

Everything You Always Wanted to Know About Robot Structural Analysis, but Were Afraid to Ask

Dominik Malec

Senior Technical Support Specialist in Autodesk





About the speaker

Dominik Malec

Structural Engineer with 4 years of experience as Structural Engineer Assistant – designing of steel structures, bridges, residential and office buildings, 3 years of experience as Civil Engineer Estimator– preparation of tenders and schedules for highway bridge and road contracts.

And over 15 years of experience as Technical Support Specialist and Trainer (Robobat, Graitec). - Robot Structural Analysis, Autodesk Advance Steel and other software.

Trainings and implementations for engineers in China, Denmark, Estonia, France, India, Indonesia, Ireland, Italy, Latvia, Lithuania, Norway, Romania, Russia, Spain, United Kingdom and Poland.

Currently working as Senior Technical Support Specialist in Autodesk.

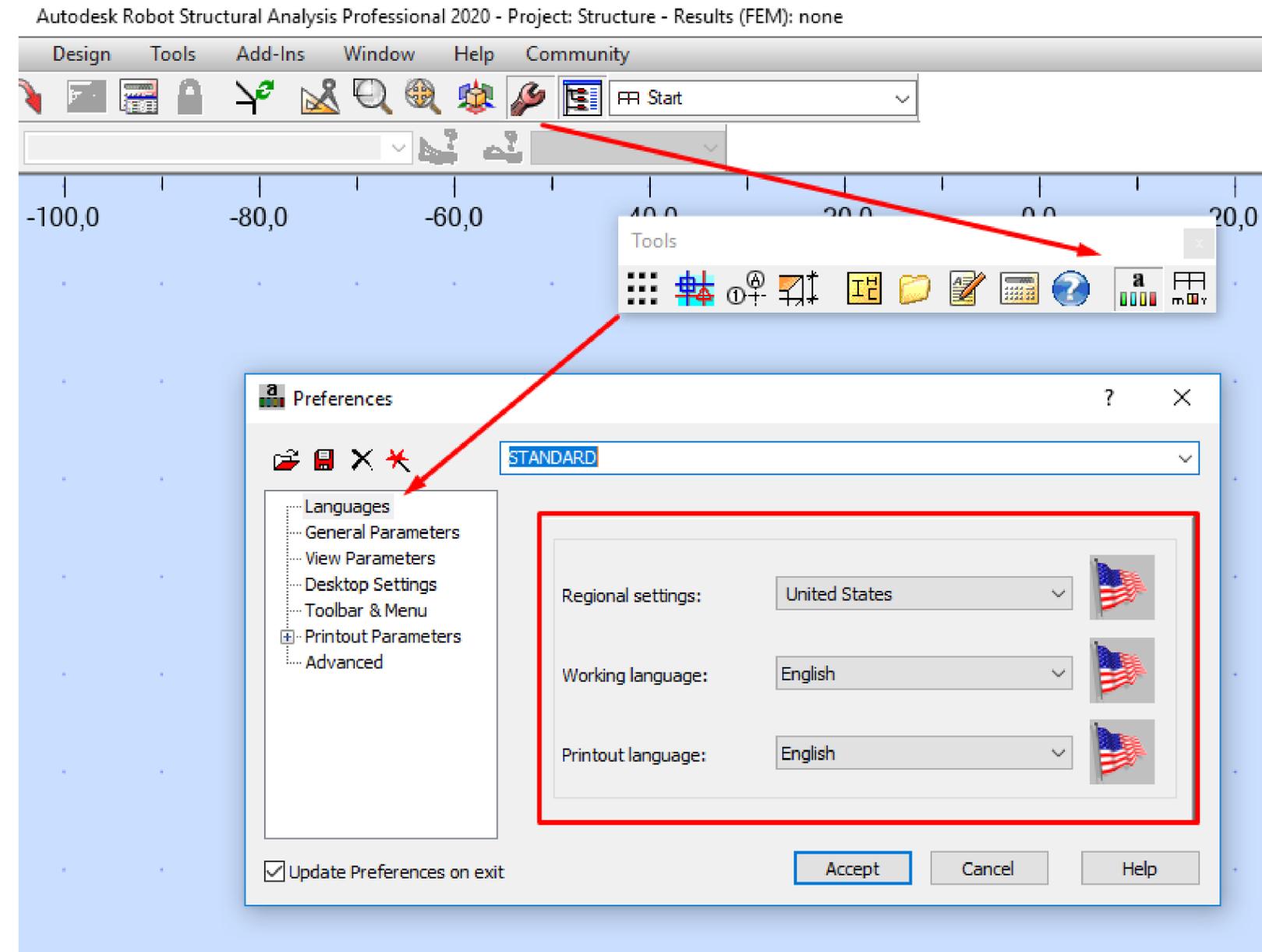
Does Robot Work?



Robot is just a tool!

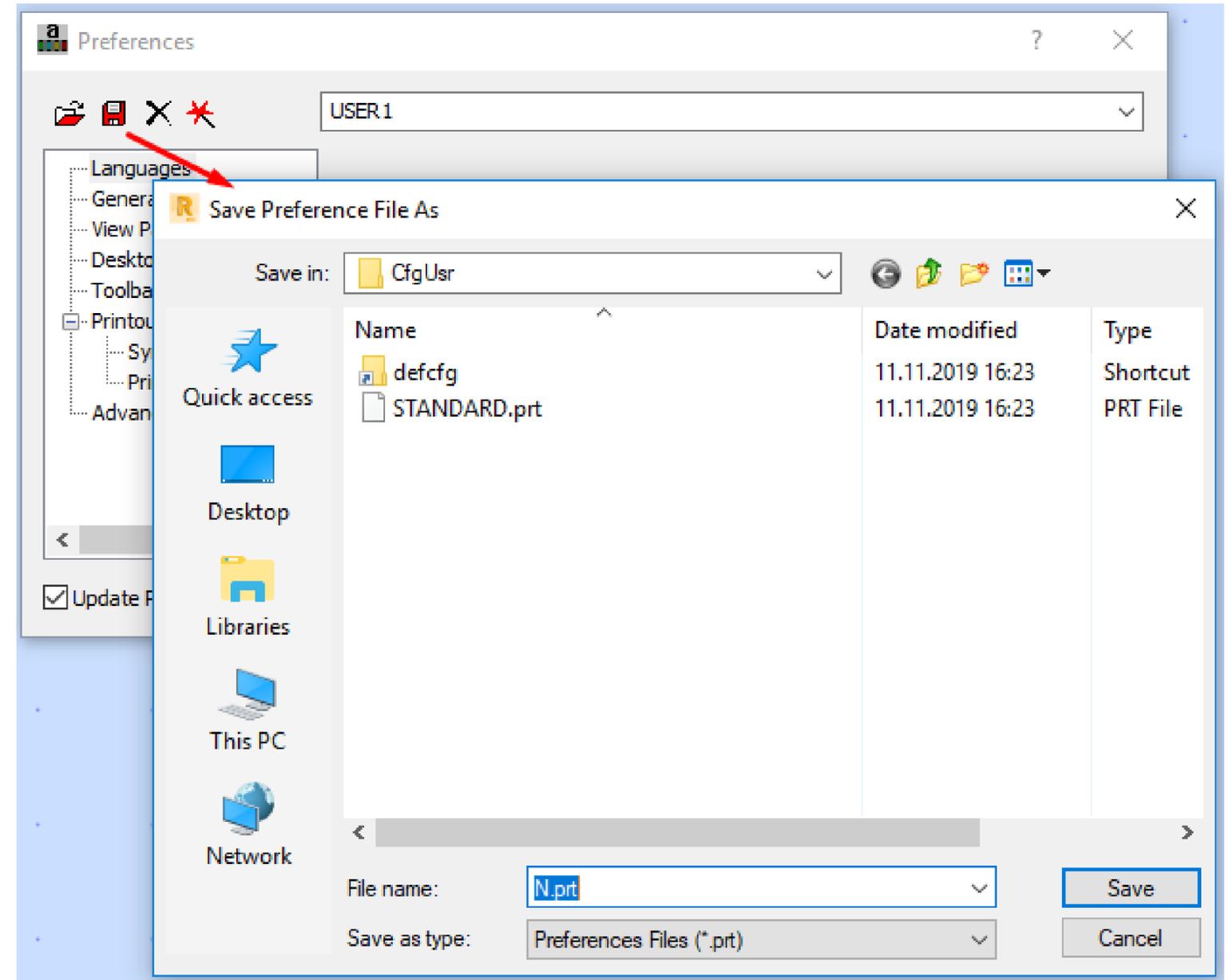
In order to use it effectively, remember to prepare it for work. It should be configured in such a way that both the regional settings, the interface language and the settings of the print-out language match our needs.

Please note that the regional settings are related to the operating system settings.



Save your configuration settings!

