SESAM TUTORIAL

GeniE

Transportation With Contact

Valid from program version 8.3

Sesam Tutorial

GeniE – Transportation With Contact

Date: November 2021

Valid from GeniE version 8.3

Prepared by: Digital Solutions at DNV

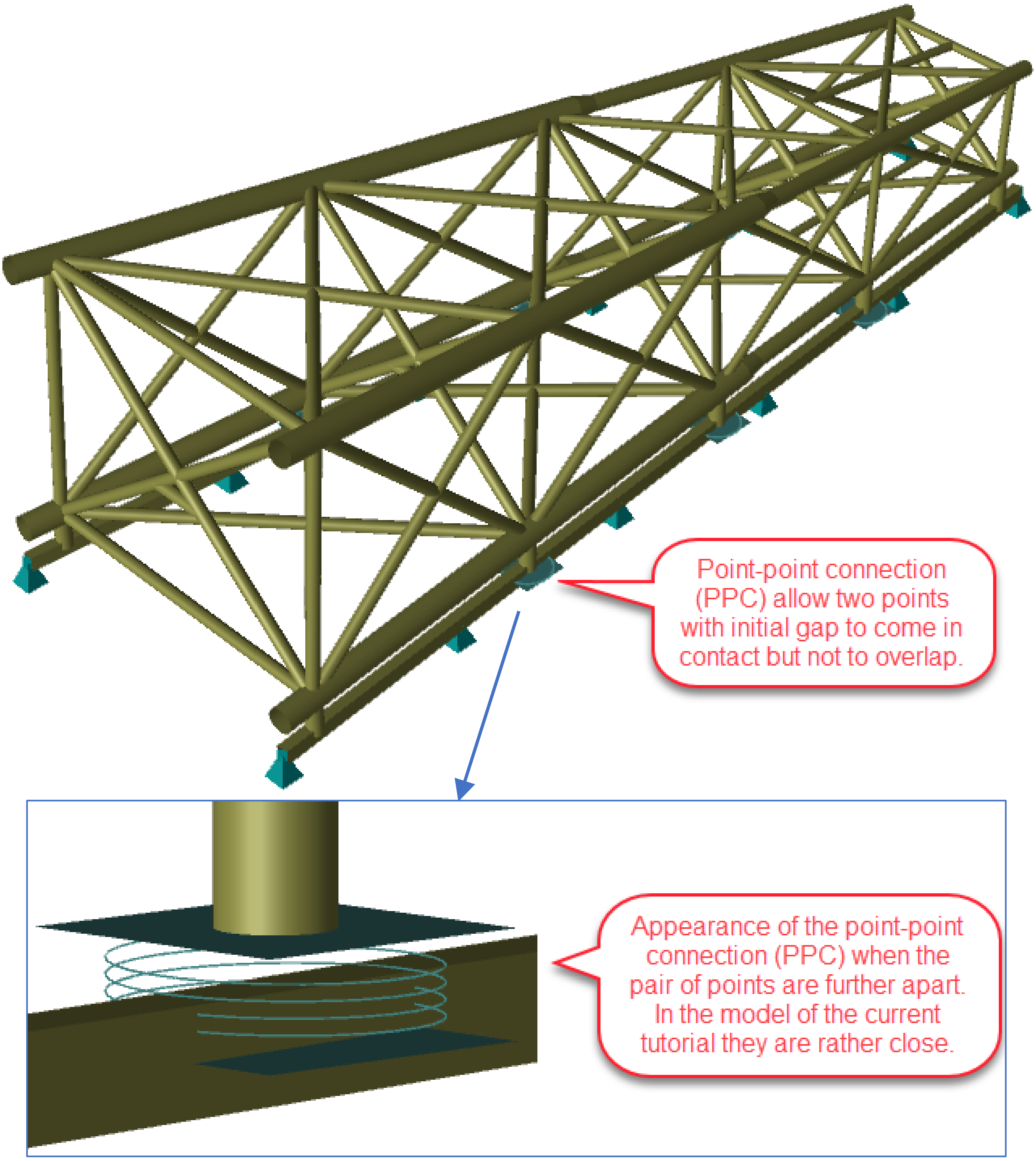
E-mail support: software.support@dnv.com

E-mail sales: digital@dvn.com

TABLE OF CONTENTS

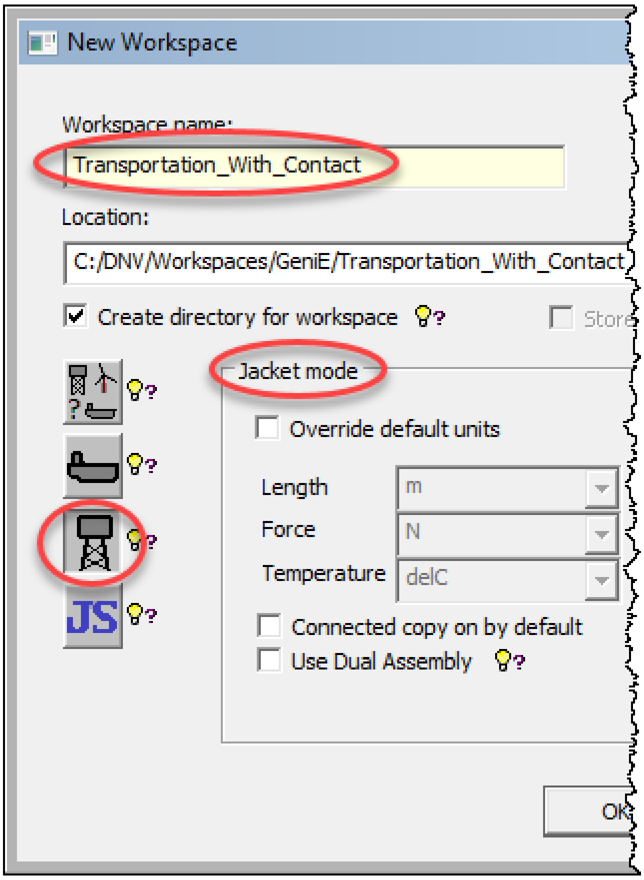
1 INTRODUCTION

- This tutorial explains how to perform a gap/contact analysis. This is a non-linear analysis in which so-called point-point-connections (PPCs) are inserted between pairs of points that are spaced slightly apart (initial gaps). The PPCs allow the pairs of points to displace so that the gaps close (become zero) and increase. The gaps are not allowed to become negative.
  - Note that a gap-contact analysis is similar to a tension/compression analysis in which selected beam elements are assigned properties of a truss element that can only take compression or tension. In fact, the Sestra analysis is the same. The difference is merely that in a gap/contact analysis there are PPCs while in a tension/compression analysis there are tension/compression truss elements.

