ProtaStructure®
ProtaSteel®
ProtaDetails®
ProtaBIM®

ProtaStructure Design Guide

Effective Stiffness Modifiers

Version: 1.0 16.06.2022

Please get in touch with us for your training and technical support queries

asiasupport@protasoftware.com

globalsupport@protasoftware.com

Limitation of Responsibilities

Prota shall not be held responsible for any losses caused by documentation, software, or usage errors.

In addition to Prota License Agreement Terms, it is the responsibility of the user

- to check results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- make sure that the software is used correctly per the reference manual and documentation,

Intellectual Property

ProtaStructure is a registered trademark of Prota Software Inc., and all intellectual property rights belong to Prota Software Inc. Documentation, training, and reference manuals. Any program component cannot be copied, distributed, and used in violation of the license agreement.

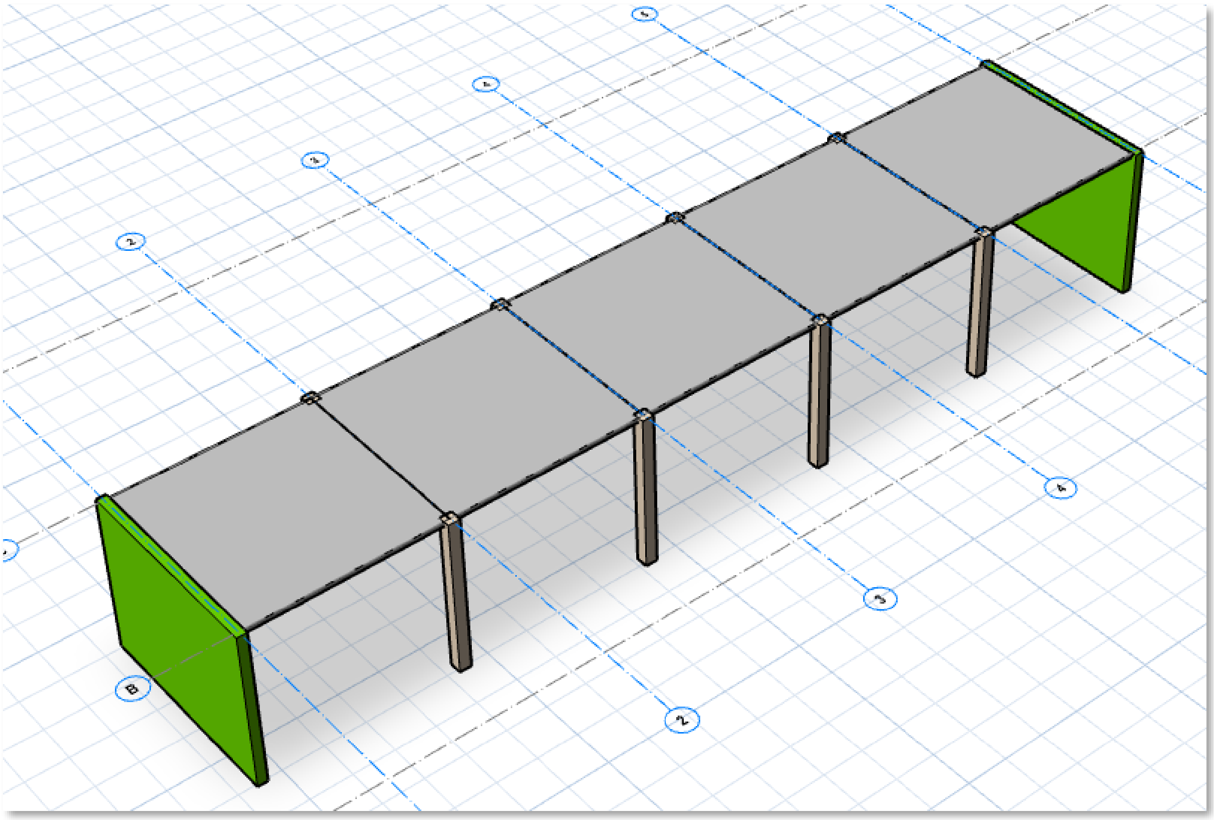
Trademarks

ProtaStructure®, ProtaDetails®, ProtaSteel® ve ProtaBIM® are registered trademarks of Prota Software Inc. Prota logo is a trademark of Prota Software Inc.

Table Of Contents

Model

A slender floor plan with shearwalls at the far edges is chosen for this case study. A flexible diaphragm option will be used. Seismic forces will be calculated using the equivalent static load method. The behavior of the structure with different slab stiffness modifiers will be investigated.



