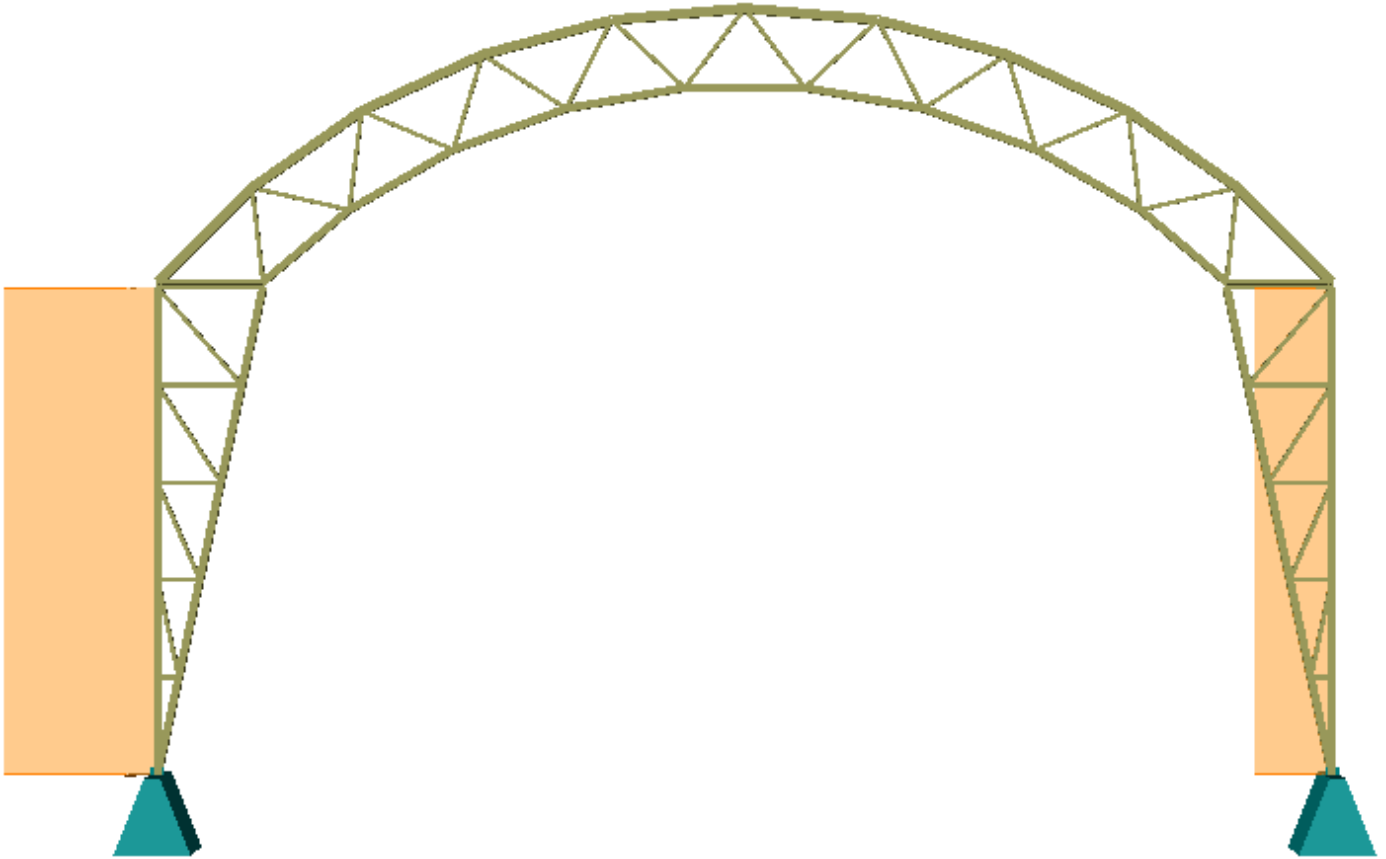
- Set Snow as the current load case.
- Use **Loads | Explicit Load | Line Load** to create a linearly varying line load as shown. The shape factor at the upper point is 0.35.