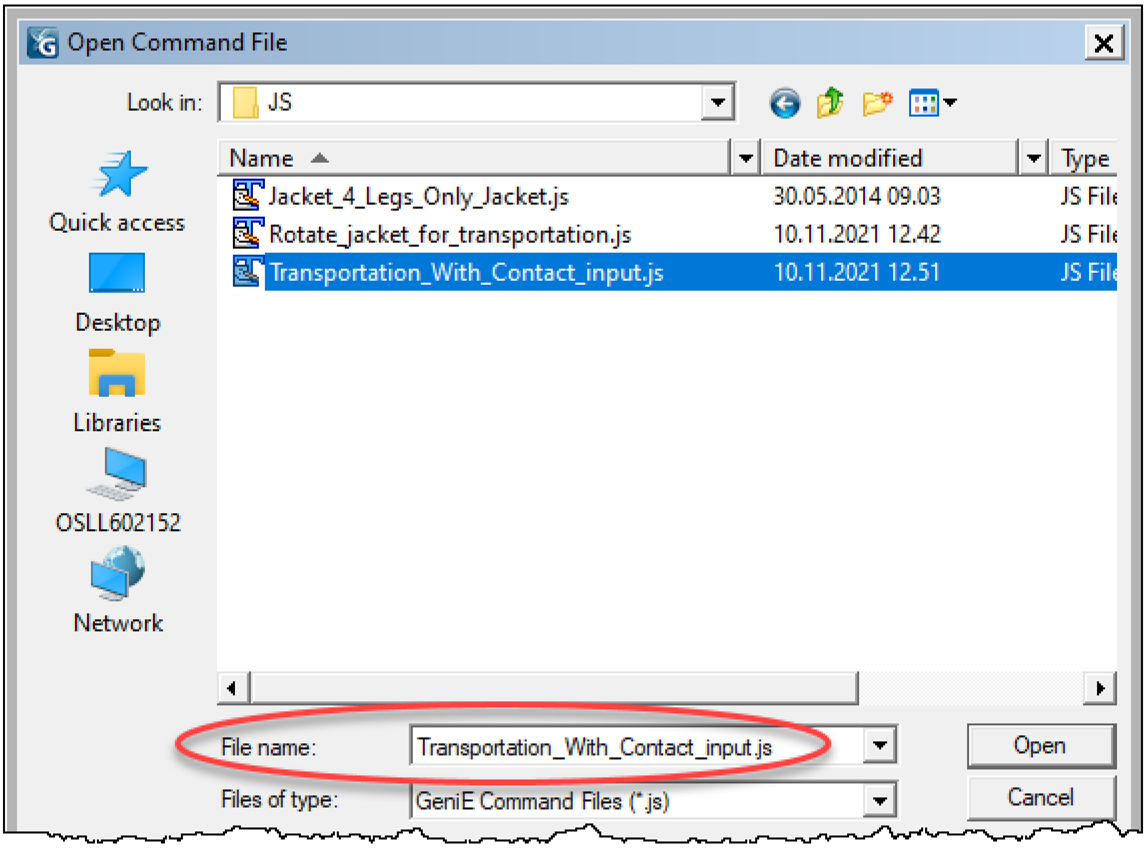


2 CREATE THE MODEL

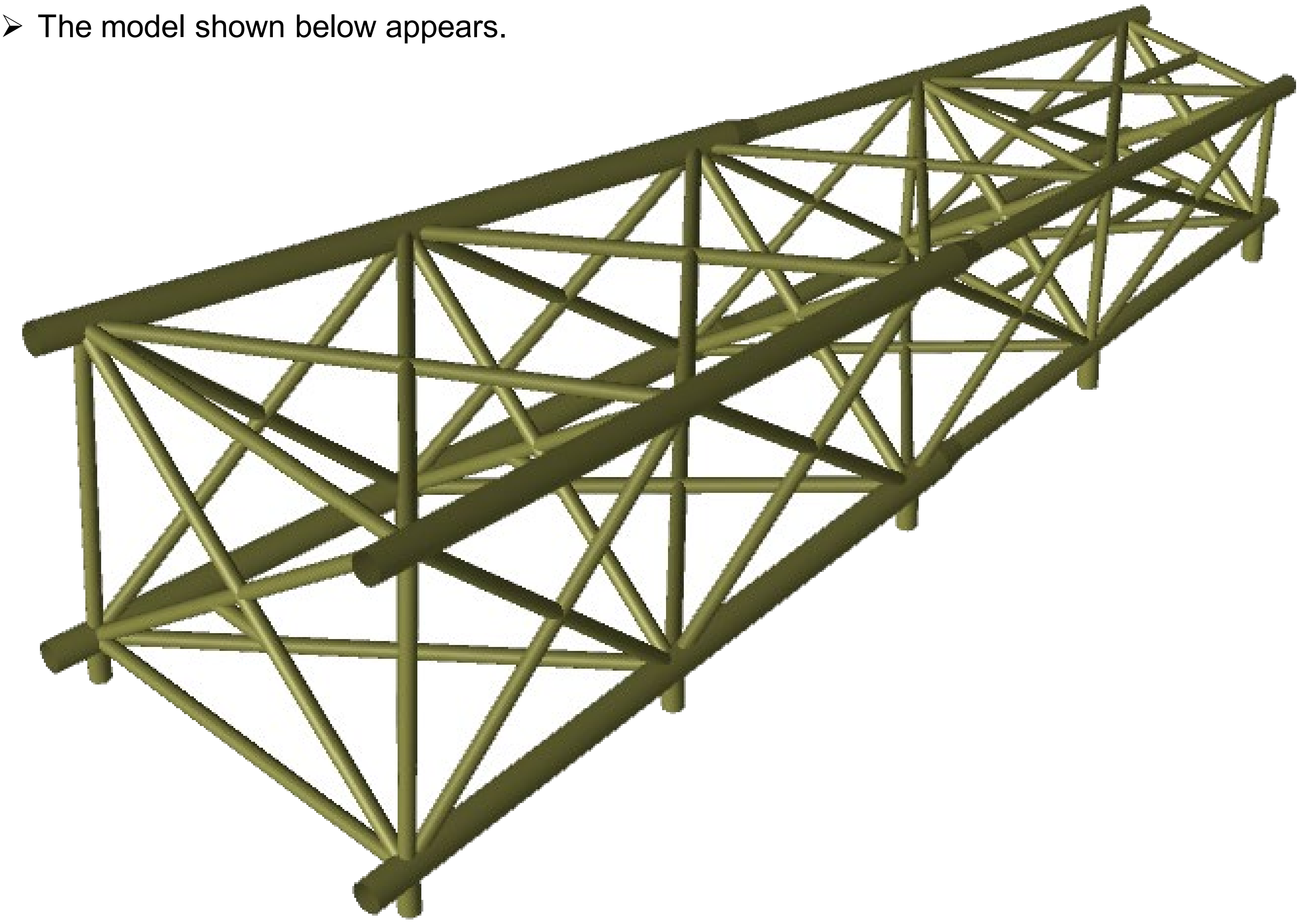
- Start GeniE and open a new workspace.
  - Give a *Workspace name*.
  - Accept default units m and N and click *OK*.
    - Unless otherwise specified, all values in this tutorial are in these units.
  - Select *Jacket mode* to limit menus to those relevant for beam modelling.



- Use *File* | *Read Command File* to read the file Transportation_With_Contact_input.js.
  - The file is found as part of the installation, typically at: C:\Program Files\DNV\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsAdvancedModelling\A8_GeniE_Transportation_With_Contact\JS
  - Transportation_With_Contact_input.js reads the two additional js files Jacket_4_Legs_Only_Jacket.js and Rotate_jacket_for_transportation.js found in the same folder.



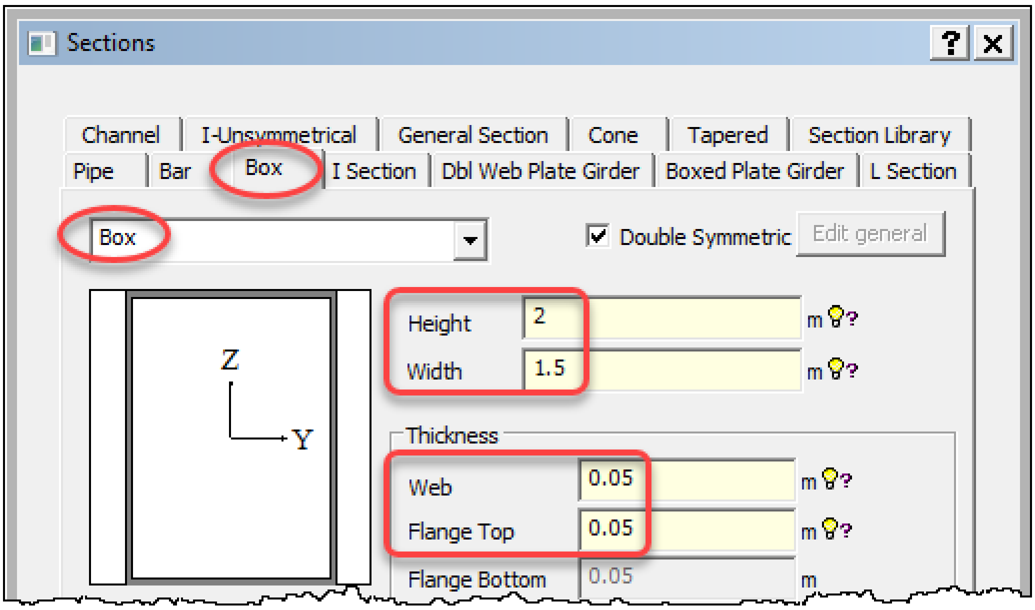
➢ The model shown below appears.



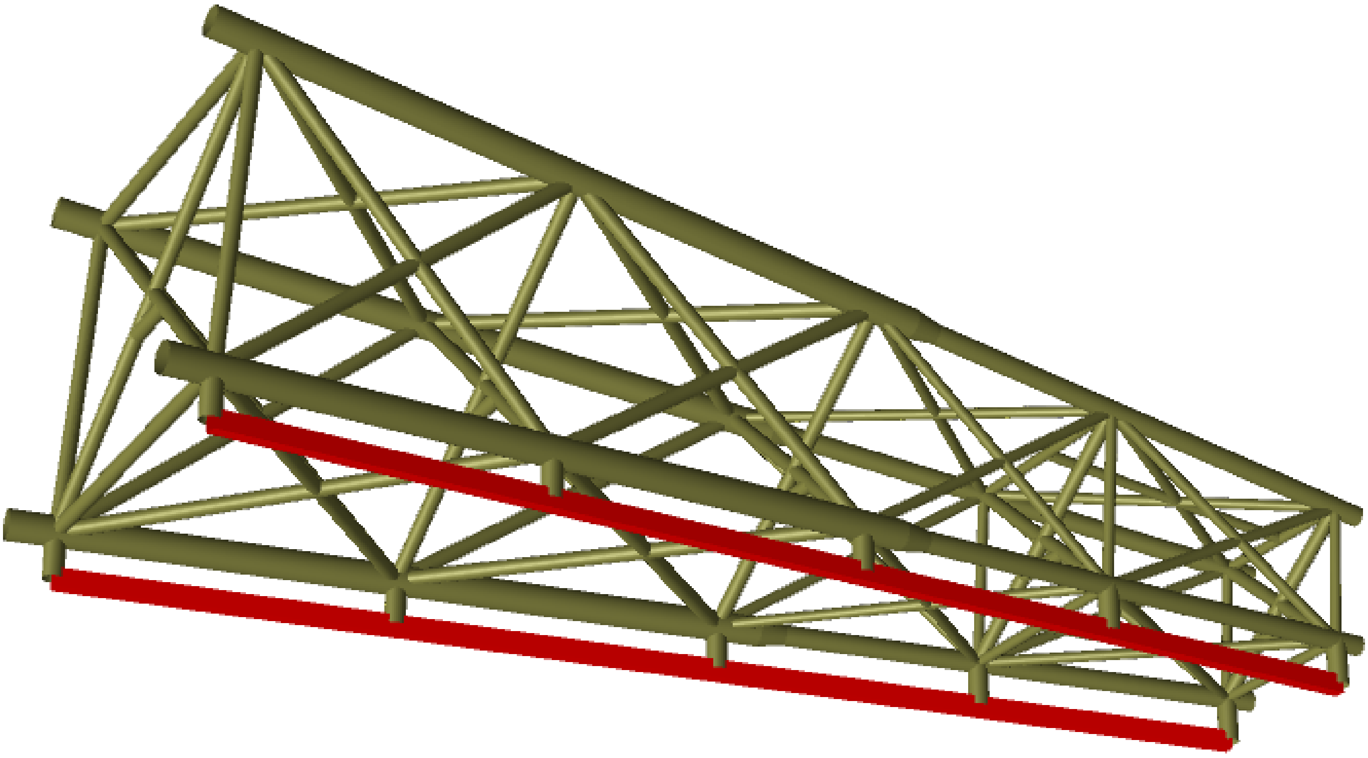
➢ Notice that the model includes the two load cases LCGrav (gravity) and LCRoll (rotational acceleration), the latter representing a case where the model is placed on a barge rolling in the sea. The two load cases are included in a load combination named LCGravAndRoll.