3D model of the structure

Slab Analysis and Design Methods

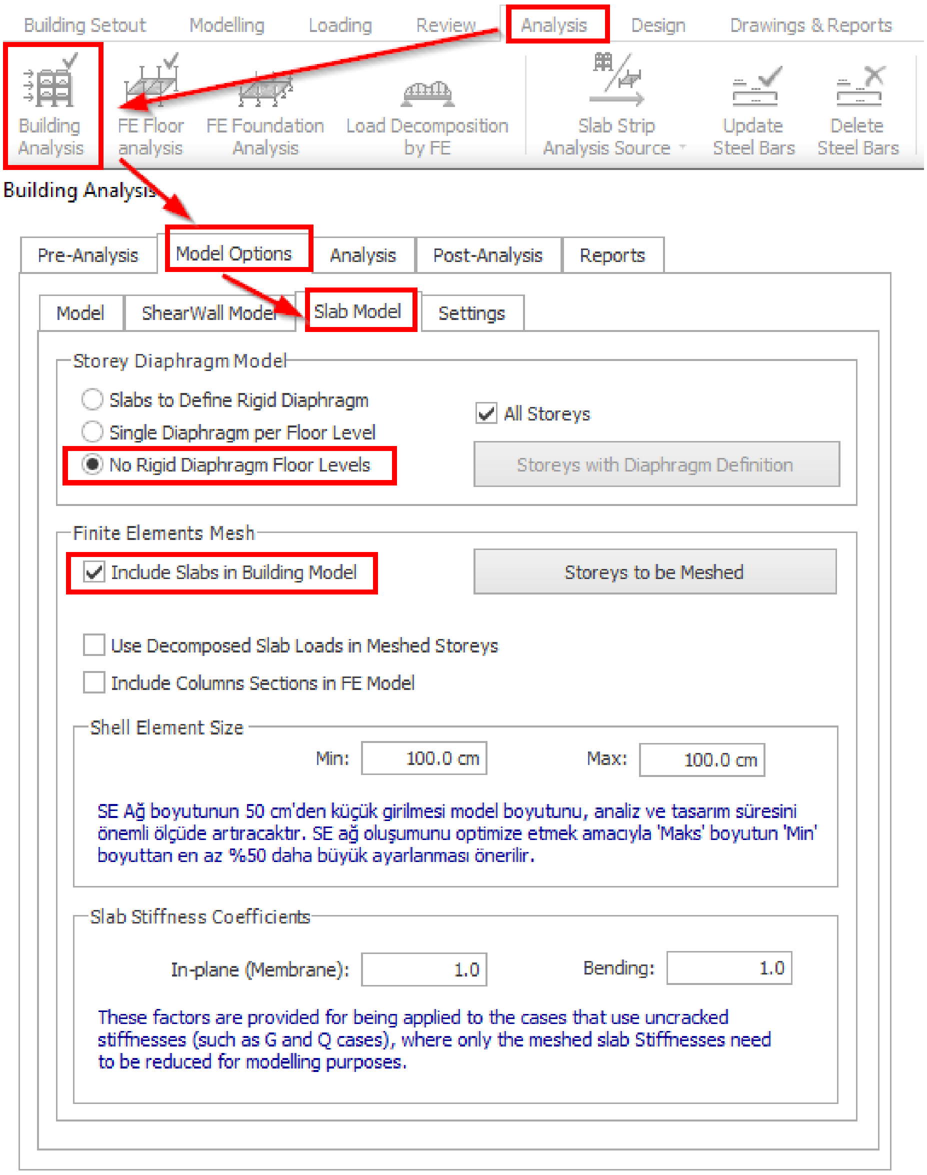
In ProtaStructure, slabs can be analyzed and designed with three different methods. These methods are:

- Analytical approach with code-based Moment Coefficients Method
- FE Analysis by including slabs in the global building model for all load combinations
- Analysis using the "Finite Elements Floor Analysis" option independent from the building under vertical load cases only.

In this study, effective stiffness modifiers will be discussed by using the last two methods

Inclusion of Slabs to Building Model for All Load Combinations

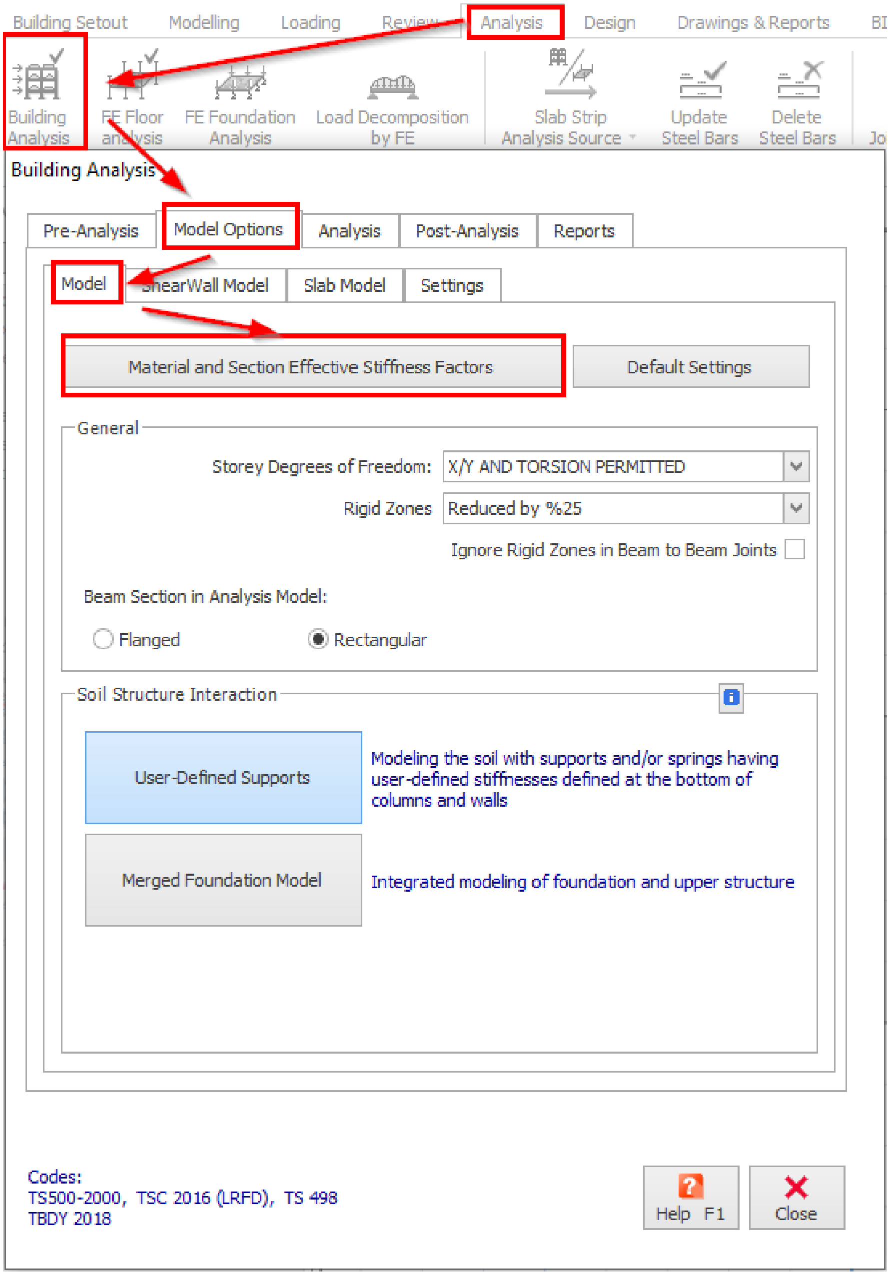
As shown below, slabs can be included in the building model in the “Analysis > Building Analysis” section.