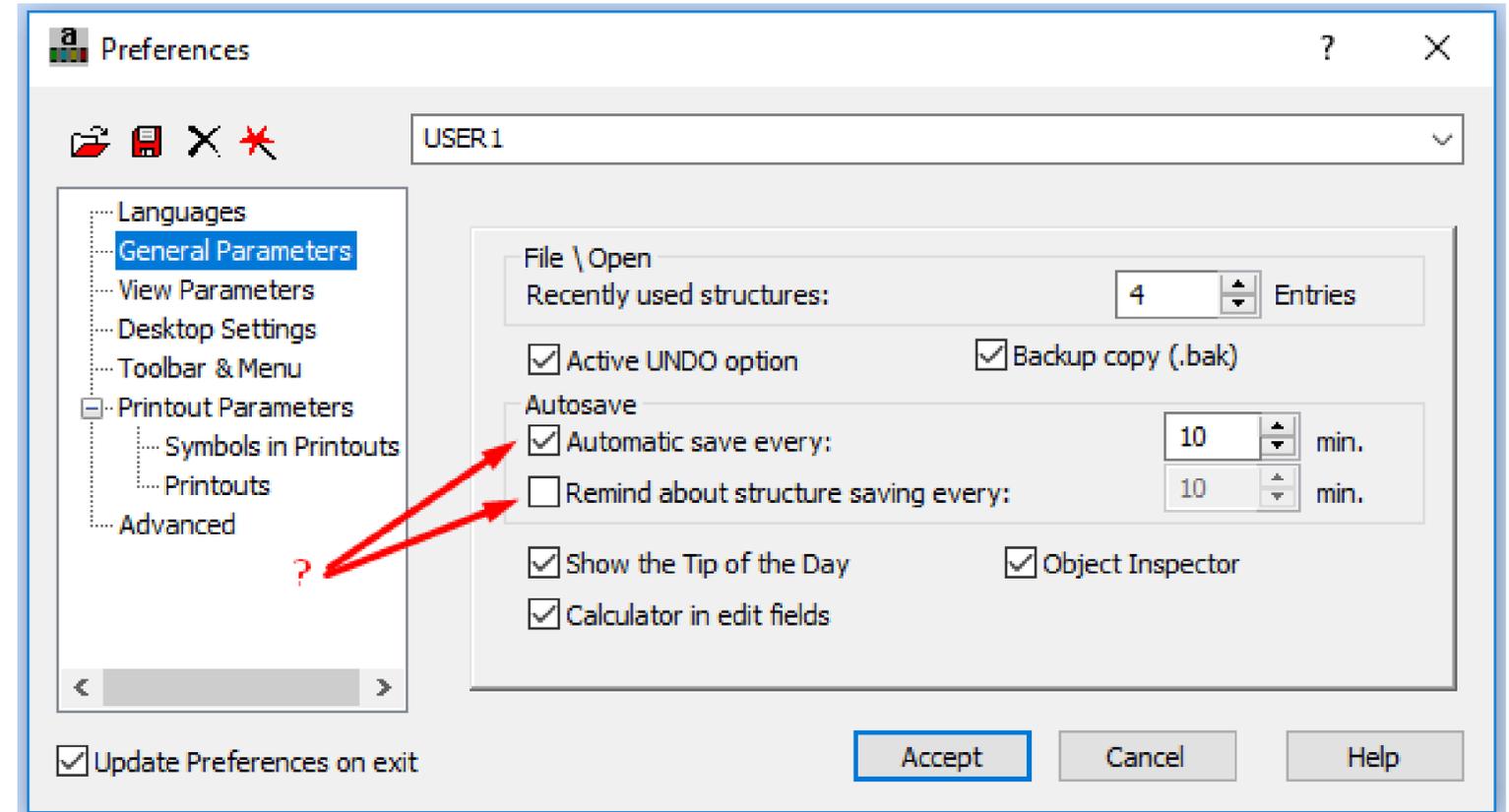
We don't have to do the same thing every time. Any Preferences settings can be saved in an external file and set as default. What matters is where we save these files!
The same rule applies to Job Preferences!



Don't lose the results of your work!

We do not like to lose the effects of our work! It seems even more annoying when we set the "autosave" and it doesn't seem to work!

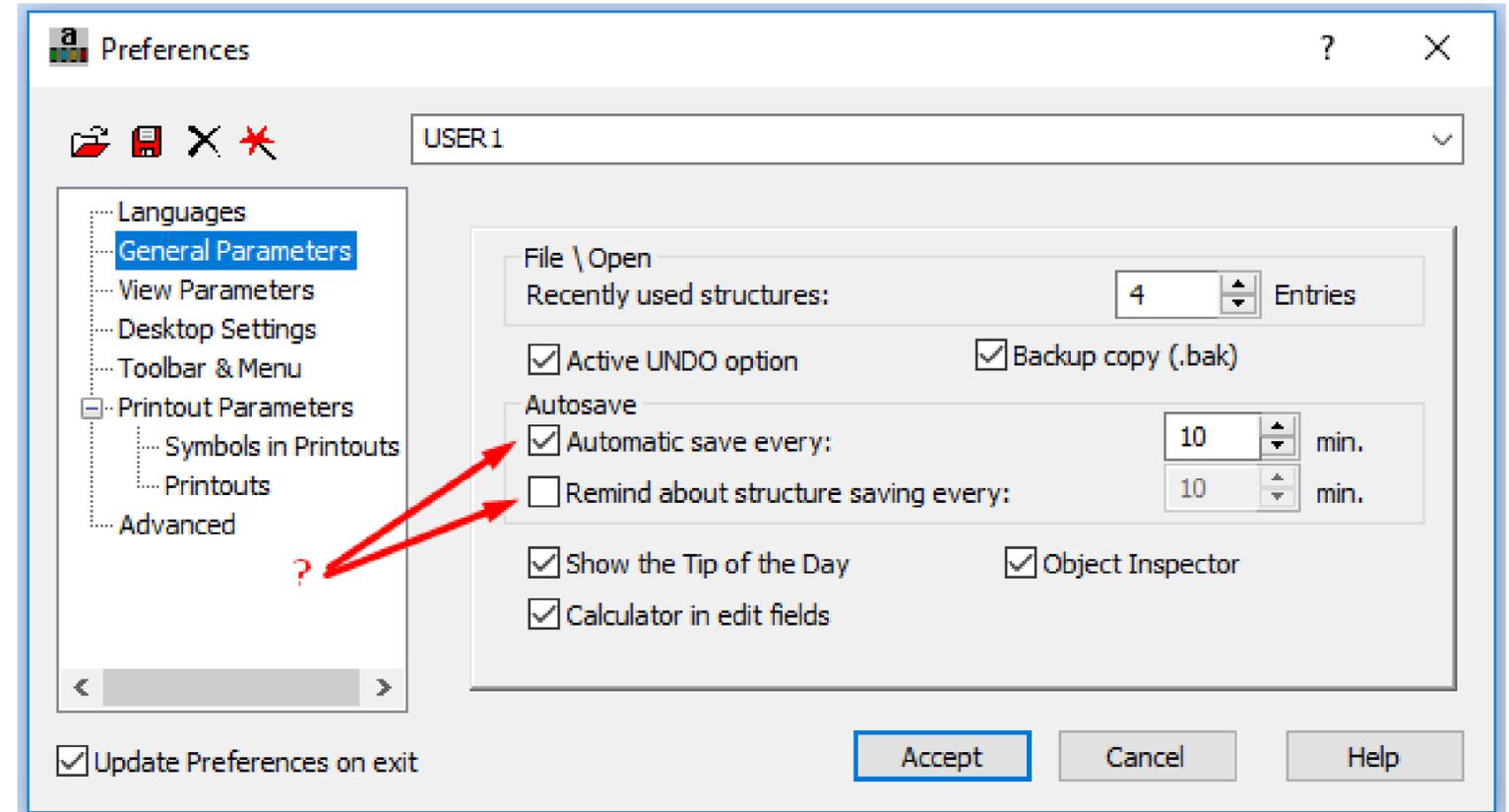
Let's use "reminder" and save separate files from each stage of the work.



Don't lose the results of your work!

We do not like to lose the effects of our work! It seems even more annoying when we set the "autosave" and it doesn't seem to work!

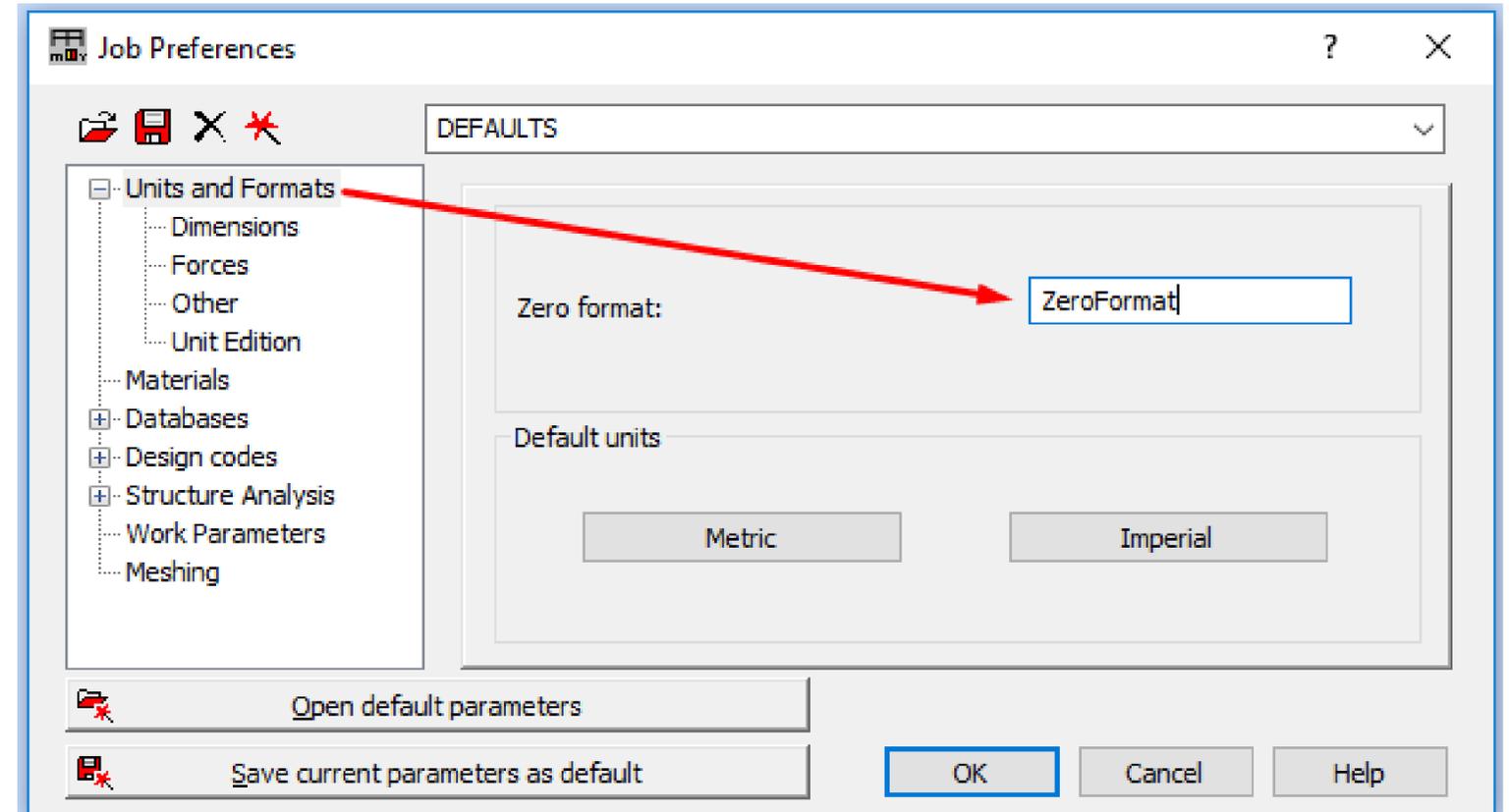
Let's use "reminder" and save separate files from each stage of the work.