➢ Create two supporting beams for the model. First, create a box cross section for these beams.

- Use *Edit* | *Properties* | *Section* and click *Create/Edit Section* to create the box cross section shown to the right.
- Set this as default cross section.

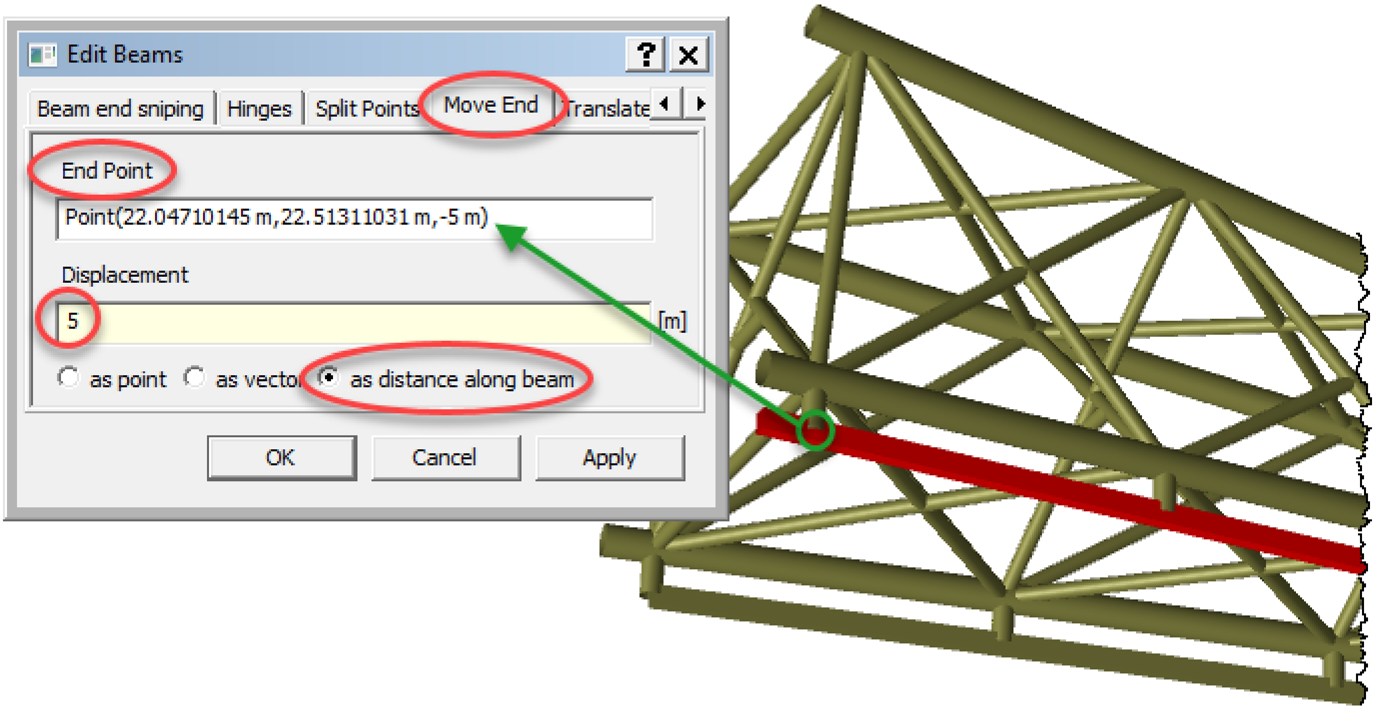


➢ Create the two supporting beams highlighted below.

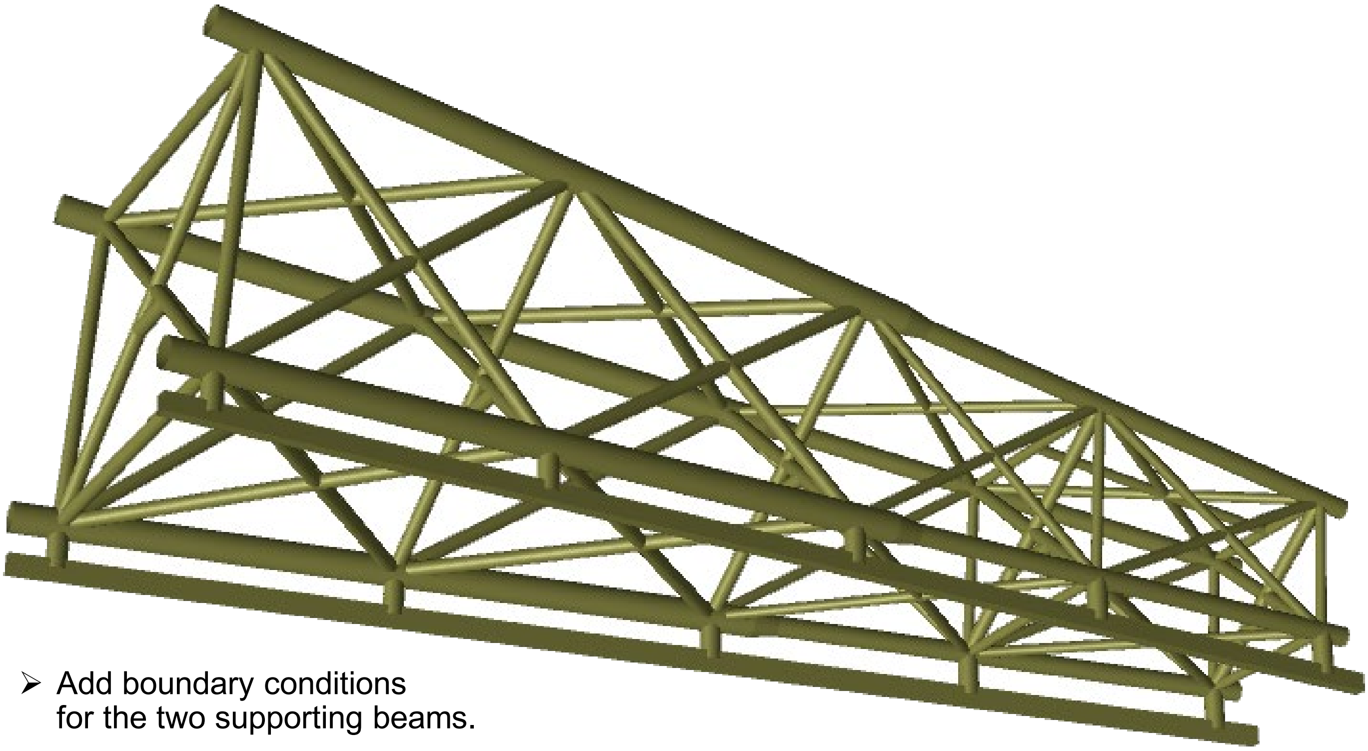


➢ Use *Edit* | *Beam* | *Move End* to extend each supporting beam 5 m in both ends.

- Click the beam end to be extended to fill the *End Point* field, see below.
- Select *as distance along beam* and enter 5 m (positive value extends, negative value shortens).
- One of the extension processes is illustrated below.