Including Slabs in Building Model

Effective Stiffness Modifiers for Earthquake Load Cases and Combinations

ProtaStructure generates effective stiffness modifiers automatically if a seismic code is selected. Also, users can customize these modifiers manually. This option can be applied as stated below.



Effective Stiffness Modifiers Location

Effective Material and Section Stiffness Factors

	Elasticity Modulus	Axial Area		Bending Stiffness	Shear Area	Torsional Constant
Shearwalls (Shell)	1.00	0.50	In Plane	0.50	0.50	1.00
			Out of Plane	0.25	1.00	
Shearwalls (Frame)	1.00	0.50	Major	0.50	0.50	1.00
			Minor	0.50	0.50	
Basement Walls	1.00	0.80	In Plane	0.80	0.50	1.00
			Out of Plane	0.50	1.00	
Slabs	1.00	0.25	In Plane	0.25	0.25	1.00
			Out of Plane	0.25	1.00	
Columns	1.00	1.00		0.70	1.00	1.00
Beams	1.00	1.00		0.35	1.00	0.10
Coupling Beams	1.00	1.00		0.15	1.00	0.10

You can modify the elasticity modulus, section areas, moment of inertias and torsional constants of the member groups to be used in the analysis model. For example, you can enter 0.05 to reduce the moment of inertia values by 95% to reduce the lateral stiffnesses of the columns.

Note: In order to apply these factors, building analysis must be repeated. These factors will be applicable only for load cases for which cracked section properties are used.

Default Values | ? | OK | Cancel

Effective Stiffness Modifiers in Turkish Seismic Code Generated Automatically

ProtaStructure uses these modifiers only in load cases where the cracked section option is used.

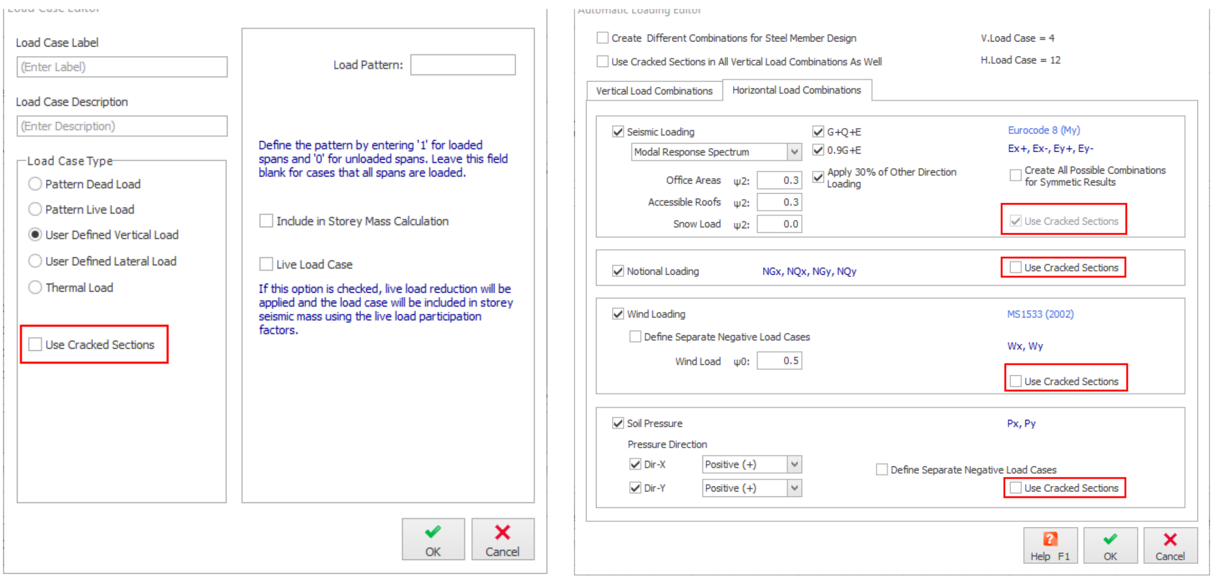
For example, according to the Turkish EQ code, a vertical load combination (1.4G + 1.6Q) must consider uncracked gross sections, whereas a seismic combination (i.e., G + Q + E) must consider cracked sections. That means we have to analyze the structure for two sets of G and Q; one cracked and one uncracked. ProtaStructure creates Gc and Qc load cases for cracked section analysis and uses effective stiffness modifiers for cracked sections for these load cases. Thus, the final load combination becomes Gc + Qc + E.

Explicit Use of Effective Stiffness Modifiers

By default, ProtaStructure will consider cracked sections only for earthquake load cases (if required by the selected seismic code). Uncracked gross section properties are used for non-seismic load cases.

It does not have to be this way. Suppose the "Use Cracked Sections" option or "Use Cracked Sections in All Vertical Load Combinations As Well" option is selected while generating the load combinations. The specified load cases will also use cracked section modifiers in that case.

The cracked section option can also be activated for "User Defined Vertical Load" and "User-Defined Horizontal Load".

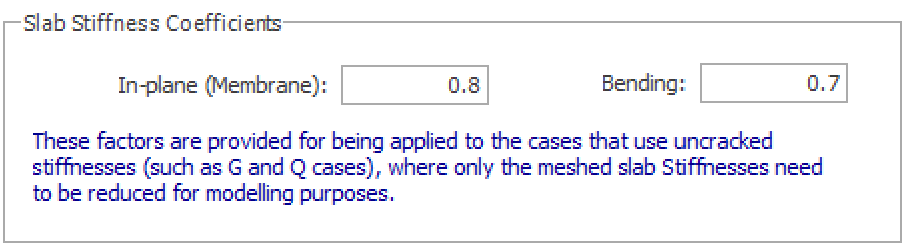


Activating the "Use Cracked Section" Option

Slab Stiffness Coefficients for Load Cases using Uncracked Sections

ProtaStructure pays special attention to slab modeling when it comes to cracked sections. Unrealistic slab section modifiers can affect the analysis results considerably.