Zero isn't always zero!

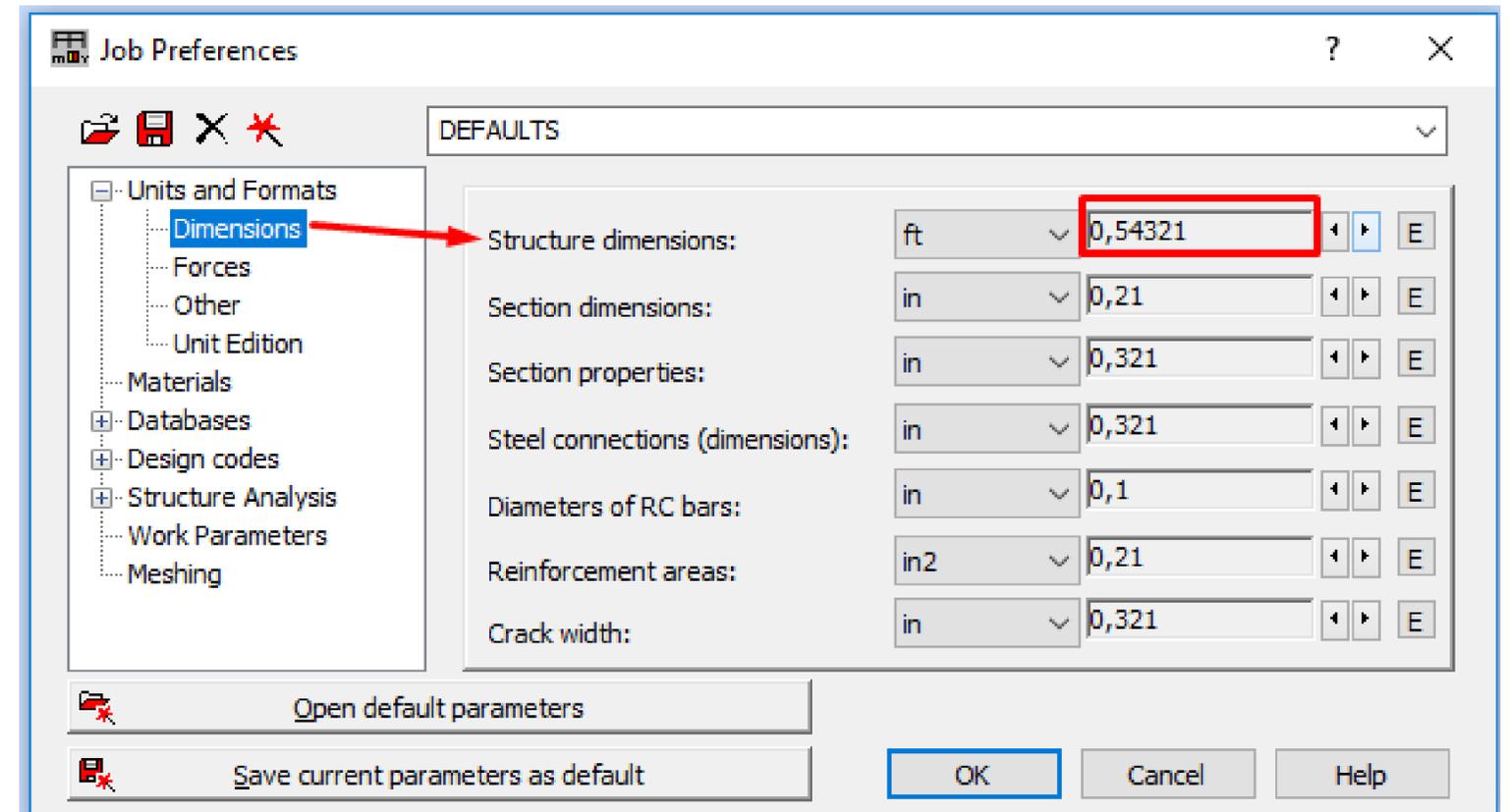
Sometimes zero has a value. Especially numerically.

When we need to distinguish between "real zero" and "almost zero" it is worth to use the possibility of using your own zero format.



Dimensions are not equal to dimensions!

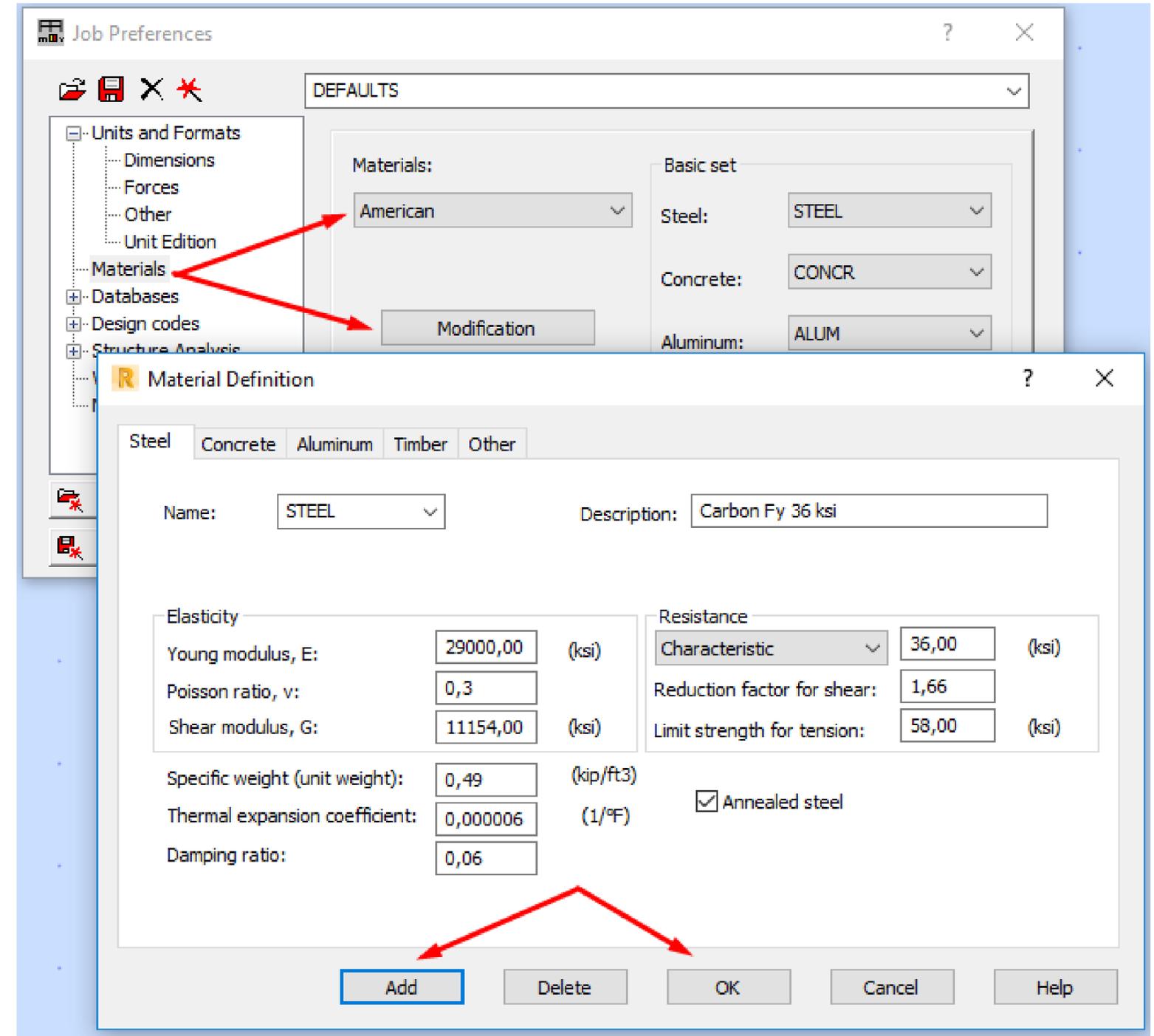
The position of nodes in the model is crucial for further calculations. Actual node positions are presented in numerical form, among others, in tables and properties. Not displaying them with the correct accuracy can lead to problems.



Choose the right material!

The choice of construction material has an influence on the load-bearing capacity of the structure. We use specific design standards to check the load capacity.

The selected or newly developed material must contain the complete information required by the selected standard.



Verify according to the standard!

The choice of the standard for the design of the structure requires the use of appropriate loads. At the same time, the individual loads do not work independently. Their interaction and coefficients are defined by the standard setter.

If the code regulations are modified, their copies should be used!

The image shows two overlapping windows from Autodesk Robot Structural Analysis Professional 2020. The top window is 'Job Preferences' with the 'DEFAULTS' tab selected. It shows settings for 'Code combinations' (LRFD ASCE 7-16), 'Snow/wind loads' (ASCE 7-05), and 'Seismic loads' (ASCE 7-16). A red arrow points from the 'Code combinations' dropdown to the 'Editor of code combination regulations' window below. The bottom window is the 'Editor of code combination regulations' for 'LRFD ASCE 7-16' (Version: 33). It contains two tables: a load coefficient table and a combination table.