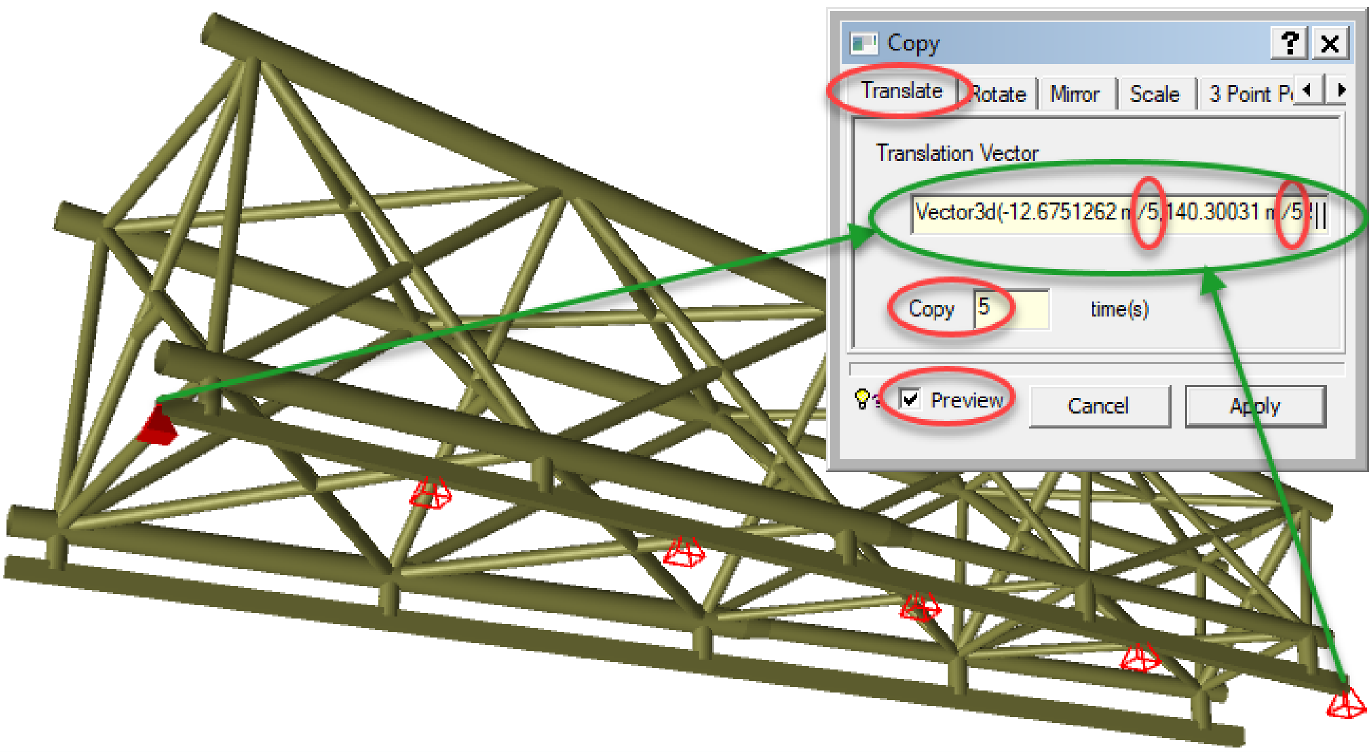


➢ The model after extending all four supporting beam ends is shown below.



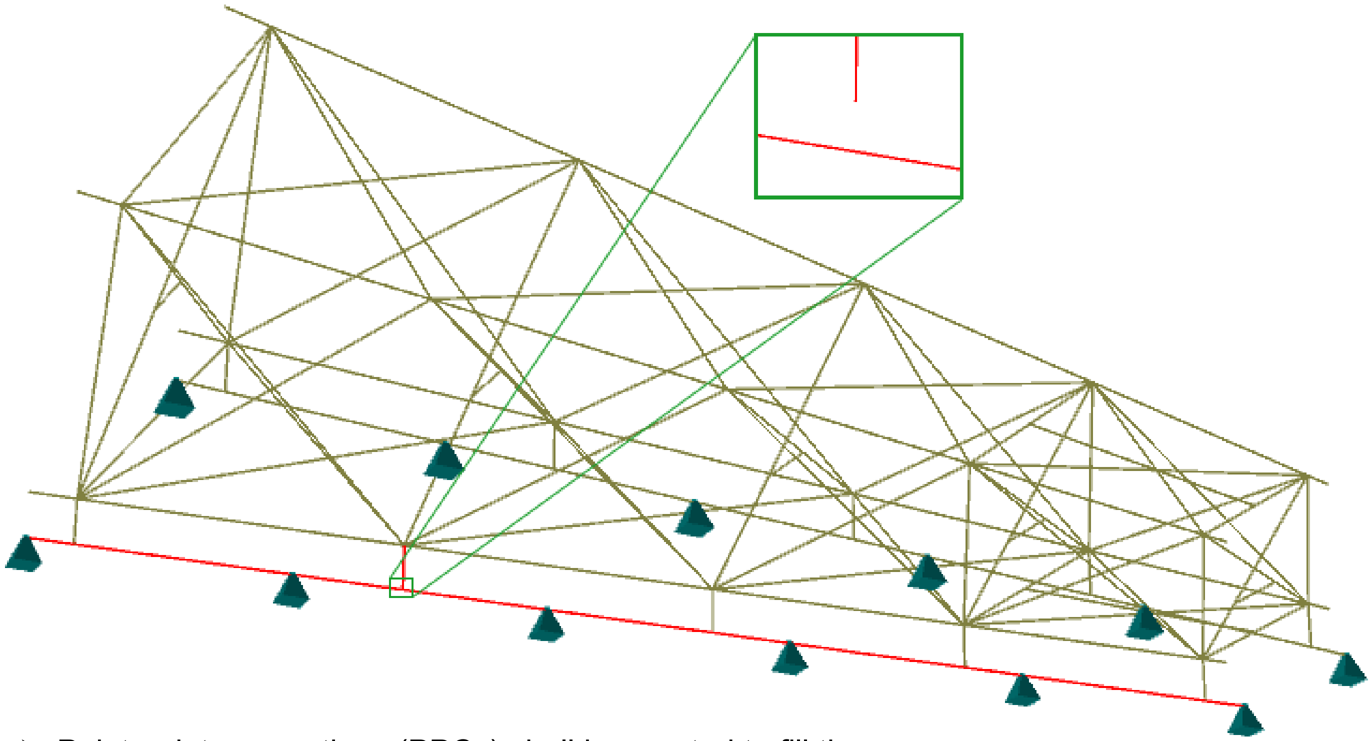
➢ Add boundary conditions
for the two supporting beams.

- First create a pinned boundary condition (three translations fixed and three rotations free) at one end of a supporting beam.
- Distribute five copies of this support point along the supporting beam by first clicking the two ends to fill the *Translation Vector* field, then divide the X and Y vector components by 5 (the Z component is 0) as shown below, finally, specify 5 copies.
- Do so for both supporting beams.



3 CREATE POINT-POINT CONNECTIONS

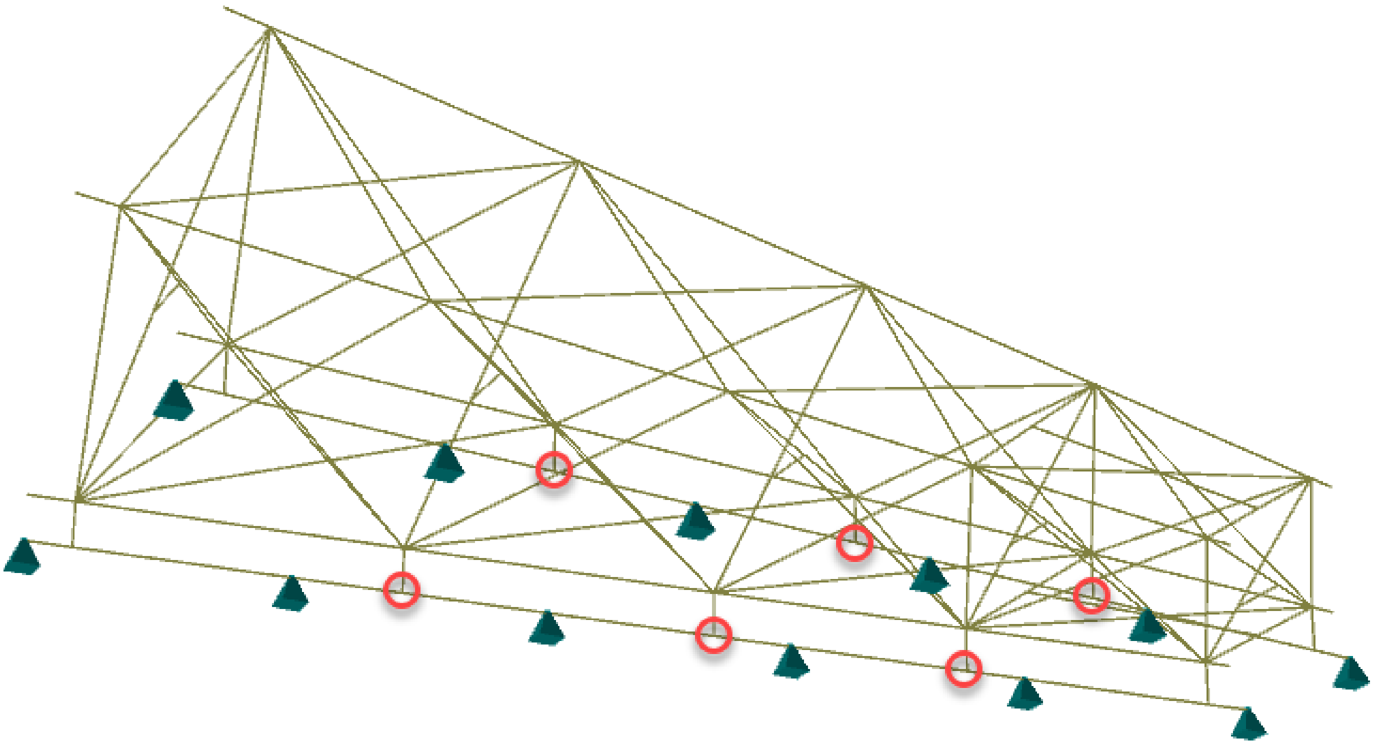
- While it from the display looks like all ten stools are connected to the supporting beams, for six of them there are in fact gaps of 5 cm. This can be seen when switching to *Wireframe* and zooming in as shown below.