Users can specify additional slab stiffness modifiers for load cases that do not use cracked sections using the "**Analysis > Building Analysis > Model Options > Slab Model** >**Slab Stiffness Coefficients**" fields. "**In-Plane**" and "**Bending (Out-of-Plane)**" stiffness modifiers can be chosen between 0.3 - 0.8 and 0.25 – 0.7 by engineering judgment, depending on the structure.

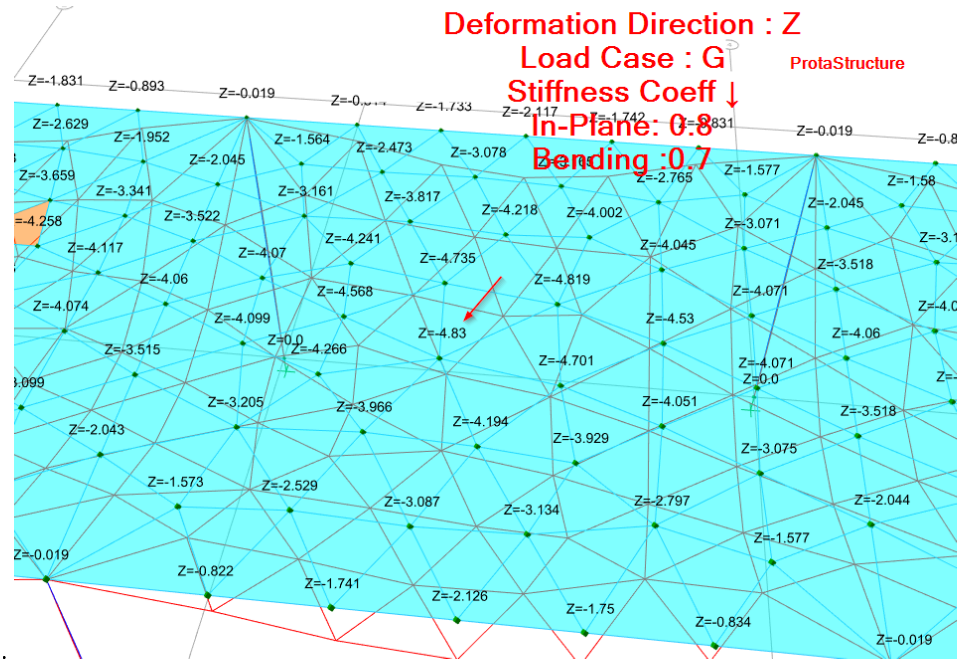


Choosing Stiffness Modifiers for Slabs

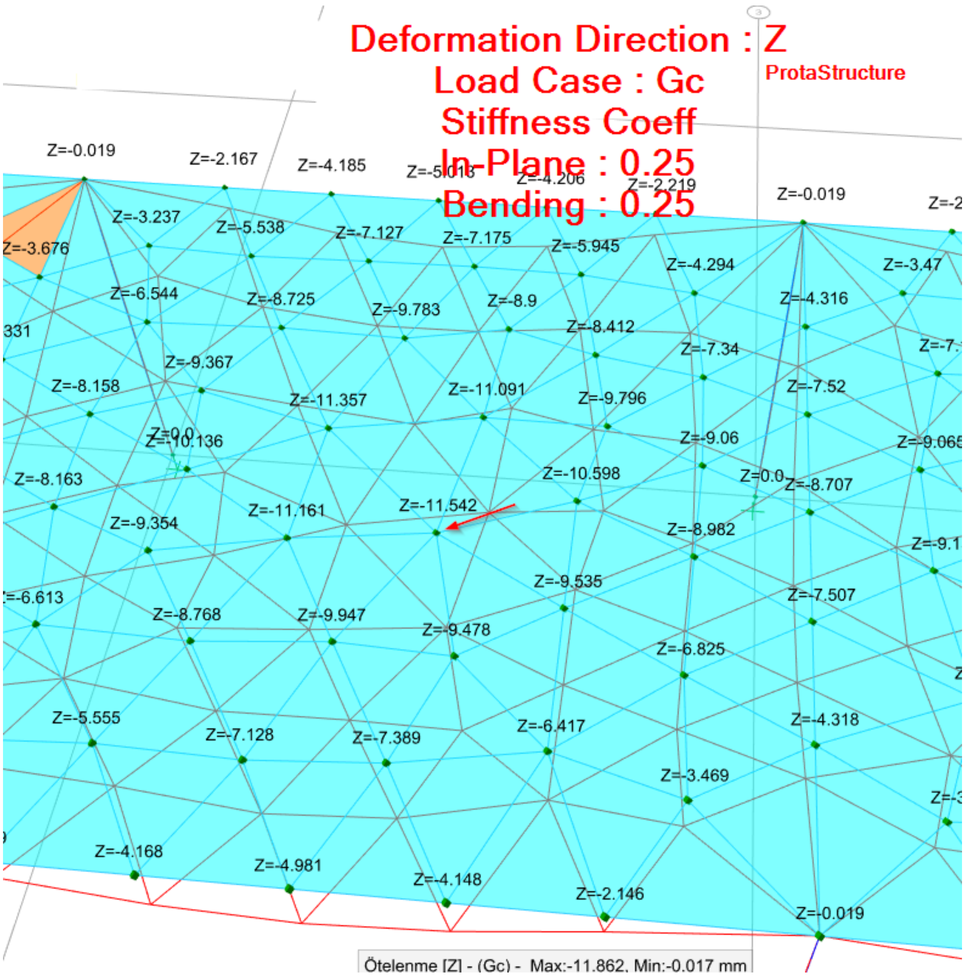
Important

These coefficients are only valid for load cases using uncracked sections. Load cases that are adjusted to use cracked sections utilize the modifiers explained in the previous section.

To explain it better, two analysis results are shown below for both **G** and **Gc** load cases. In the first one, slab stiffness coefficients are determined as 0.8 for In-Plane behavior and 0.7 for Bending behavior. In the second one, coefficients were taken from the code as 0.25 for both In-Plane and Bending behaviors.