	Nature	Subnature	γ_{max}	γ_{min}	γ_s	γ_a	$\Psi_{0,1}$	$\Psi_{0,2}$	$\Psi_{0,3}$	$\Psi_{0,n}$	Ψ_1	$\Psi_{2,1}$	$\Psi_{2,n}$	Ψ_K	ξ_i	ξ_z
1	Dead		1.2	0.9	1	1.4					1.6	1	1			
2	Live		1		1					1						
3	Wind		1		1		0.5			1	1			1		
4	Snow		1		1	1.6				1	1.6	0.5	0.2			
5	Snow	Roof live	1		1	1.6				1	1.6	0.5	0.2			
6	Snow	Rain	1		1	1.6				1	1.6	0.5	0.2			
7	Seismic					1										
8																

	Combination type	User-defined type	Loads				
			Dead	Live	Accidental	Seism	
1	ULS	USR	1. 1.4D	(5) $\sum_{i \geq 1} G_i \cdot \gamma_a^{(i)}$	(0) _____	(0) _____	(0) _____
2	ULS	USR	2. & 4. 1.2D+1.6W+0.5(Lr/S/R) & 1.2D+1.0W+L+0.5(Lr/S/R)	(2) $\sum_{i \geq 1} G_i \cdot \gamma_{max}^{(i)}$	(20) $Q_i \cdot \Psi_1 + \sum_{j \geq 1, i \neq j} Q_j \cdot \Psi_{2,1}$	(0) _____	(0) _____
3	ULS	USR	3. 1.2D+1.6(Lr/S/R)+L+0.5W	(2) $\sum_{i \geq 1} G_i \cdot \gamma_{max}^{(i)}$	(23) $\sum_{i \geq 1} Q_i \cdot \gamma_i \cdot \Psi_{0,i}$	(0) _____	(0) _____

What Is Instability?



Stay calm and stable!

It is no coincidence that the standard Robot calculation type is static analysis. A structure cannot be a mechanism, it cannot "drive away" from the construction site.

The instability message allows us to locate and prevent errors.

The screenshot displays the Autodesk Robot Structural Analysis Professional interface during a non-linear analysis. The main window shows the progress of the analysis with three green bars: Reordering (100%), Solution (approx. 25%), and Resolution stage (100%). A dialog box is open, displaying the following message:

Autodesk Robot Structural Analysis Professional 2019

Instability (of type 1) detected at node 698 in direction RZ.
Do you want to continue?
[Press ESC to ignore all warnings]

The dialog box has three buttons: Yes, No, and Cancel.

Below the dialog box, the Statistics section is visible, showing the following data:

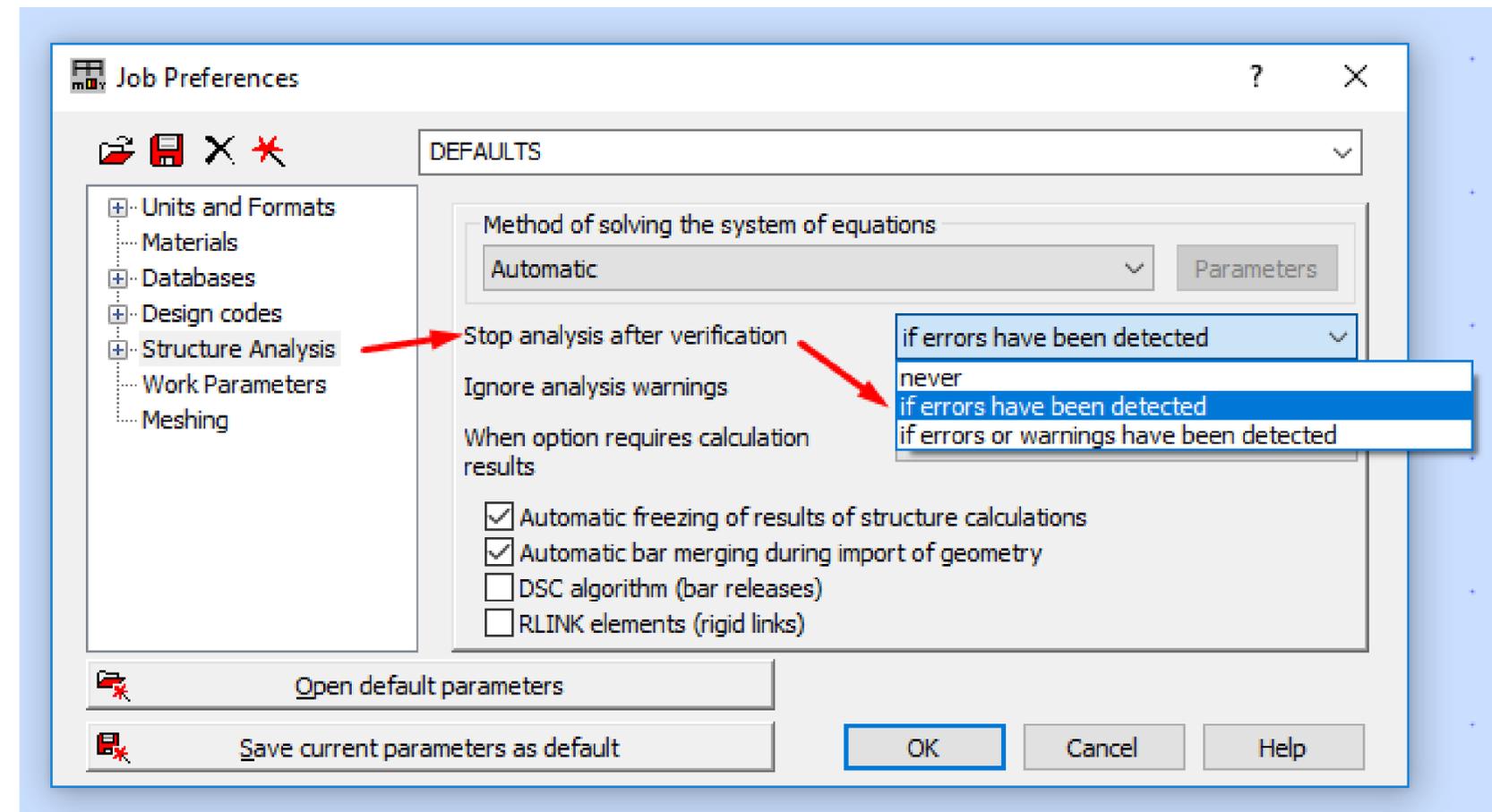
Statistics	Value
Number of Nodes	686
Number of Elements	742
Number of Equations	4113
Frontwidth	
Initial	
Optimized	

Additional statistics shown include Memory (19491.242) and Disk (82551.633). The Case section shows Case 1, Start of Analysis at 18:24:22, and Estim. Run Time. The Calculation Priorities are set to Normal.

Never set "never"!

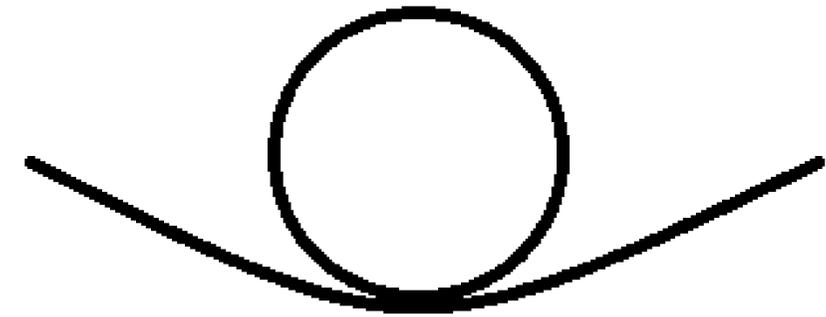
We can't ignore messages about model stability errors! It is even worse to turn them off.

Unless we know what's going on... Then let's check if we really know!



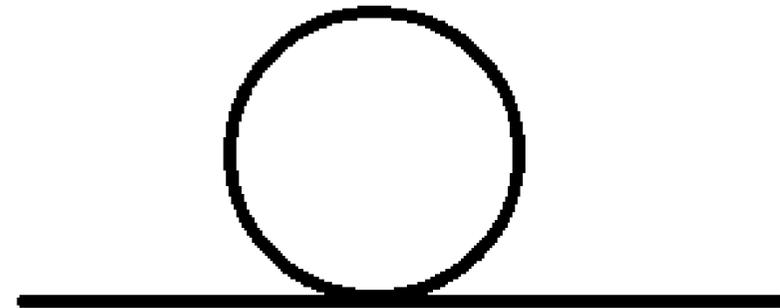
Type 3!

Type 3: 'mathematical' instability signalled when two elements of high size disproportion are detected in the matrix.



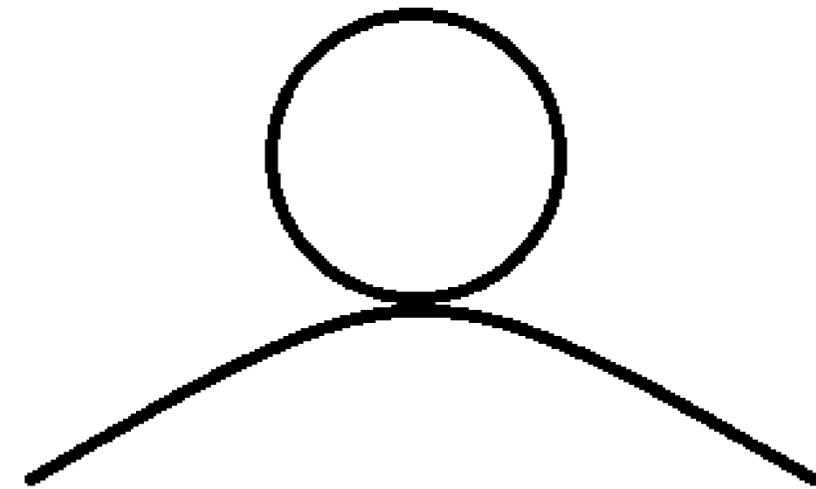
Type 2!

Type 2: ,potential' instability indicated when a zero-value element is detected in an inverted matrix.



Type 1!