- Create line loads for the four remaining pieces of the upper chord using shape factors as shown to the right.
- Copy by mirroring the snow load to the other side of the roof as shown below.



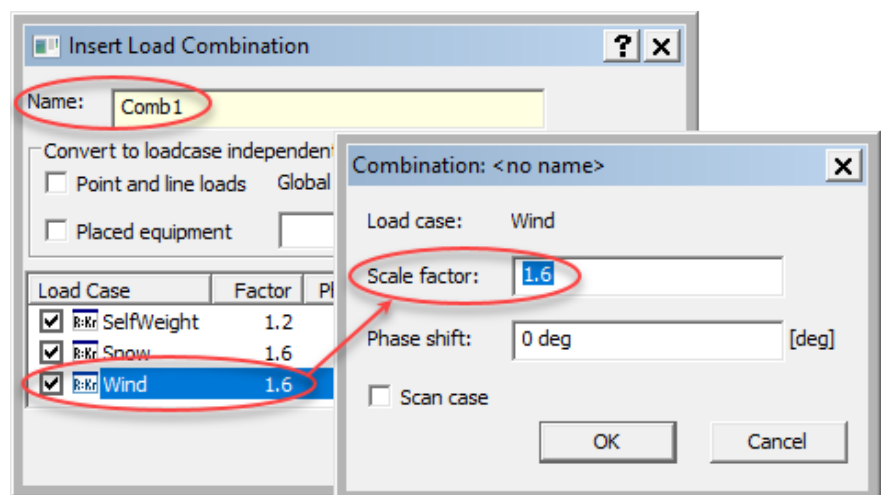
- Wind exerts a constant pressure of 1.3 kN/m^2 on the left wall and a suction of half of this value on the right wall.
 - Set Wind as the current load case.
 - Create a constant line load of $1.3 \cdot 6 \text{ kN/m}$ on the left wall and $1.3 \cdot 6 \cdot 0.5 \text{ kN/m}$ on the right wall.



- Use *Loads | Load Combination* to create a combination named Comb1 of all load cases with the following scaling factors:

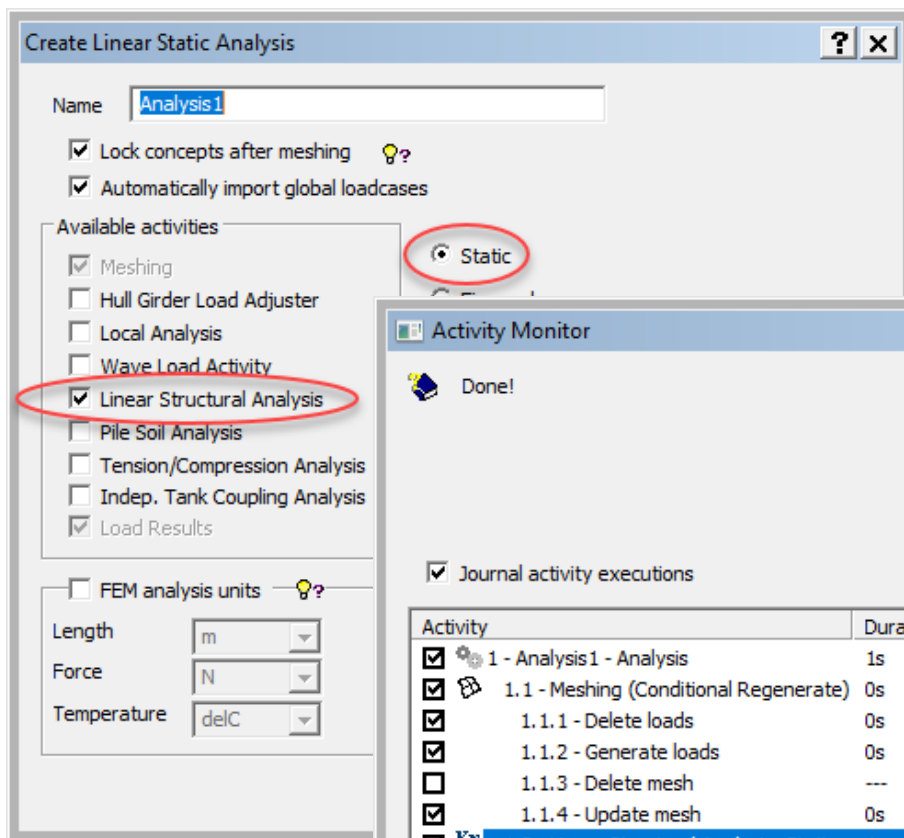
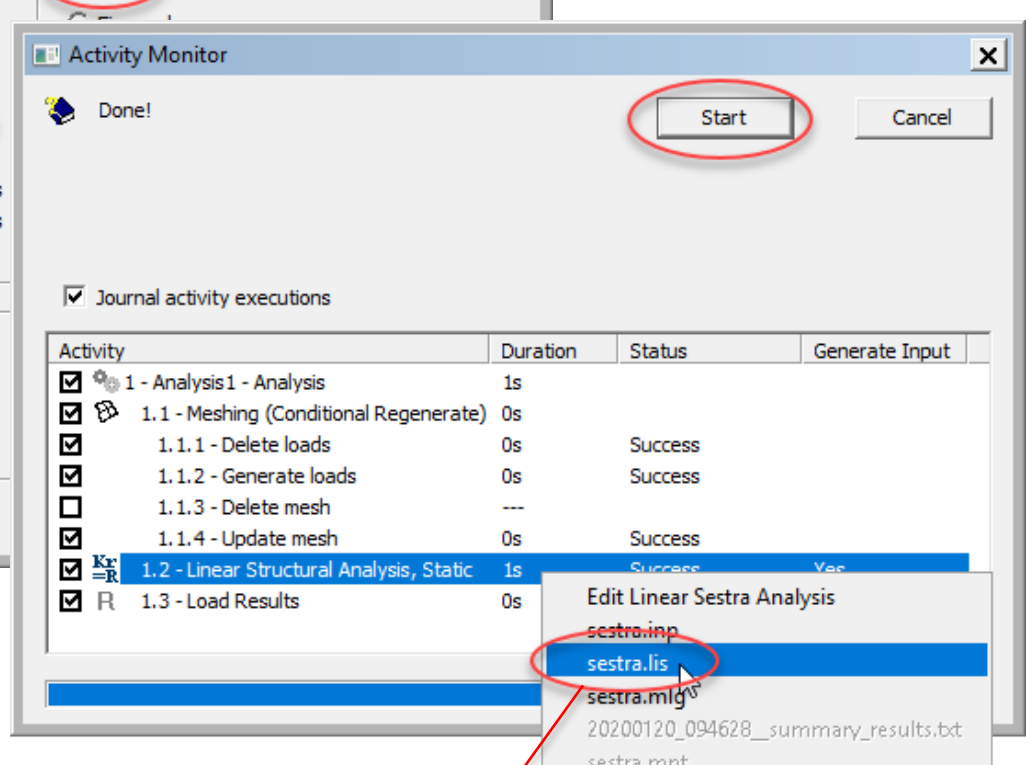
- SelfWeight 1.2
- Snow 1.6
- Wind 1.6

- Double-click a load case in the *Insert Load Combination* dialog to adjust its scaling factor.



9 STRUCTURAL ANALYSIS

- Do a linear structural analysis (Alt+D).
- Make sure the analysis succeeds and open the Sestra.lis file to ensure the sums of loads and reaction forces are zero for all load cases, i.e. there is balance between applied loads and reactions.
 - Notice that only the three load cases are listed and not the load combination. This is because so-called smart load combinations are by default used involving that the combination is made based on results rather than on loads. (This can be done for linear but not for non-linear analyses.)