Deformation in Z For Load Case G

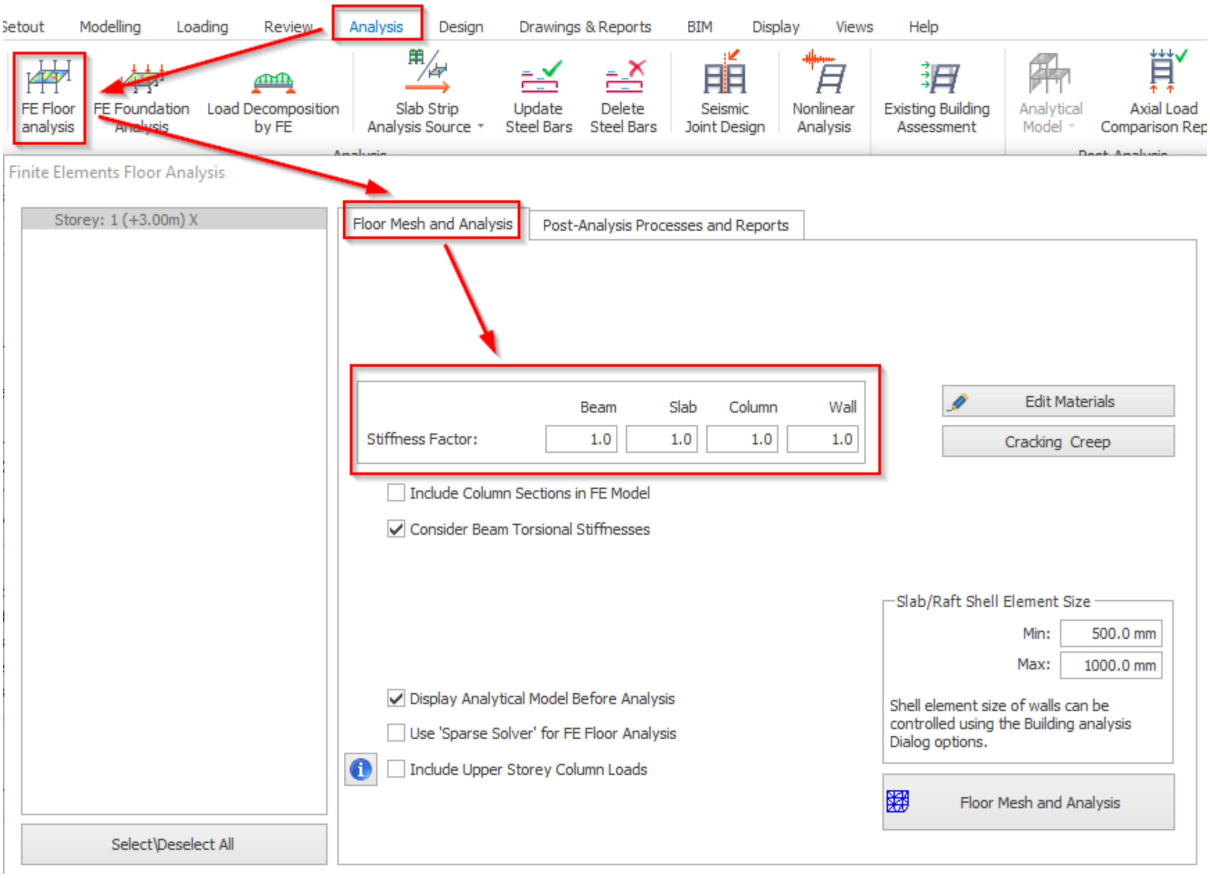


Deformation in Z For Load Case Gc

Analysis Using “Finite Elements Floor Analysis” Option

This method analyses the floors by isolating them from the building analytical model. This analysis option is only for load combinations that include vertical load cases. In other words, all horizontal load cases are ignored. This method is analogous to “Short Frame Analysis.”

This method can be reached from **Analysis > FE Floor Analysis** tab. In this window, effective stiffness modifiers can be adjusted. The stiffness modifiers here only affect the bending behavior of the slabs.