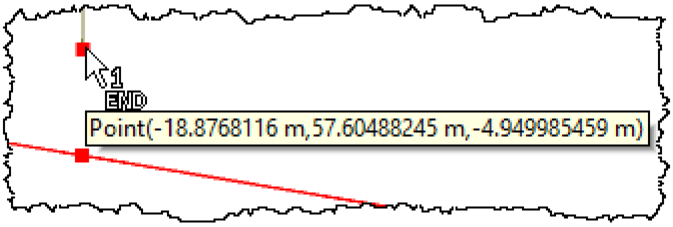
- Point-point connections (PPCs) shall be created to fill these gaps.

- Do so by following the four-step procedure for all six stools encircled below :

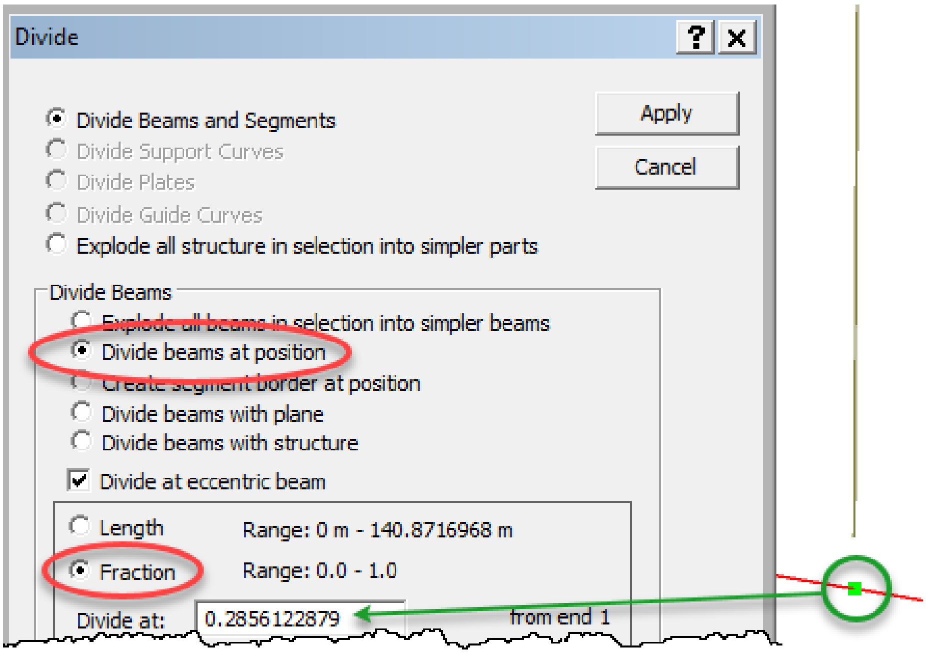
  1. Zoom in.



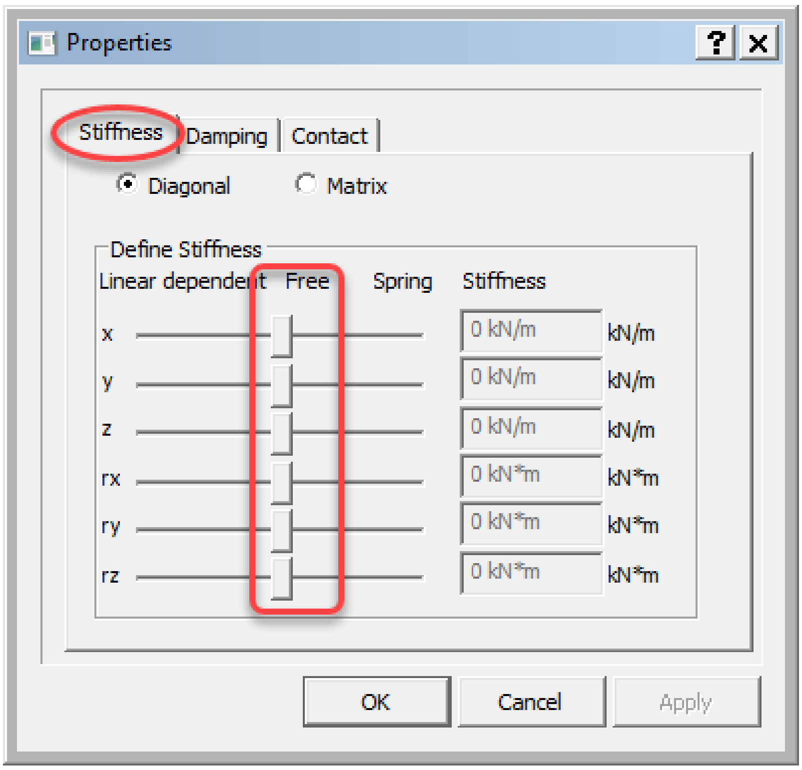
2. Select the supporting beam and use *Guiding Geometry* | *Points* | *Closest to Point on Selection* ( ) to click at the lower end of the stool, thereby creating a guide point on the supporting beam just beneath the stool.



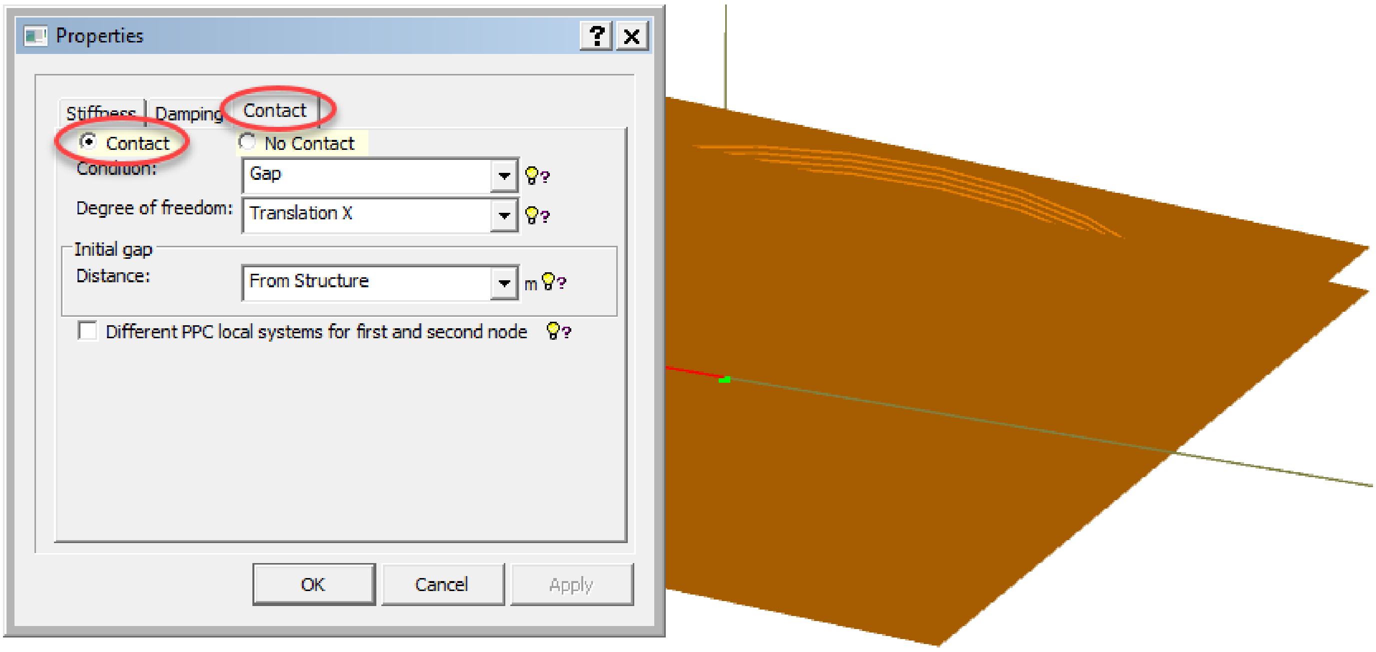
3. Divide the supporting beam using the point created above.
   - The pair of points with initial gaps have now been established.