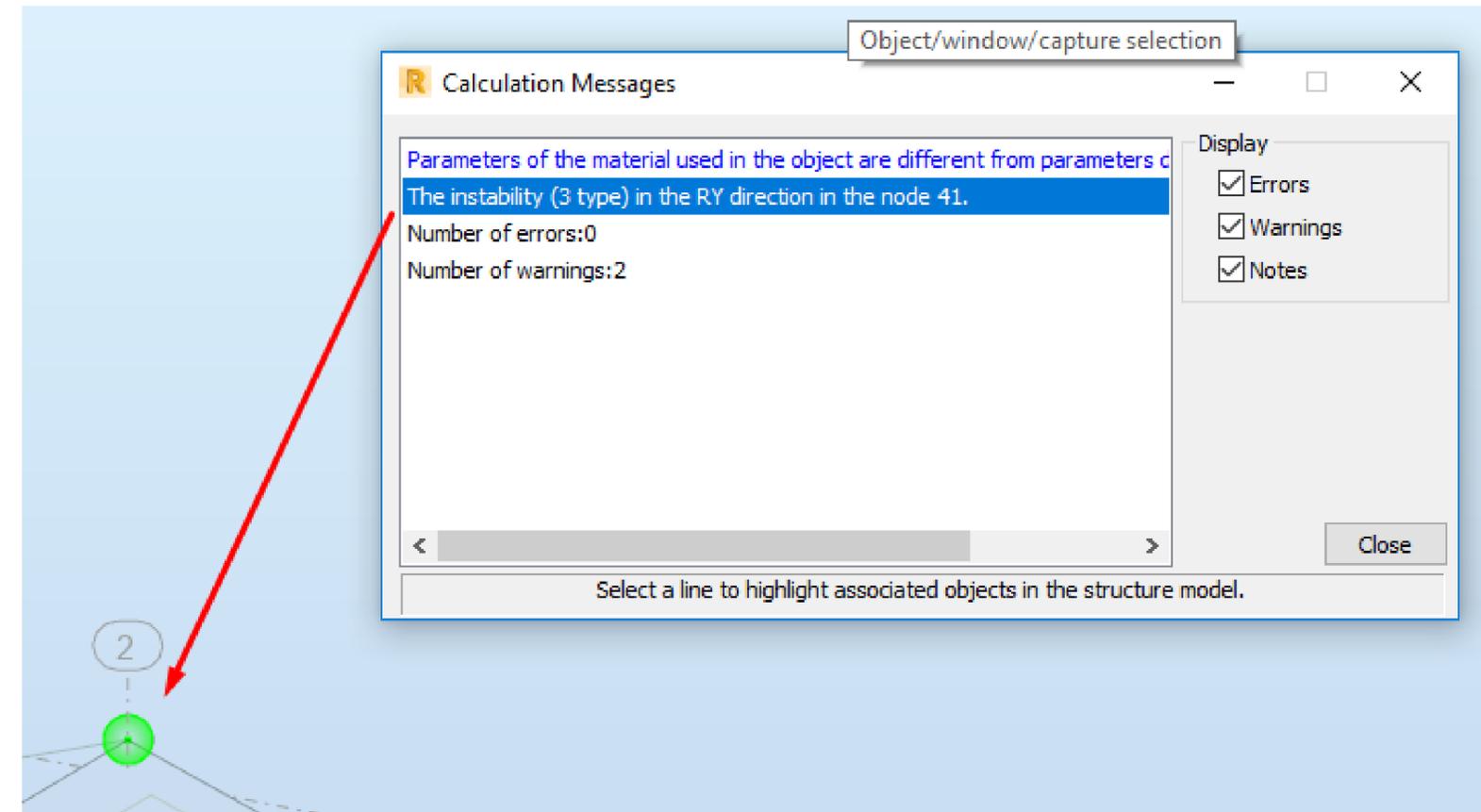
Type 1: 'real' instability indicated when a zero-value element is detected in the matrix.



Find this node (or these nodes)!

We need to find the node where the instability was detected!
Simply click on the message and the node will light up green.

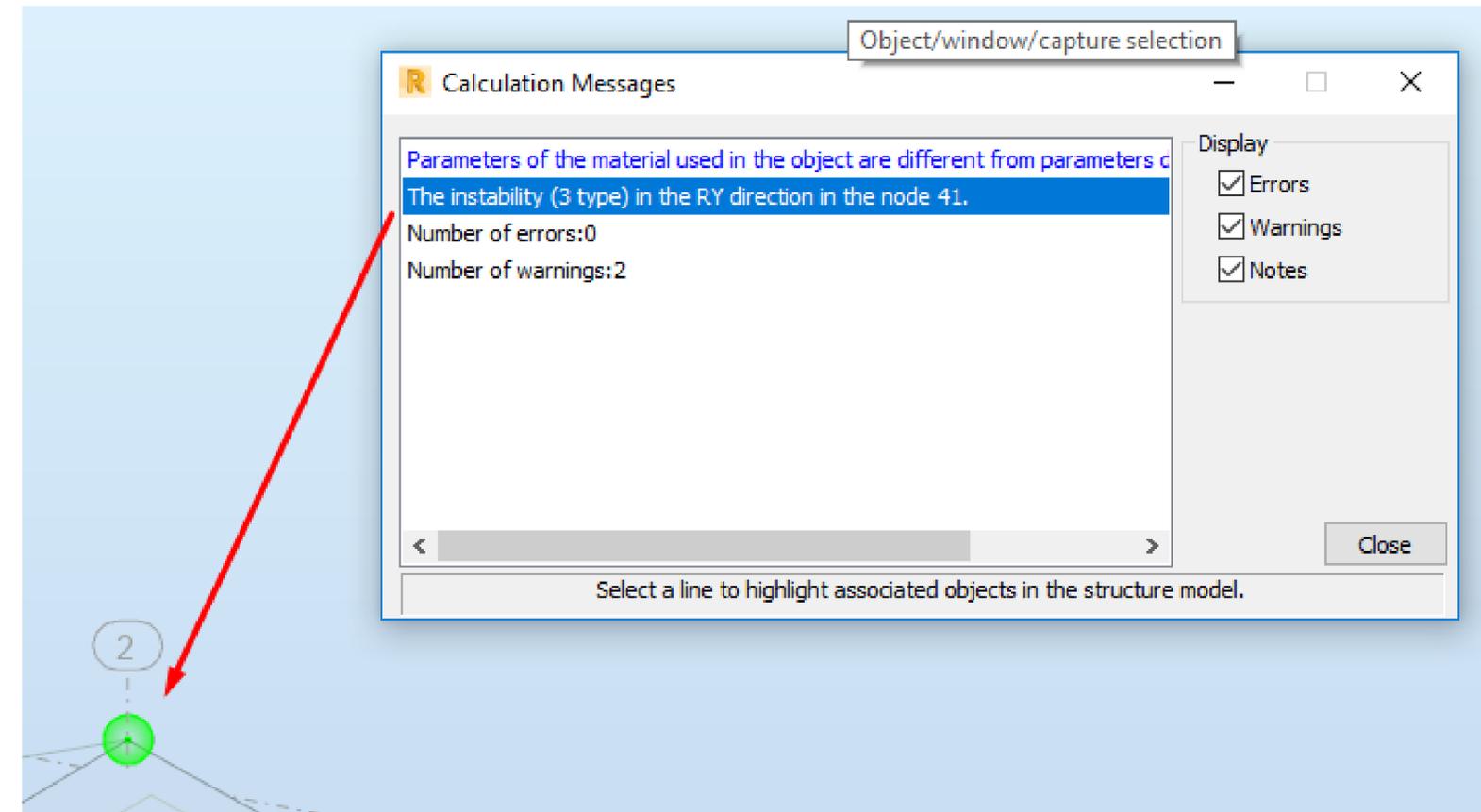
We can also display only selected nodes or finite elements connected to them.



Find this node (or these nodes)!

We need to find the node where the instability was detected!
Simply click on the message and the node will light up green.

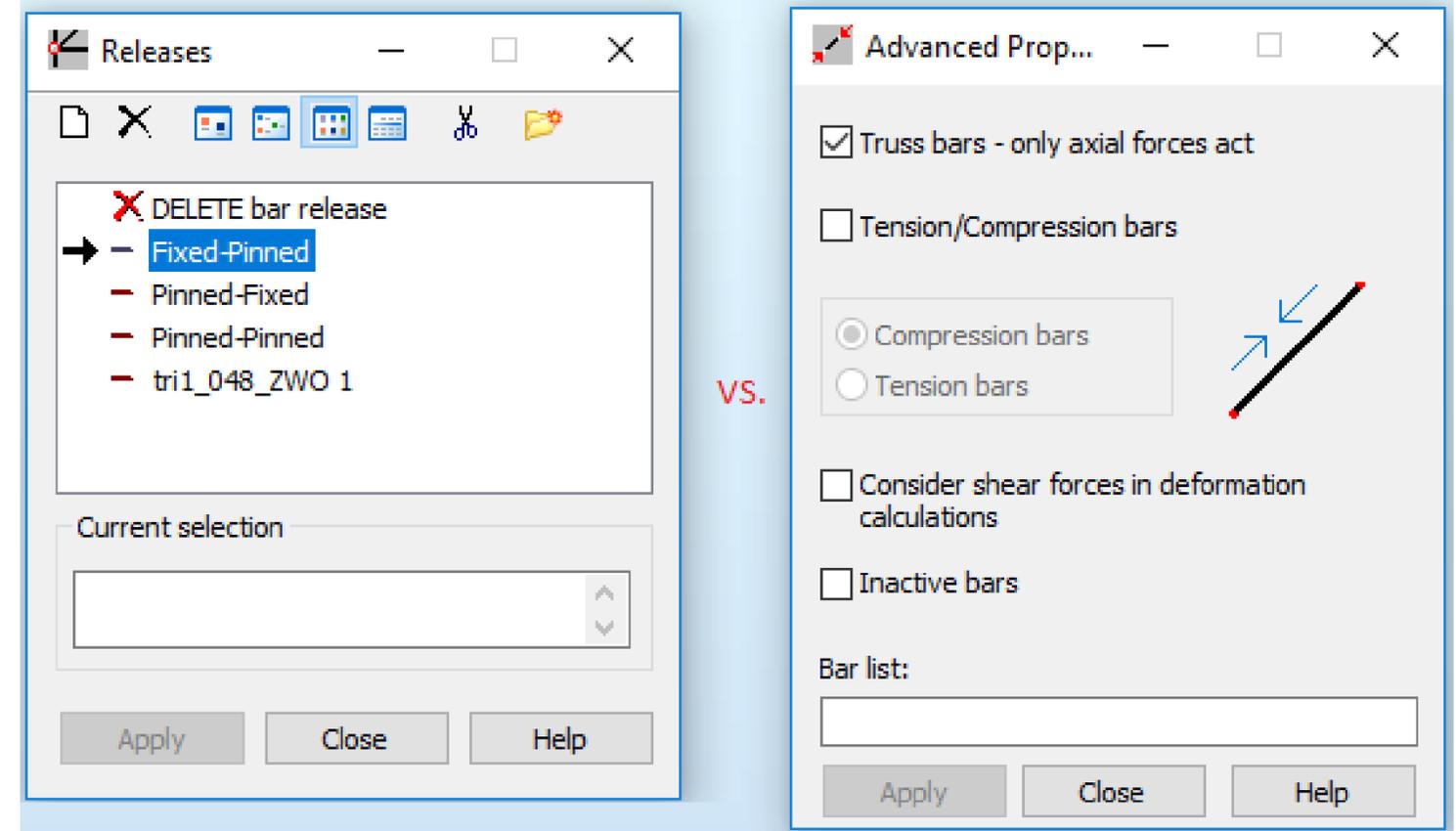
We can also display only selected nodes or finite elements connected to them.