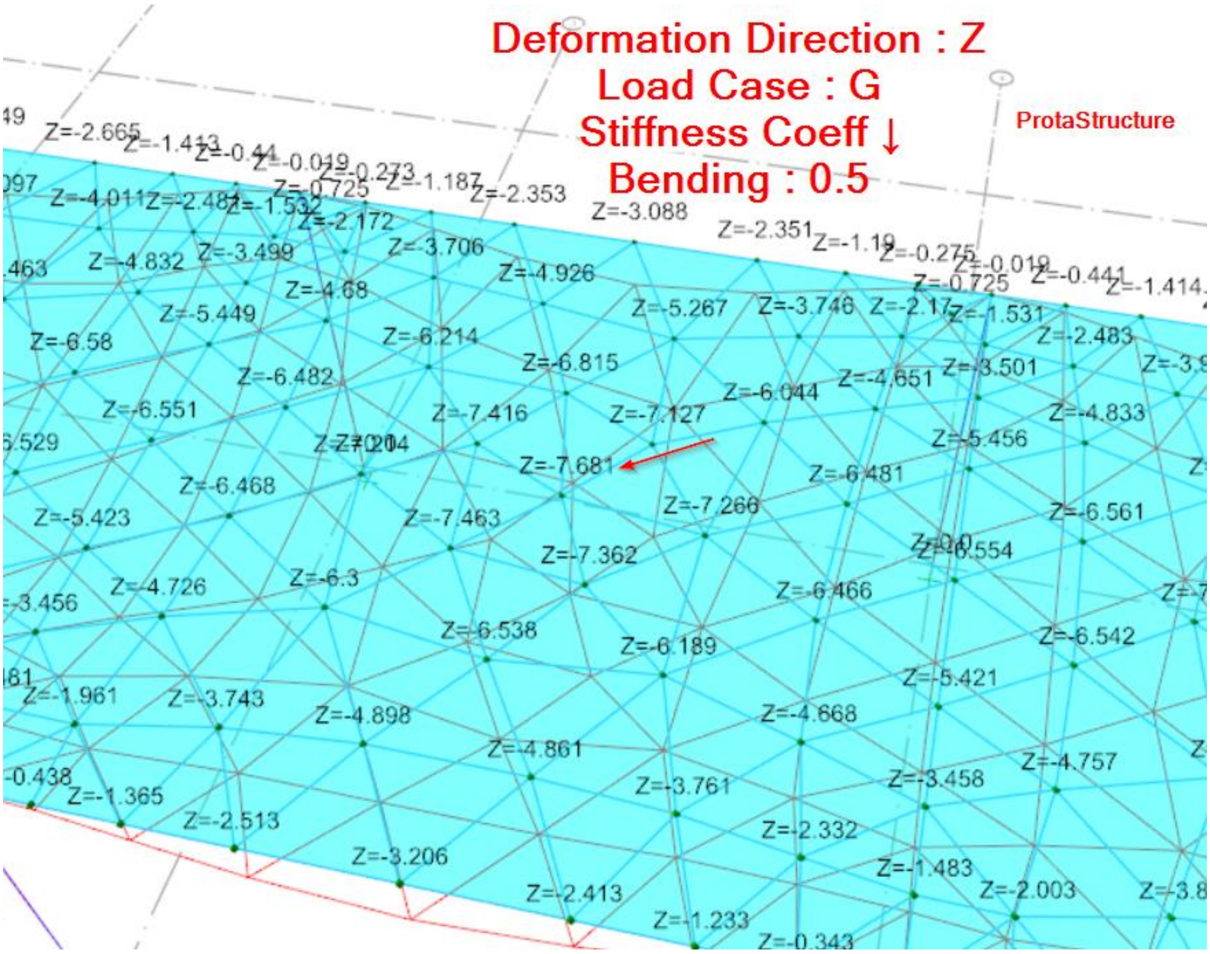
Stiffness Modifiers for Finite Element Floor Analysis

This analysis option recommends picking a stiffness modifier value between 0.25 – 0.7.

Unlike the building analysis model, in-plane degrees of freedom (DoF) are locked in the Finite Element Floor Analysis. In other words, in-plane shear, tension, and compression stresses will not be computed. This analysis method is developed to observe slabs' bending deformation under vertical load cases.

Using the first analysis option (including slabs in the building model) is advised to obtain more complete analysis results.

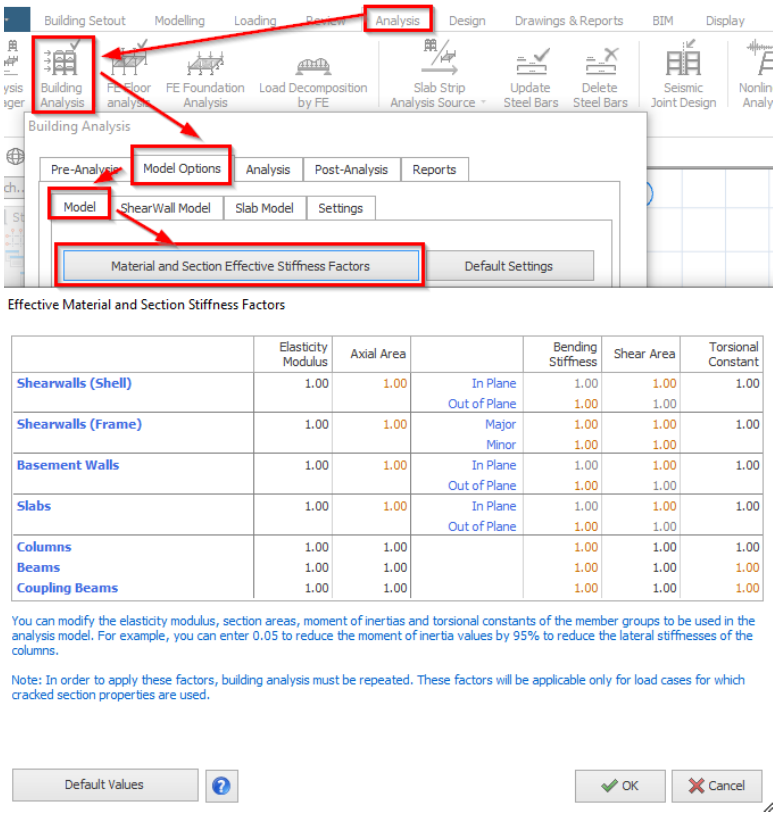
The following is sample output from FE Floor Analysis. The stiffness coefficient for the slab is taken as 0.5.



Deformation in Z For Load Case G

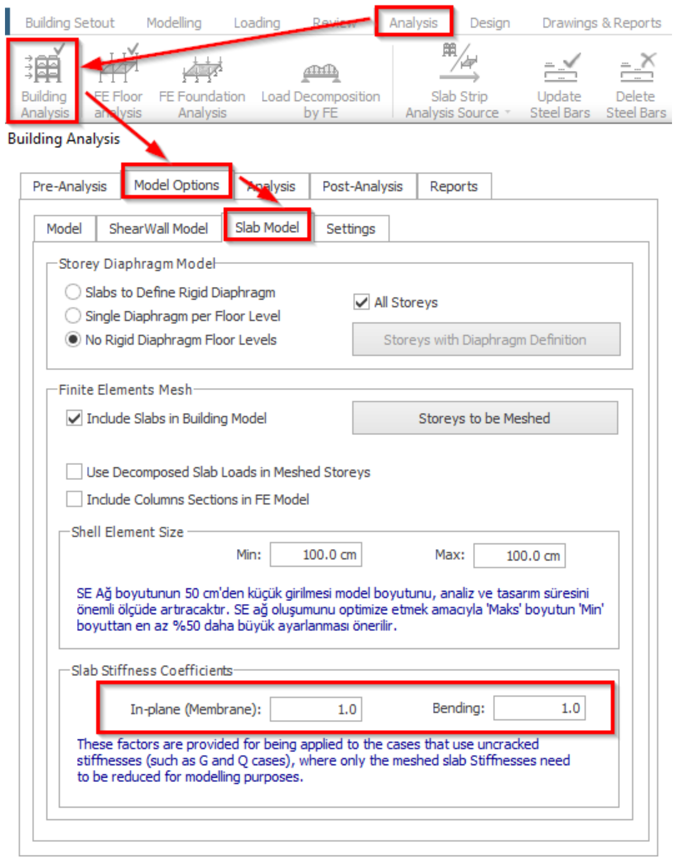
Comparison of Results

To compare results, an analytical model created automatically by ProtaStructure can be exported directly to SAP2000. Since cracked and uncracked sections are used together in the same model, and SAP2000 does not support this feature, ProtaStructure always exports cracked section stiffness coefficients. Because of this issue, all stiffness coefficients should be set to 1 to obtain comparable values.