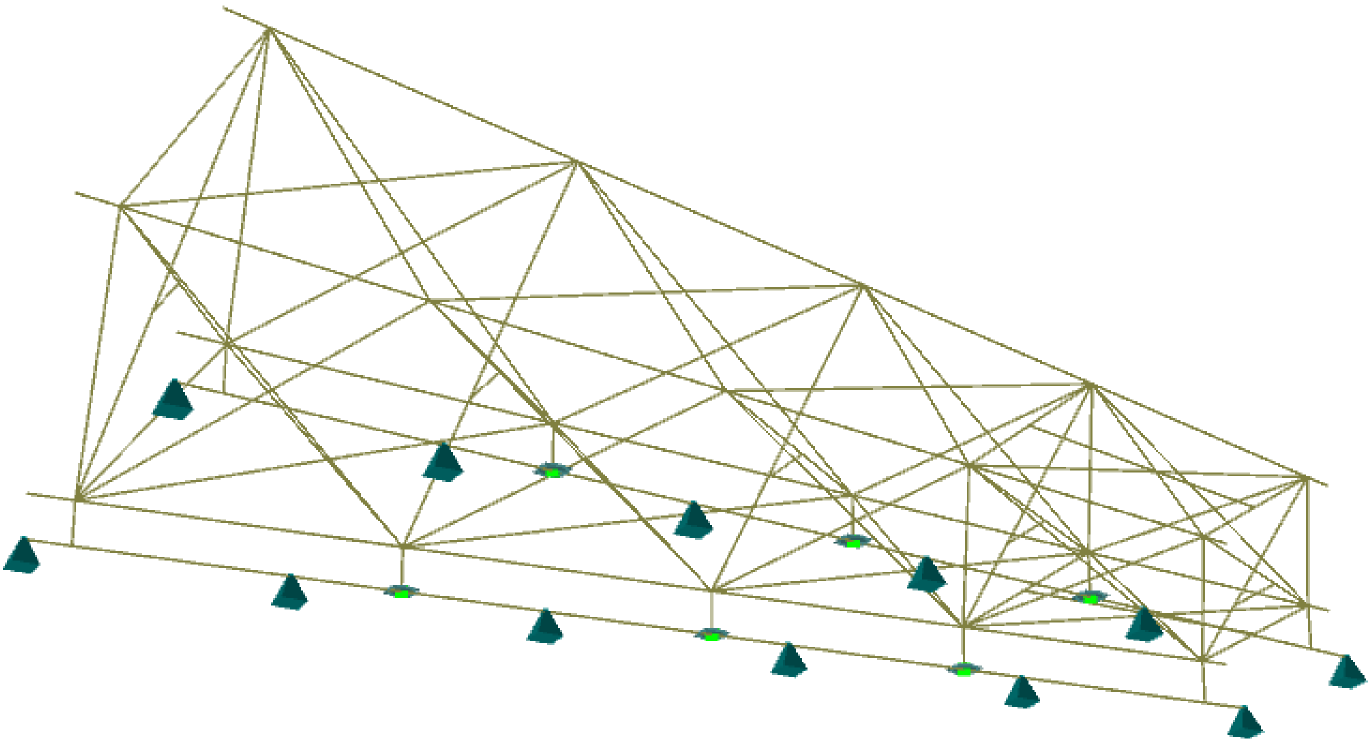
4. Create the PPC as follows:
   - Use *Structure* | *Connection* | *Point-Point Connection* ( ) and click first the lower end of the stool (this ensures that the PPC inherits the x-axis from the stool) and thereafter the point at the supporting beam.
   - In the dialog opening up after clicking the second point, set all six degrees of freedom to *Free*.
   - Do not *OK* yet, the procedure continues next page.



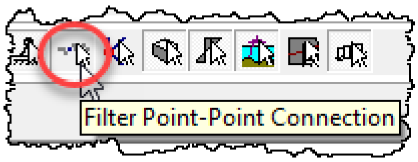
▪ In the *Contact* tab switch from *No Contact* to *Contact*.



➢ Having followed the above procedure for all six stools the model should look like this:



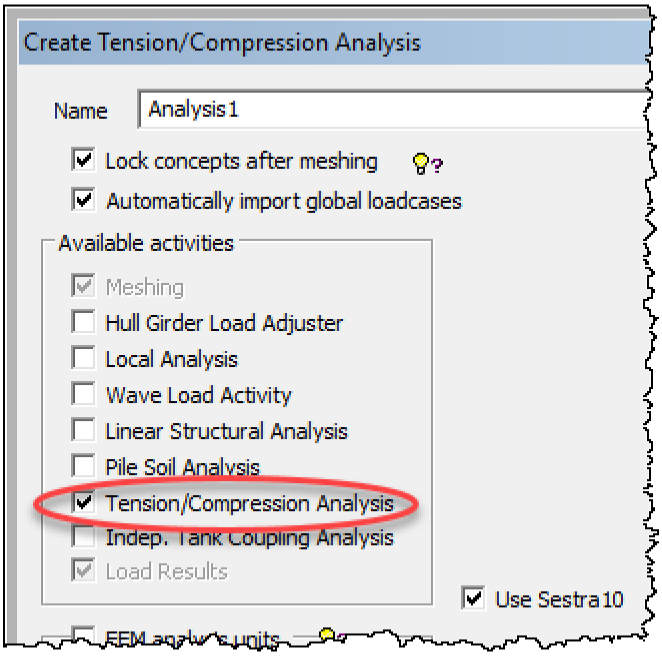
➢ The size of the PPC symbols may be adjusted by right-clicking the *Filter Point-Point Connection* button.



➢ Note that the PPCs created above only restrict the pair of points from displacing beyond contact (overlapping). No restriction on how far apart they can displace has been defined. And neither there is any restriction on the horizontal (X and Y) displacements.

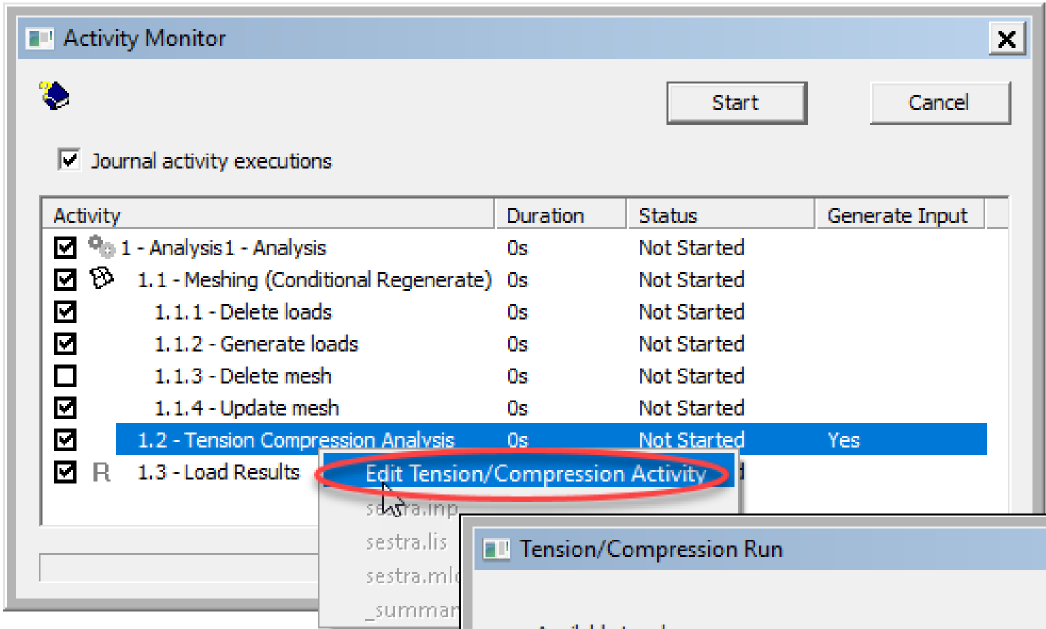
4 DO A GAP/CONTACT ANALYSIS

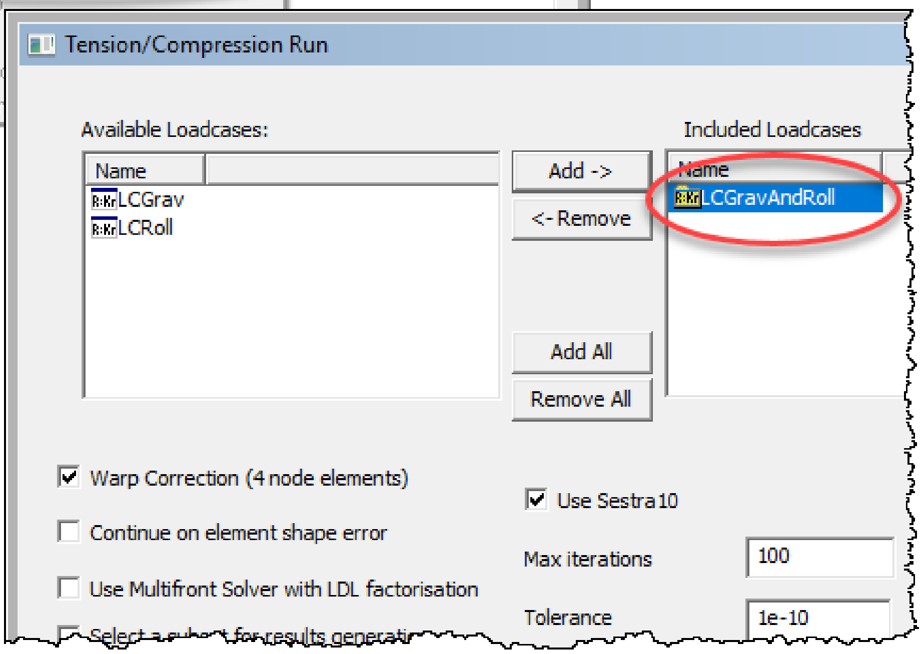
- Click *Mesh & Analysis | Activity Monitor*. In the dialog select *Tension/Compression Analysis*.
  - Note that a gap/contact analysis is in the Sestra analysis program handled in the same way as a tension/compression analysis.



- Click *OK* to close the dialog and open the *Activity Monitor* dialog.