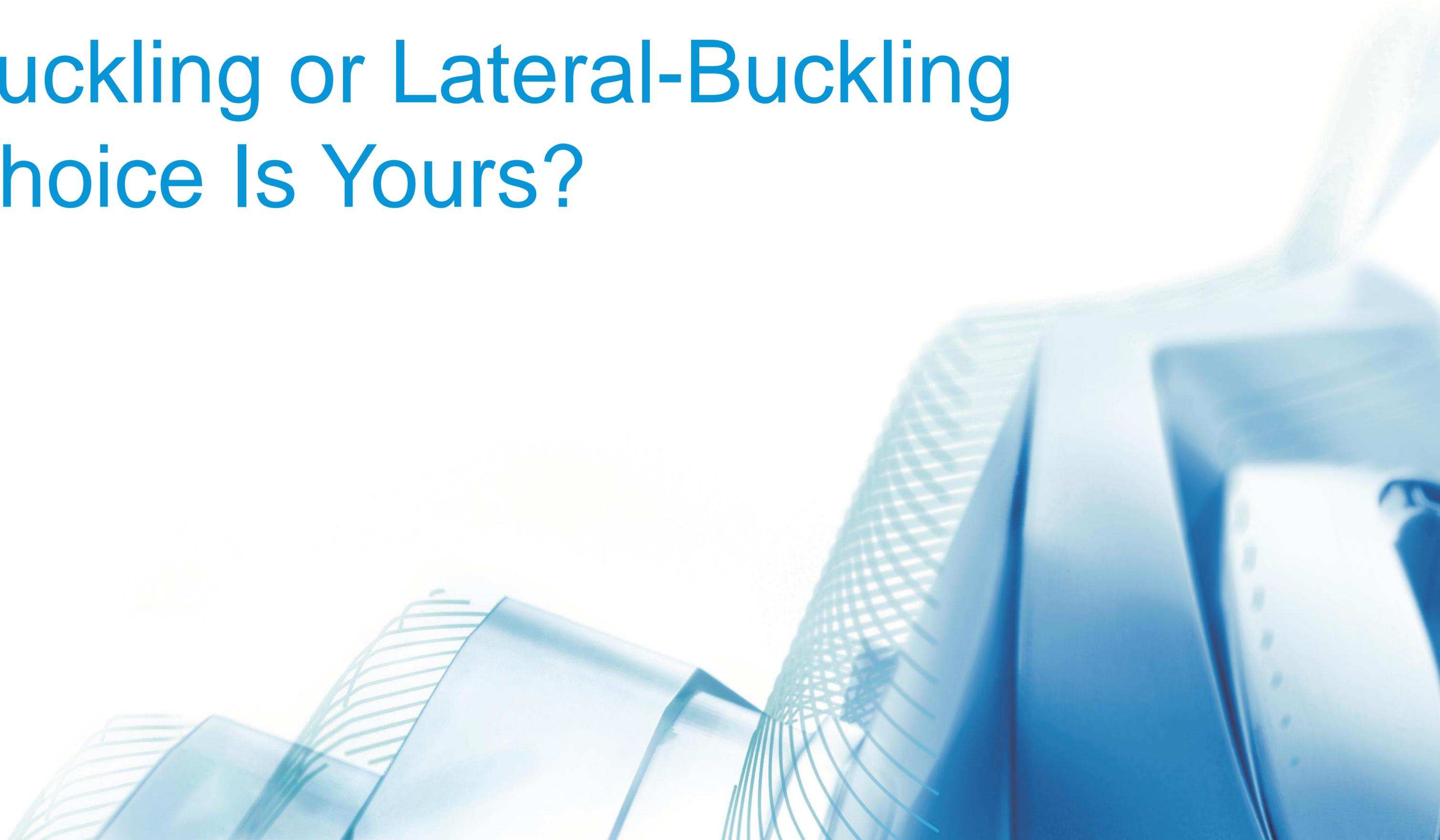
Releases vs. Truss bars!

We don't have to use too much bar releases. Especially in trusses.

In such situations, Truss bars are much better and more stable.



Buckling or Lateral-Buckling Choice Is Yours?

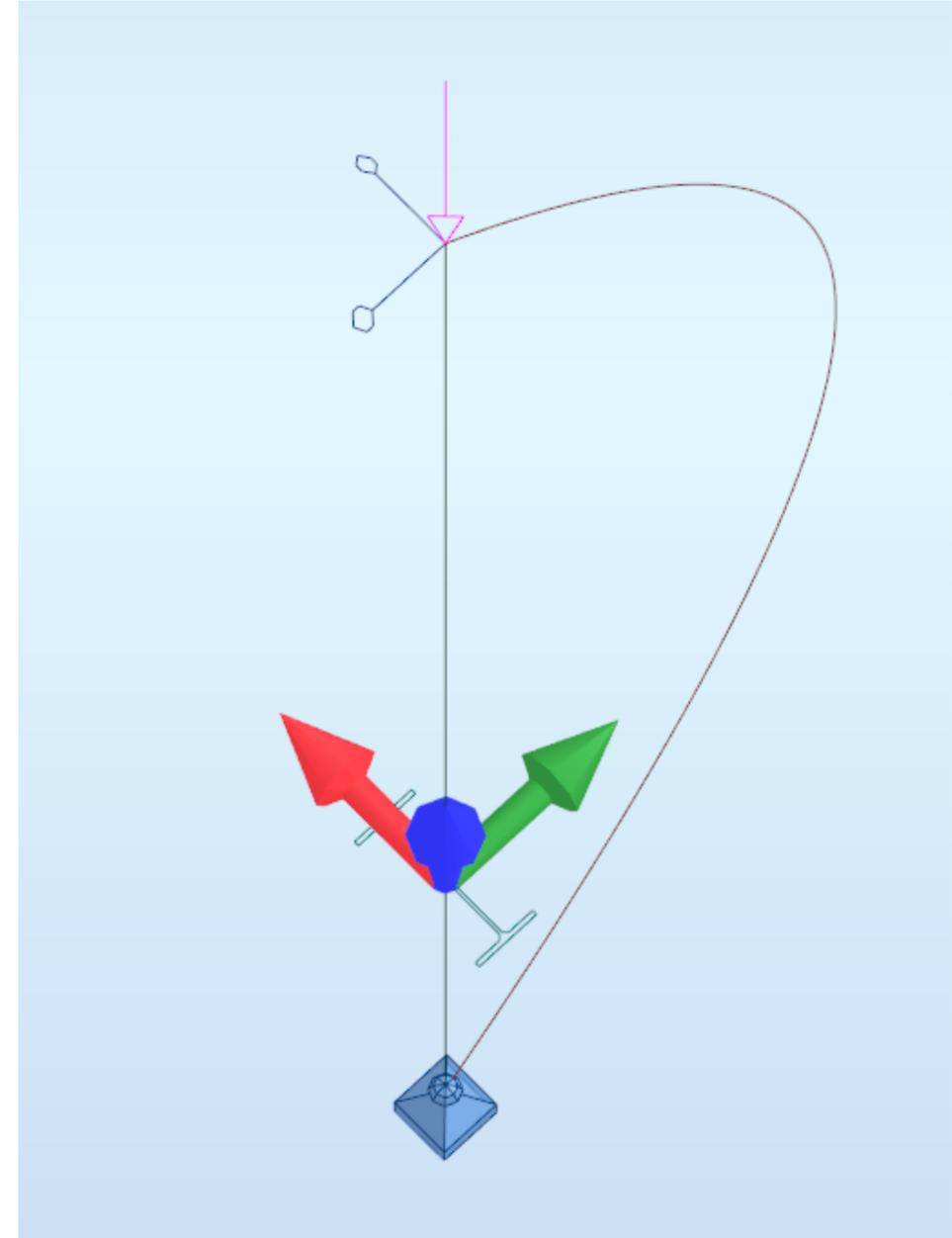


Buckling!

Buckling may occur in the elements subjected to compressive axial stress. The entire cross-section of the element must be compressed.

In such a situation, instability may occur where the element "bends" in the weaker direction.

In engineering practice, we assume that stretched elements cannot buckle.



Buckling in steel member definition!

In the case of steel dimensioning, we determine whether the indicated element will be verified for buckling.

Remember, not only the columns are compressed!

The image displays two software windows related to steel member buckling analysis.