Activity	Duration	Status	Generate Input
1 - Analysis1 - Analysis	1s		
1.1 - Meshing (Conditional Regenerate)	0s		
1.1.1 - Delete loads	0s	Success	
1.1.2 - Generate loads	0s	Success	
1.1.3 - Delete mesh	---		
1.1.4 - Update mesh	0s	Success	
1.2 - Linear Structural Analysis, Static	1s	Success	Yes
1.3 - Load Results	0s		


Context menu for '1.2 - Linear Structural Analysis, Static':

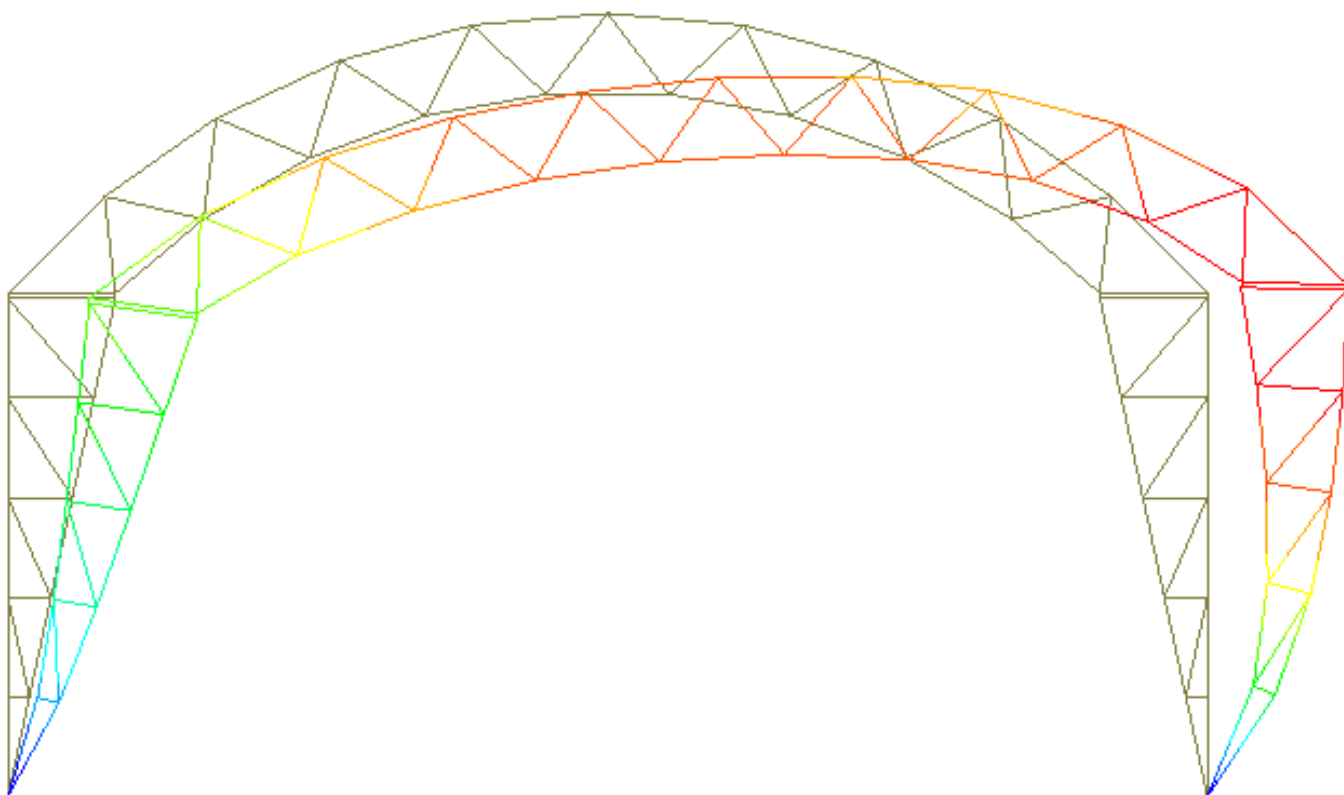
- Edit Linear Sestra Analysis
- sestra.inp
- sestra.lis** (highlighted)
- sestra.mlg
- 20200120_094628_summary_results.txt
- sestra.mnt