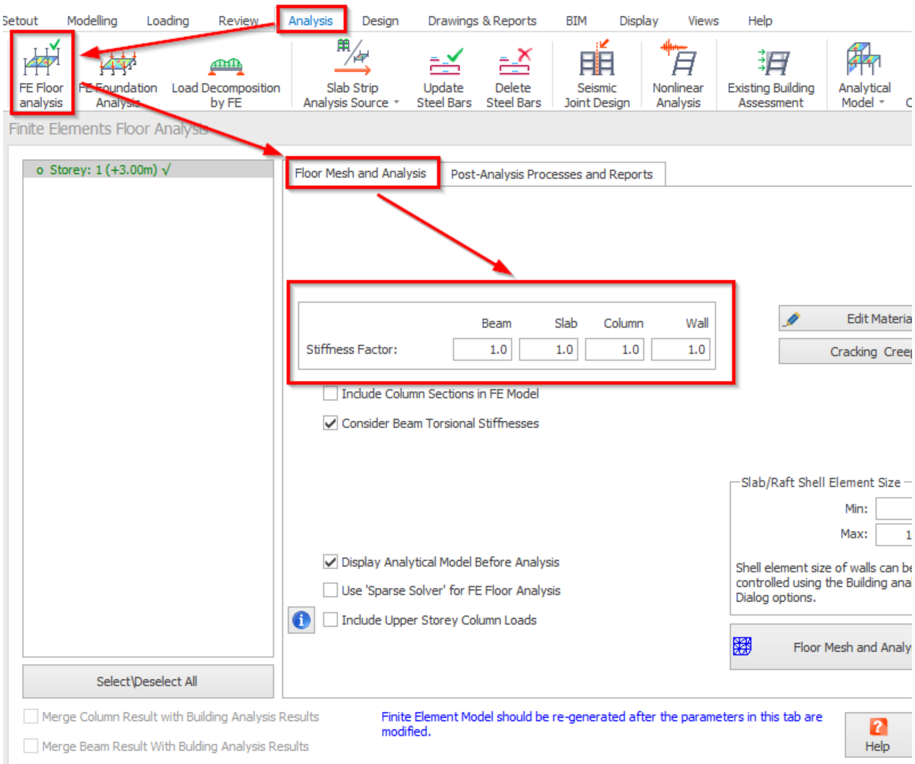


	Elasticity Modulus	Axial Area		Bending Stiffness	Shear Area	Torsional Constant
Shearwalls (Shell)	1.00	1.00	In Plane	1.00	1.00	1.00
			Out of Plane	1.00	1.00	
Shearwalls (Frame)	1.00	1.00	Major	1.00	1.00	1.00
			Minor	1.00	1.00	
Basement Walls	1.00	1.00	In Plane	1.00	1.00	1.00
			Out of Plane	1.00	1.00	
Slabs	1.00	1.00	In Plane	1.00	1.00	1.00
			Out of Plane	1.00	1.00	
Columns	1.00	1.00		1.00	1.00	1.00
Beams	1.00	1.00		1.00	1.00	1.00
Coupling Beams	1.00	1.00		1.00	1.00	1.00

Arranging Stiffness Coefficients For Cracked Sections in Building Analysis

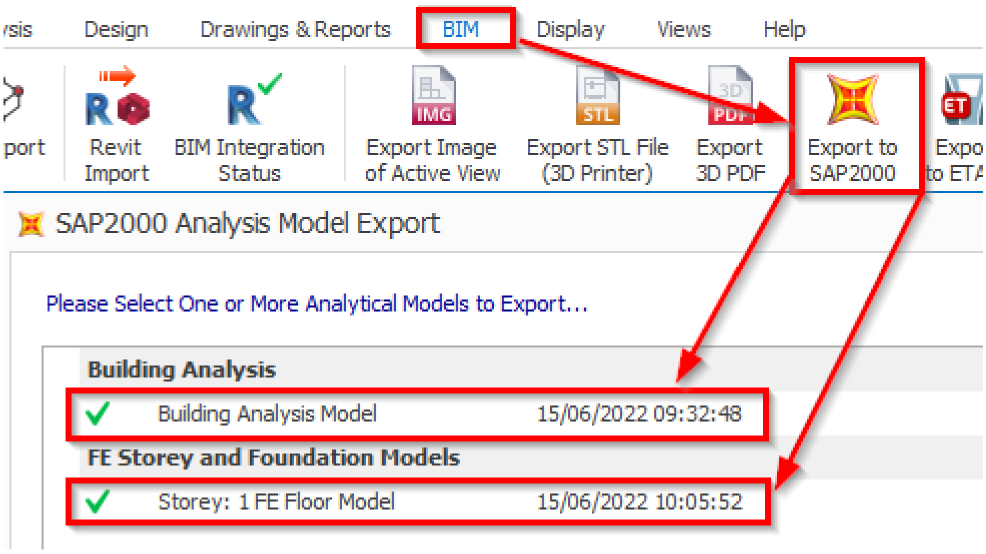


Arranging Slab Stiffness Coefficients For Uncracked Sections in Building Analysis