Member Definition - Parameters - ANSI/AISC 360-16

Member type: Simple bar

Buckling (Y axis)
Member length ly: Real Coefficient 1,00
Buckling length coefficient Y: Ky: 1,00

Buckling (Z axis)
Member length lz: Real Coefficient 1,00
Buckling length coefficient Z: Kz: 1,00

Flexural-torsional buckling

Lateral buckling parameters
 Lateral buckling
Lateral buckling length coefficient
Cb: 1,00 Upper flange Lb = l Lower flange Lb = l

Seismic analysis parameters
 Seismic calculations - ANSI/AISC 341-16
System: [SMF] Special Moment Frames
Element type: Other

Internal bracings

Buckling Y

Test for member: 1 Simple bar_1

Define segments between bracings
 Define manually coordinates of the existing bracings
 * L
 real relative

Add automatically coordinates of bracings
 at points with adjoining elements in the buckling plane
 at all points where internal nodes are located
 at points where bending moments equal zero

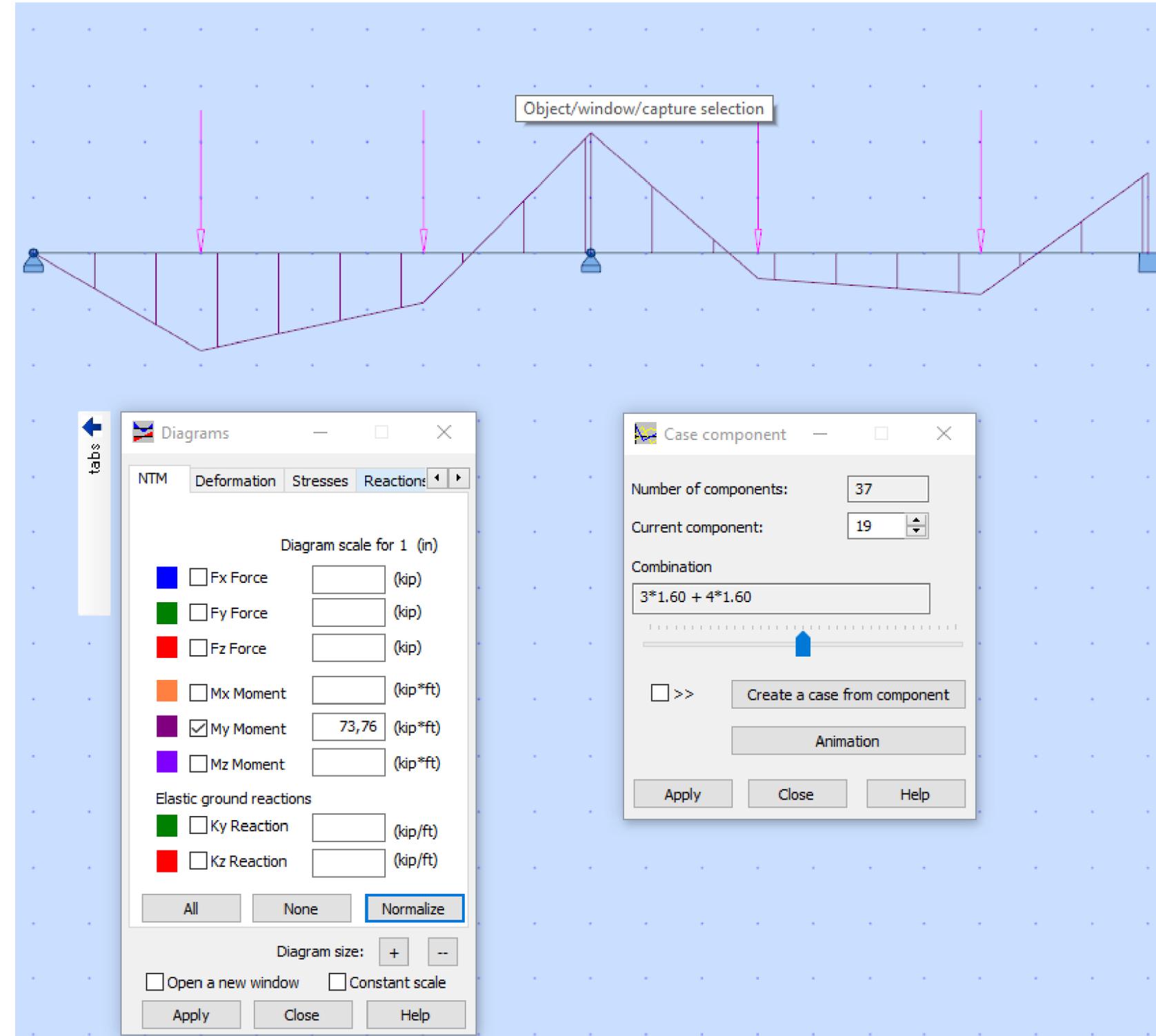
Buckling coefficients of component segments
Structure
 Sway Non-sway
1,00; 1,00

Bracing detection preview
For member no.: 1 Simple bar_1
For load case: 1 DL1
<< 0,67 * L
 real relative

Lateral Buckling!

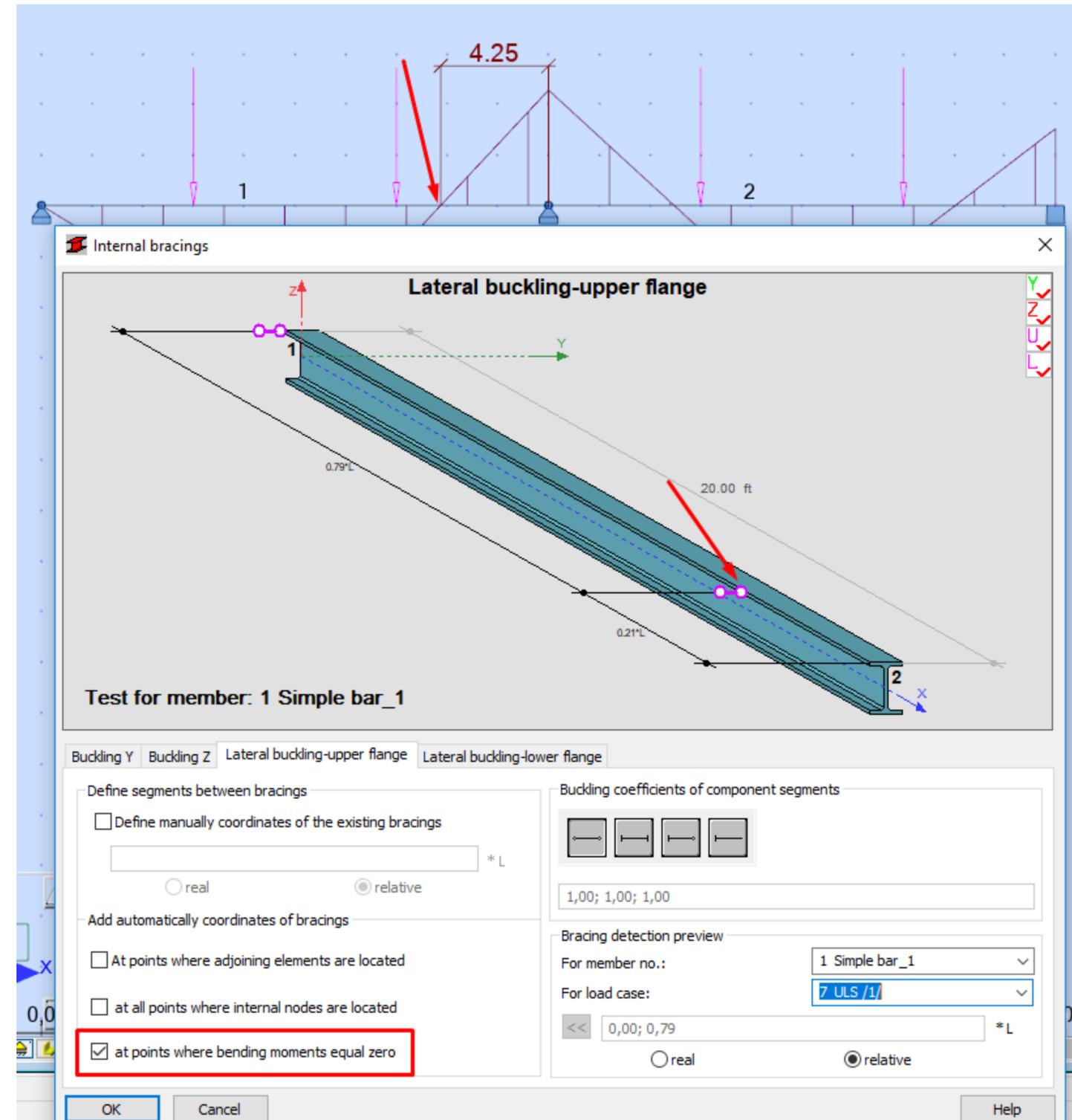
Lateral buckling may occur in the elements subjected to bending. In such case part of the cross-section of the element is compressed while the rest of the cross-section is tensioned. In such a situation, instability may occur only in compressed part.

In practice, we must determine the length of the element at which the compression of the cross-section part occurs.



Lateral Buckling in steel member definition!

Indication of the length of the element where the compression of section parts occurs can be done by using the steel dimensioning module in the Robot. The basis for this analysis is a change in the bending moment sign. It is important that the program updates the subject range itself after the load is changed.



Can you play guitar?



Sometimes it's worth reading the formula!

Thanks to this, we know what to include in the model in order to get results that correctly describe the behavior of the structure.

Focusing on the details of the structure will disturb the perception of the whole.



Eigenvalues and eigenmodes are obtained from the following formula.

$$\{K - \omega_i^2 M\}U_i = 0 \quad (1)$$

where:

K - stiffness matrix of the structure,

M - mass matrix of the structure,

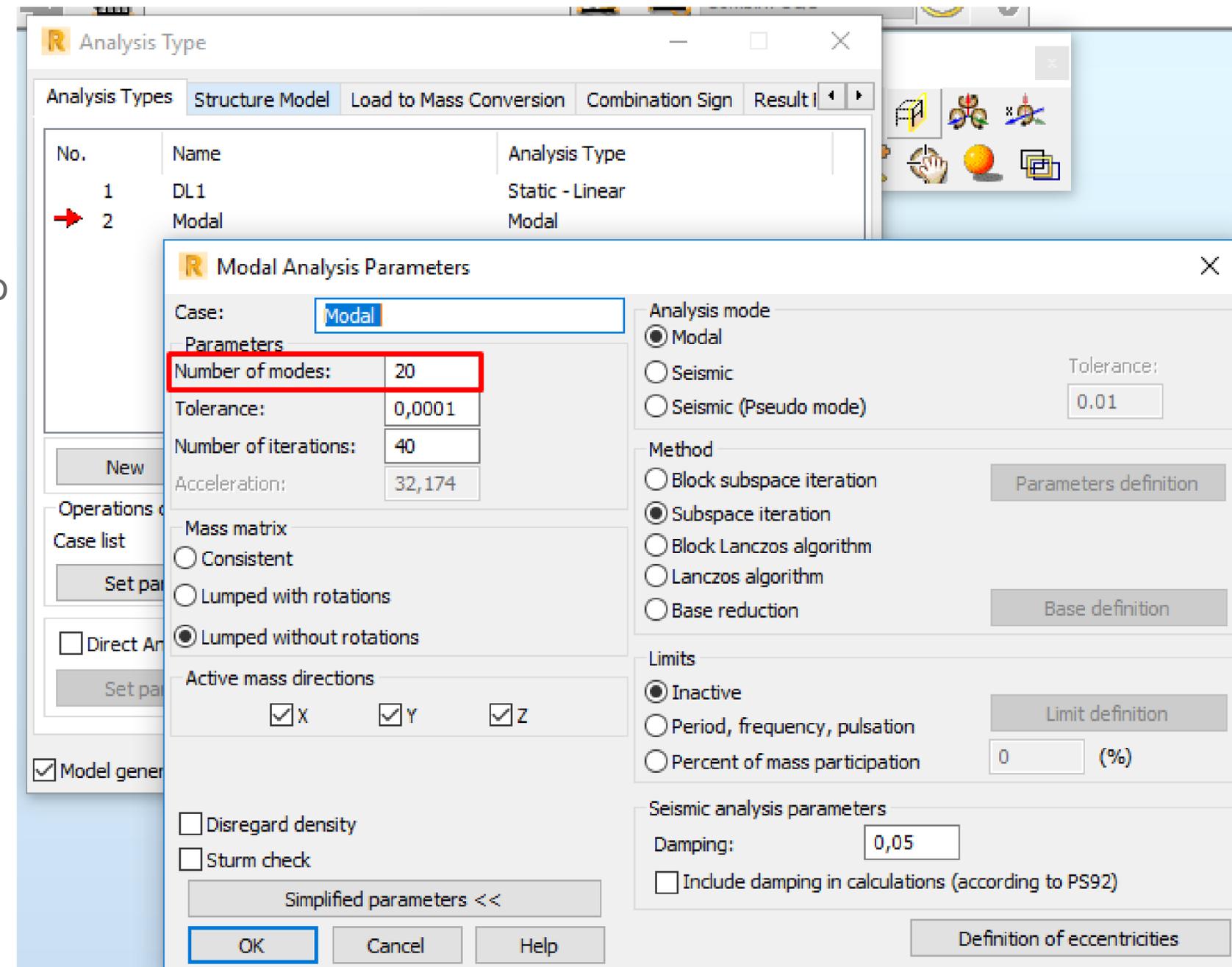
ω_i - natural pulsation (natural circular frequency) of mode "i",

U_i - eigenmode vector of mode "i".

Not too much, not too little!

Modal analysis provides information about frequencies and masses of participation for each mode. There shouldn't be too many of them, but when there are less of them, the program will reduce their number.

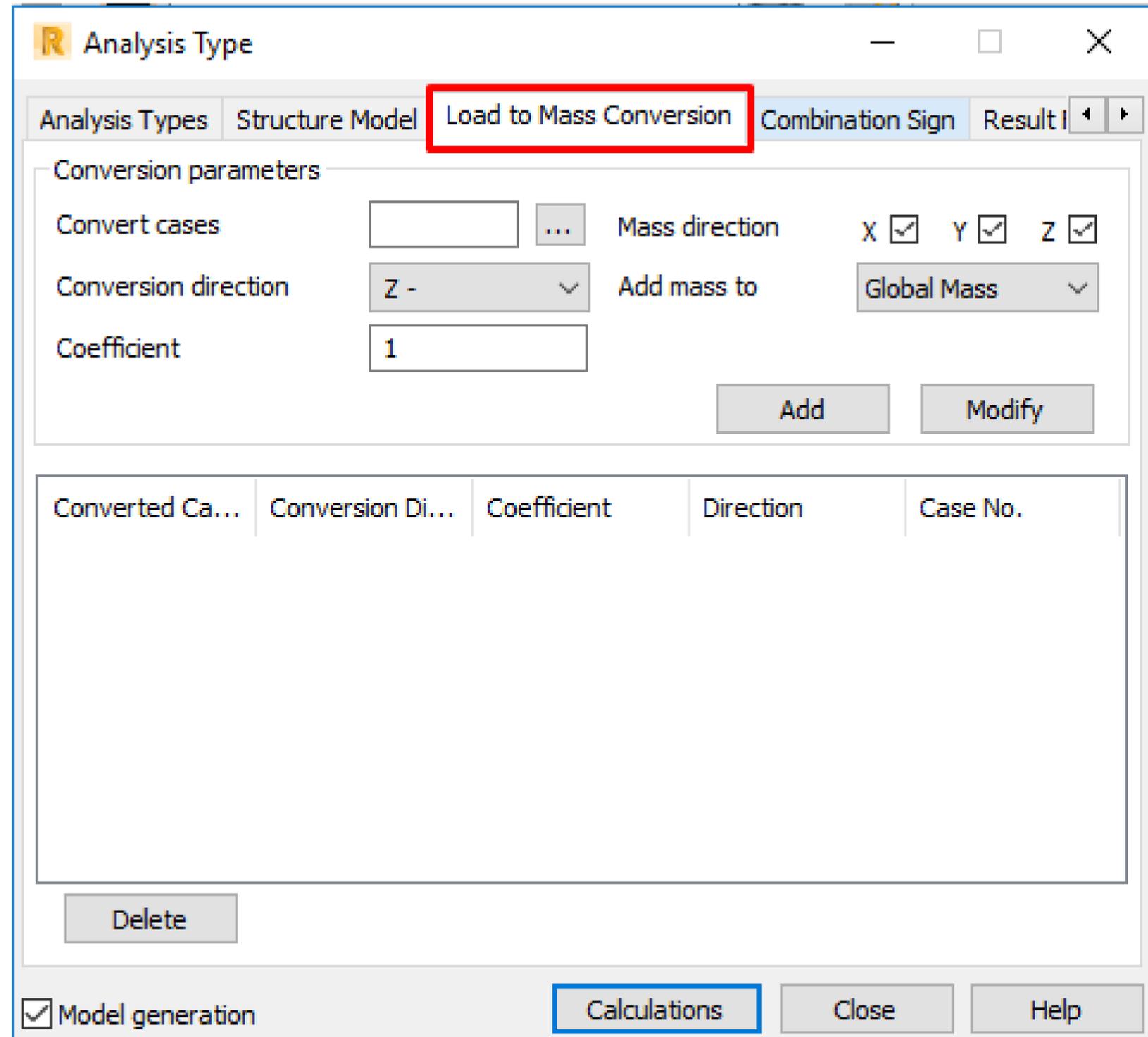
However, it will not increase the number, then we have to do it ourselves.



Massive vibrates differently!

The frequency of the pendulum's oscillations can be modified by changing its mass. The same applies to structural loads.

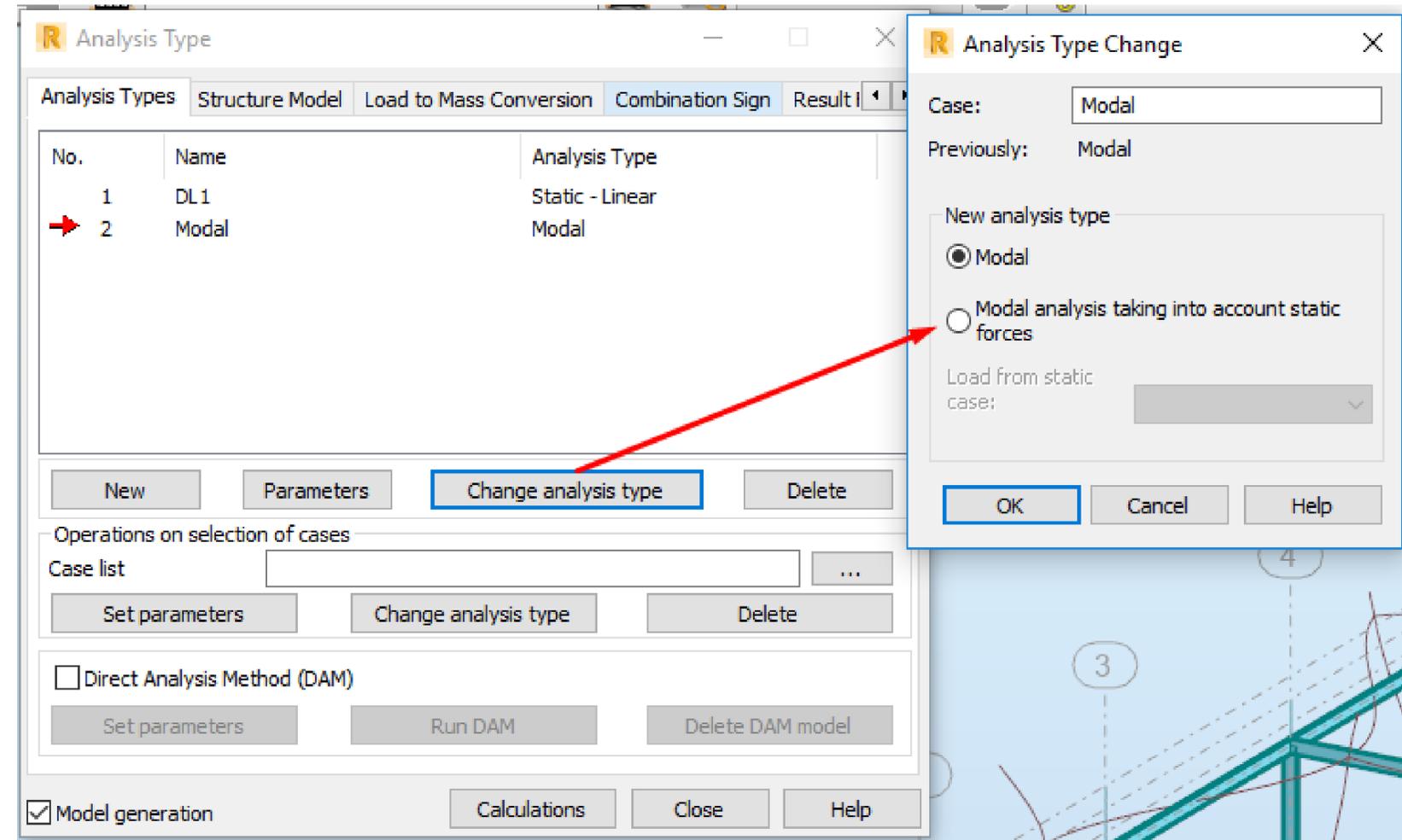
Converting loads into vibrating masses allows us to take this situation into account.



Force changes rigidity!

The more the string is tight, the higher the sound. The same applies to the construction.

Stressed components have a different frequency of vibrations.





AUTODESK®

Make anything™

Autodesk and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2019 Autodesk. All rights reserved.