Sum of load sum and reaction force sum for all result cases (internal number):

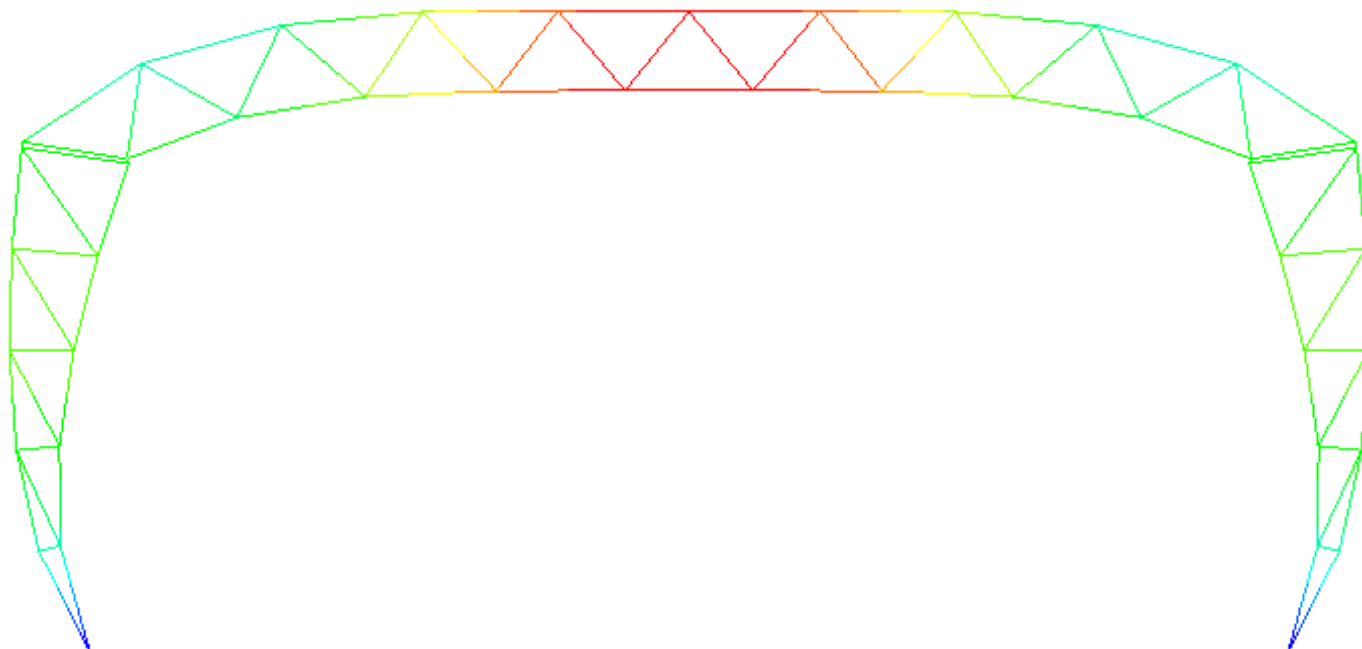
result case;	tx;	ty;	tz;	rx;	ry;	rz;	magnitude
1;	1.896812e-10;	0.000000e+00;	3.274181e-11;	0.000000e+00;	8.003553e-10;	0.000000e+00;	8.231766e-10
2;	6.950472e-09;	0.000000e+00;	9.022187e-10;	0.000000e+00;	3.422610e-08;	0.000000e+00;	3.493636e-08
3;	-3.958121e-08;	-6.486961e-26;	-4.380126e-09;	4.198759e-25;	-2.291927e-07;	-6.472446e-25;	2.326266e-07

10 VIEW RESULTS


- Select the *Results - All* display configuration, open the *Result presentation* dialog by Alt+P and select displacement component *All*. To see the deformed shape on top of the undeformed right-click the *Beam selection* button  to open the eye symbol. Also switch to wireframe display of beams. The deformation shown below is for the load combination.