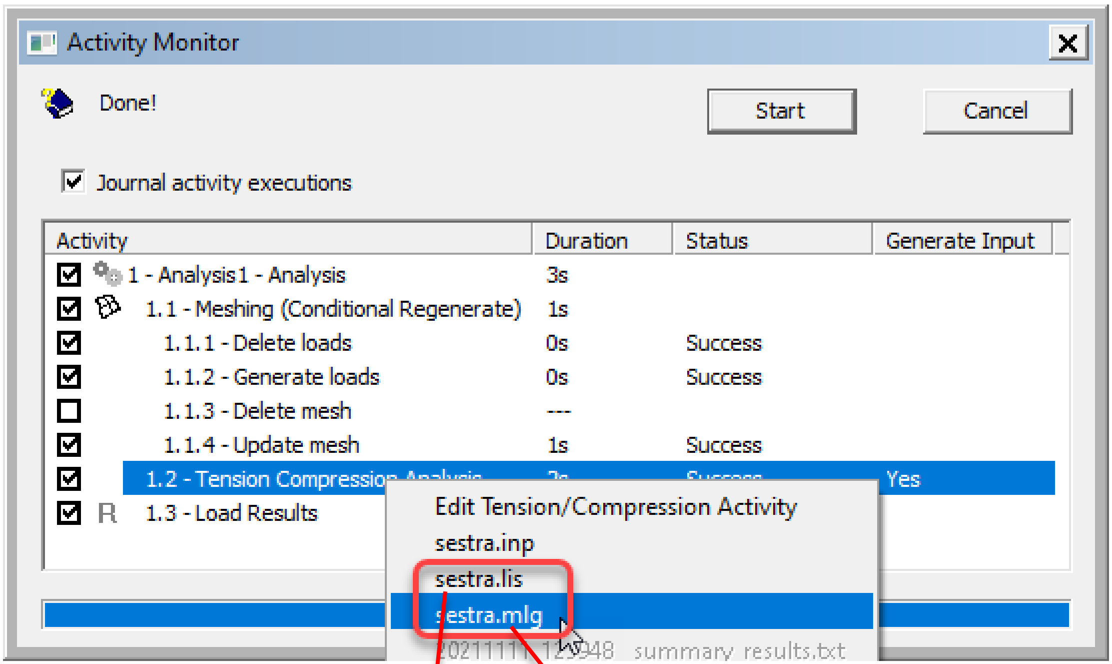
- Edit the *Tension Compression Analysis* and in the *Tension/Compression Run* dialog transfer the load combination LCGravAndRoll from the *Available Loadcases* field to the *Included Loadcases* field.





- Click *Start* to run the analysis.

➢ Open the sestra.lis and sestra.mlg files to investigate and verify the analysis. From the former file, verify that the sum of loads and reaction forces are approximately zero, and in the latter file, see the progress of the iteration of the non-linear analysis.

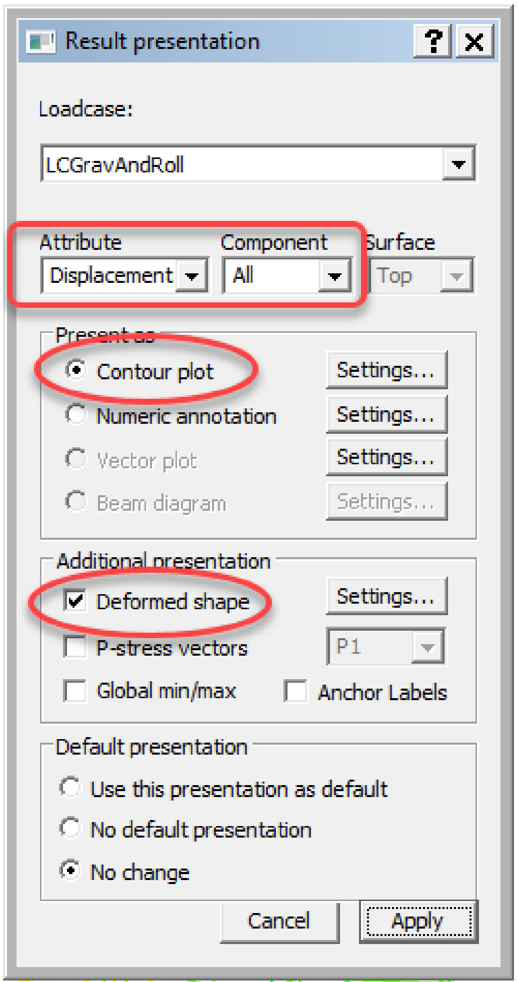



```
Sum of load sum and reaction force sum for all result cases (internal number):
   result case;          tx;            ty;            tz;            rx;            ry;            rz;      magnitude
             1;  7.989001e-09; -7.308378e-11;  1.185981e-09; -1.065433e-06;  4.195608e-07; -4.852191e-07;  1.243657e-06
```

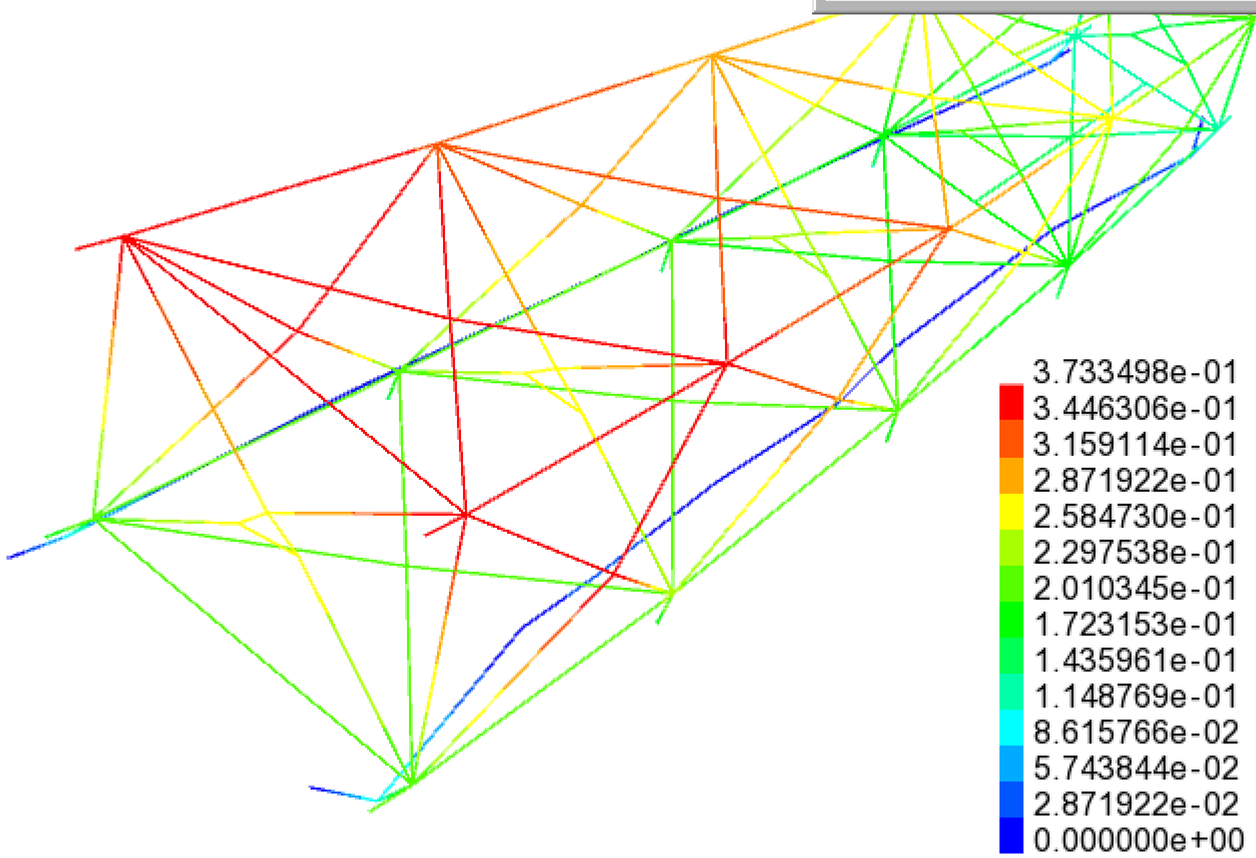
```
Interior-point algorithm convergence history, result case 1
           Max        ;     Primary     ;      Dual       ;   Duality gap
      3.641193e+00    ;   8.267943e-01  ;   3.641193e+00  ;   2.449490e+00
      2.144031e-01    ;   2.405804e-05  ;   5.469504e-04  ;   2.144031e-01
      6.050647e-02    ;   1.658083e-09  ;   3.082953e-08  ;   6.050647e-02
      5.737576e-03    ;   8.920090e-14  ;   1.603166e-12  ;   5.737576e-03
      5.149460e-05    ;   3.489983e-16  ;   1.001277e-13  ;   5.149460e-05
      2.677430e-09    ;   2.946109e-17  ;   1.021078e-13  ;   2.677430e-09
      1.341029e-13    ;   1.700937e-17  ;   1.010127e-13  ;   1.341029e-13
     ** Minimum residual 1.341029e-13 obtained after 7 iterations **
```

5 VIEW THE RESULTS

- View the results in GeniE by switching to *Results - All* display configuration, opening the *Result presentation* dialog and selecting for instance *Displacements All*, *Contour plot* and *Deformed shape*.