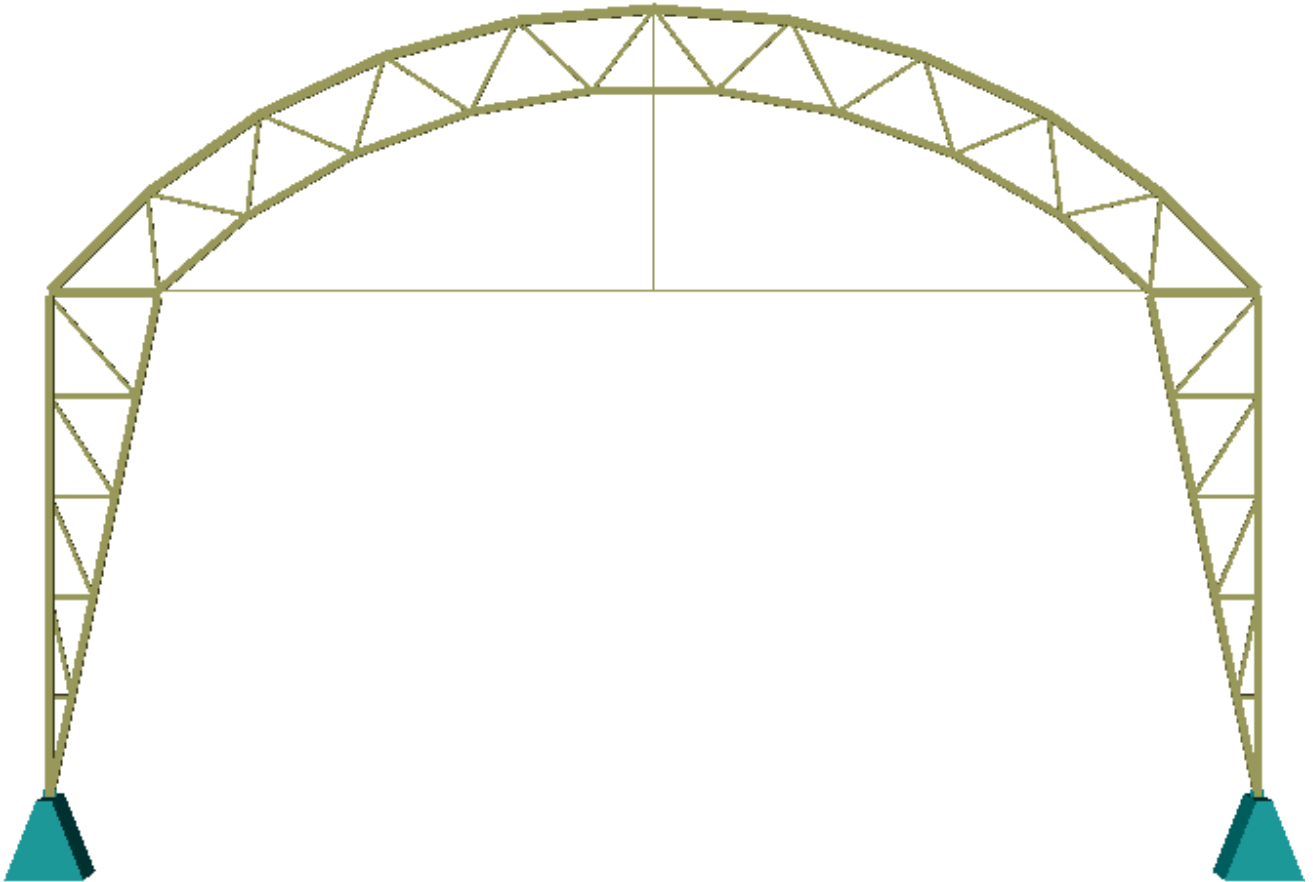


- Switch to load case Snow and see the deformation due to snow alone. The deformation at the middle of the roof is approximately 0.014 m.