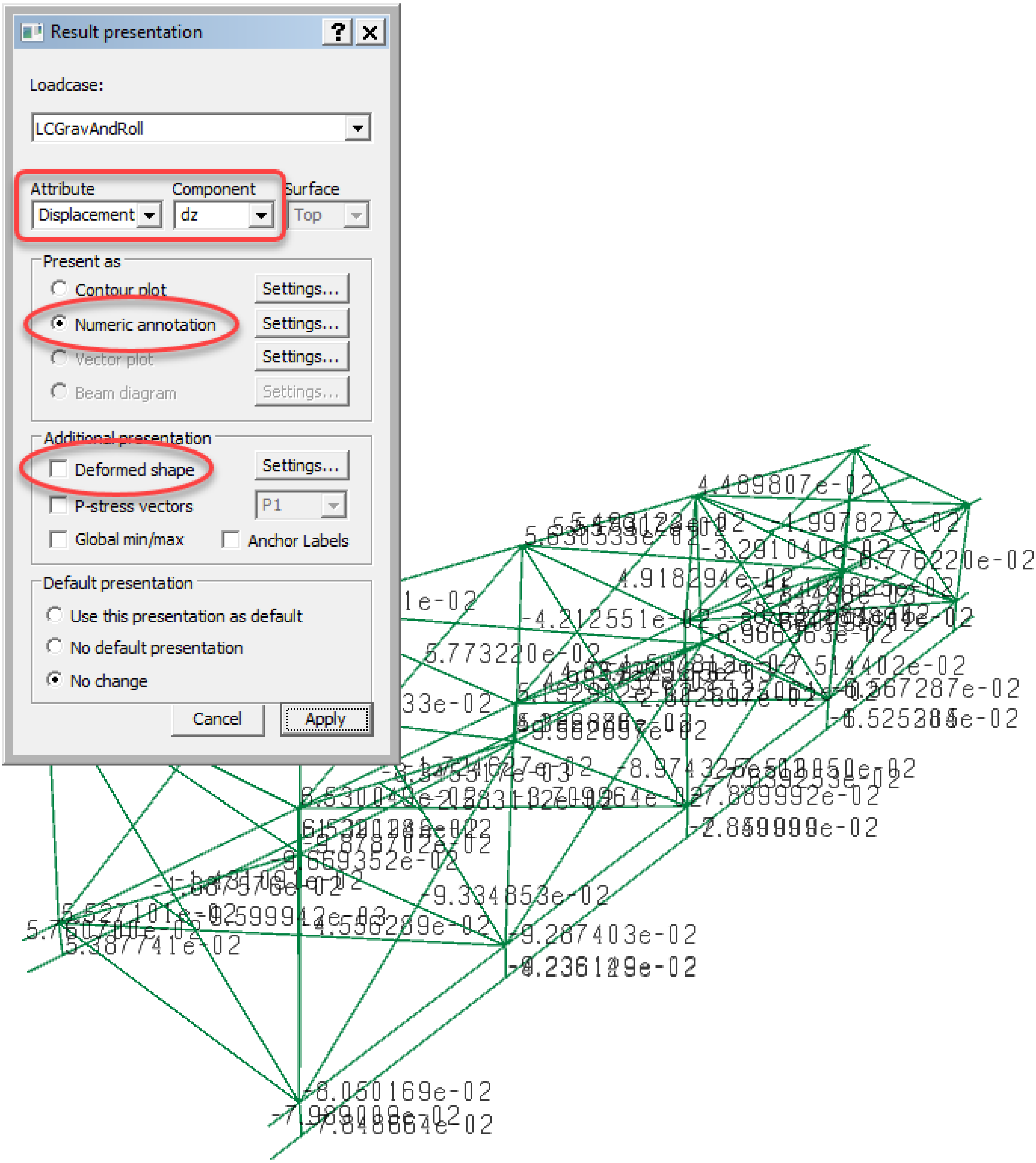


- Note that the contact conditions (PPCs) are only concerned with the vertical displacement component. There are no conditions or restrictions on the horizontal components (X and Y).
    - For this reason, the deformation displayed below with a factor exaggerating the deformation shows that the bottom of the stools and the supporting beams are unconnected whether there is contact or not between them.

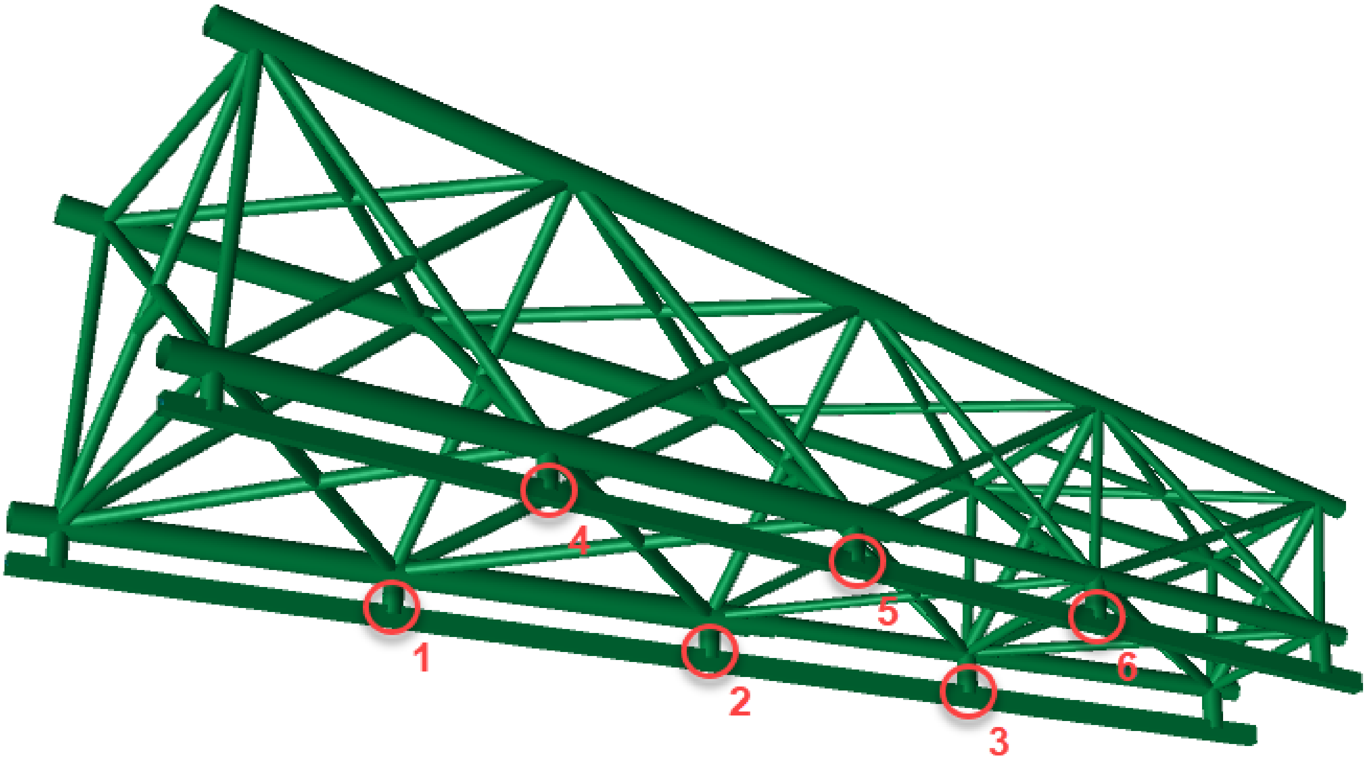


➢ A further investigation of the gap/contact results between the stools and supporting beams can be done by switching to the *Results - with Mesh* display configuration and in the *Result presentation* dialog select *Component dz* (vertical displacement component), *Numeric annotation* and uncheck *Deformed shape*.

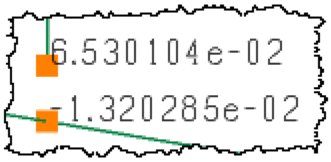
➢ Then zoom in on the gaps as shown next page.



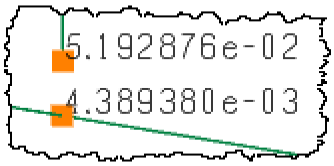
- The vertical displacement components *dz* for the six gaps identified below are presented further down.



1. The difference between *dz* for the upper (stool bottom) and lower (supporting beam) nodes is 0.0653 − (−0.0132) = 0.0785. Adding the original distance between the nodes of 0.05 gives the total distance of 0.1285.



2. Difference between *dz* for upper and lower nodes is 0.0519 − 0.0044 = 0.0475.
   Adding 0.05 gives total distance of 0.0975, i.e. no contact.

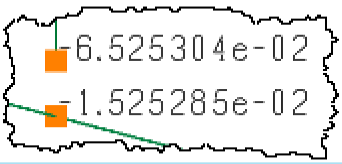


3. Difference between *dz* for upper and lower nodes is 0.0449 − (−0.0085) = 0.0534.
   Adding 0.05 gives total distance of 0.1034, i.e. no contact.

4. Difference between *dz* for upper and lower nodes is −0.0924 − (−0.0424) = −0.0500.
   Adding 0.05 gives total distance of 0.0, i.e. contact!

5. Difference between *dz* for upper and lower nodes is −0.0785 − (−0.0285) = −0.0500.
   Adding 0.05 gives total distance of 0.0, i.e. contact!

6. Difference between *dz* for upper and lower nodes is −0.0653 − (−0.0153) = −0.0500.
   Adding 0.05 gives total distance of 0.0, i.e. contact!



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.

DNV | digital@dnv.com | dnv.com/digital