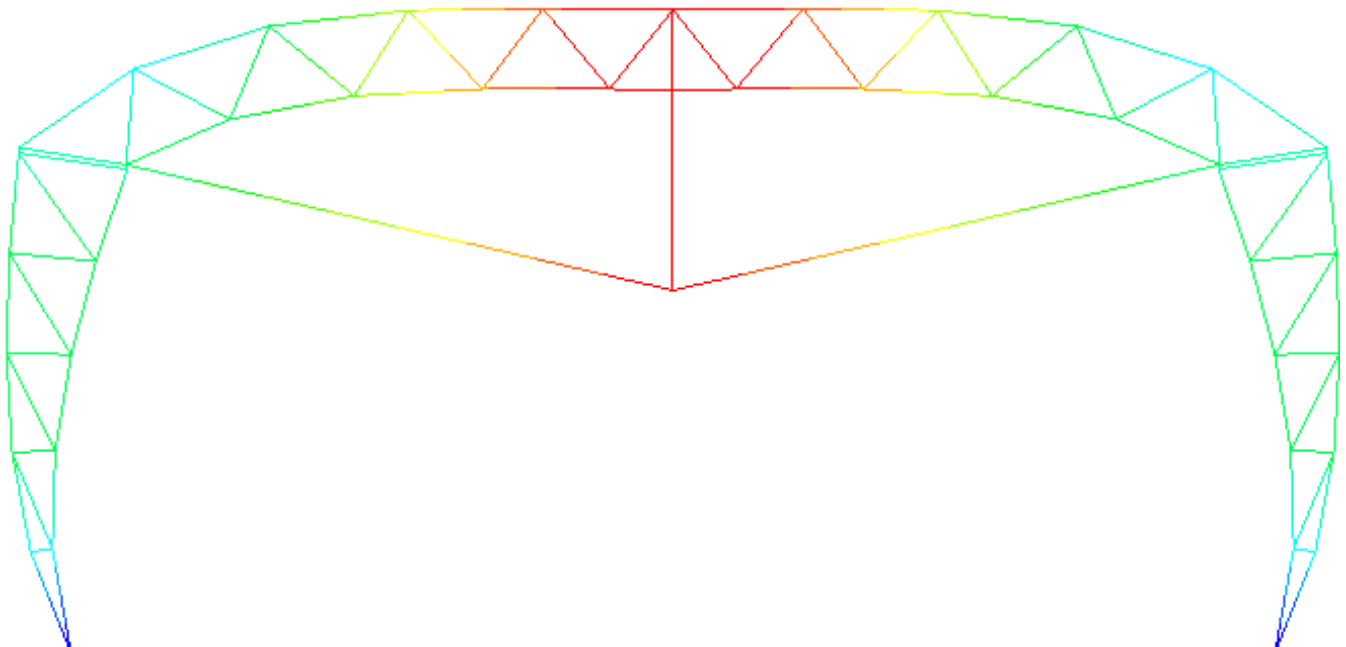


11 REDESIGN AND REANALYSIS

- To limit the deformation of the roof, add a horizontal tie bar using beam cross section Bar_25X25 as shown below. To prevent the tie bar from sagging also add a vertical bar as shown. The vertical bar is easily added by letting its first end be at top and using the *Snap perpendicular* button  to position the lower end.



- Rerun the analysis and see that the deformation at the middle of the roof is reduced





About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.