Arranging Stiffness Coefficients in Finite Element Floor Analysis

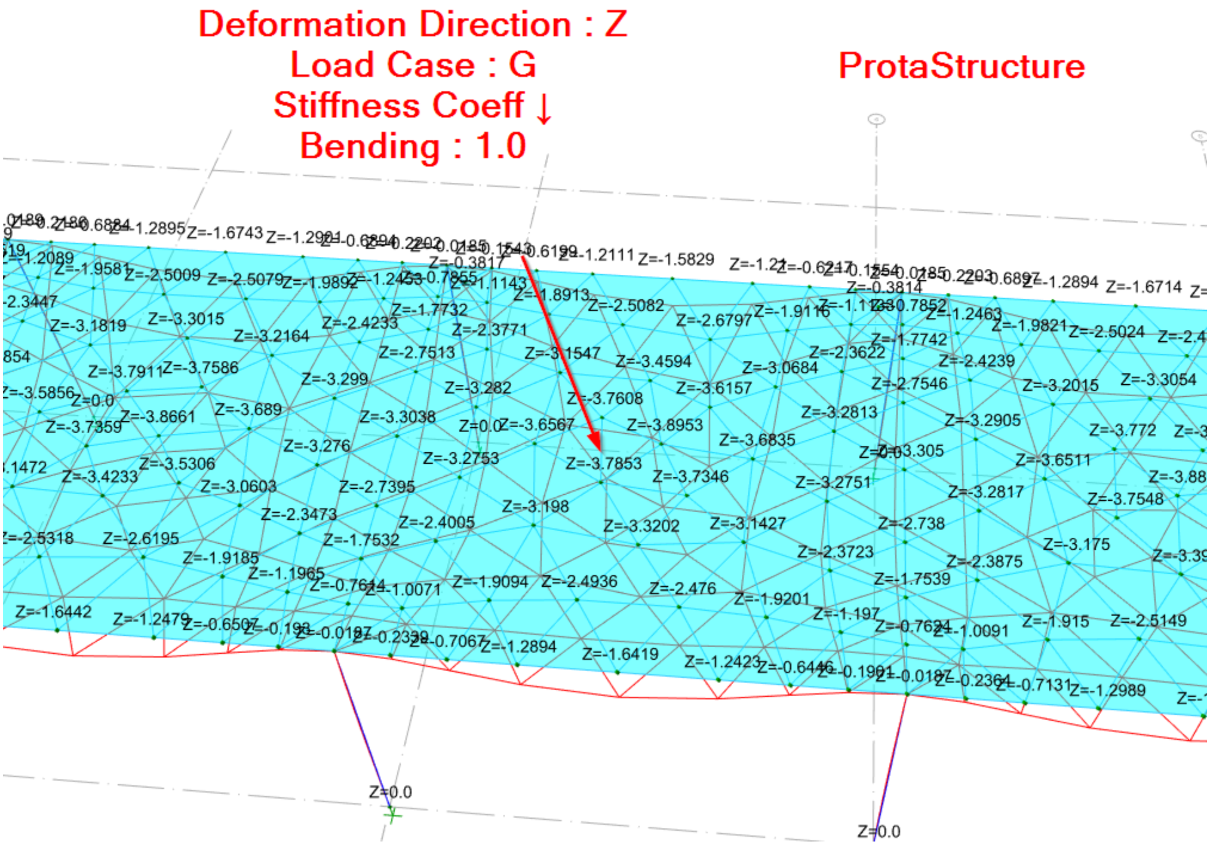
After these configurations, all analyses should be repeated to compare the results. Then, from the ribbon, BIM > Export to SAP2000 option can be used for both analysis method results within s2k format.



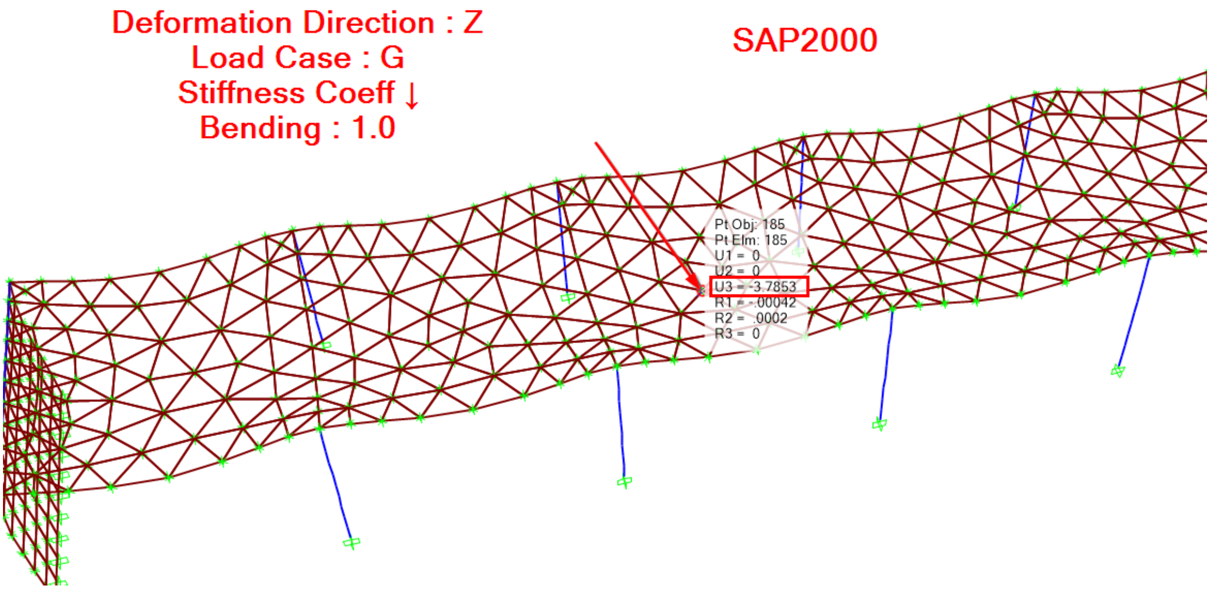
Exporting Models to SAP2000

Analysis Results of SAP2000 and Finite Element Floor Analysis Method

Analyses results can be examined below.

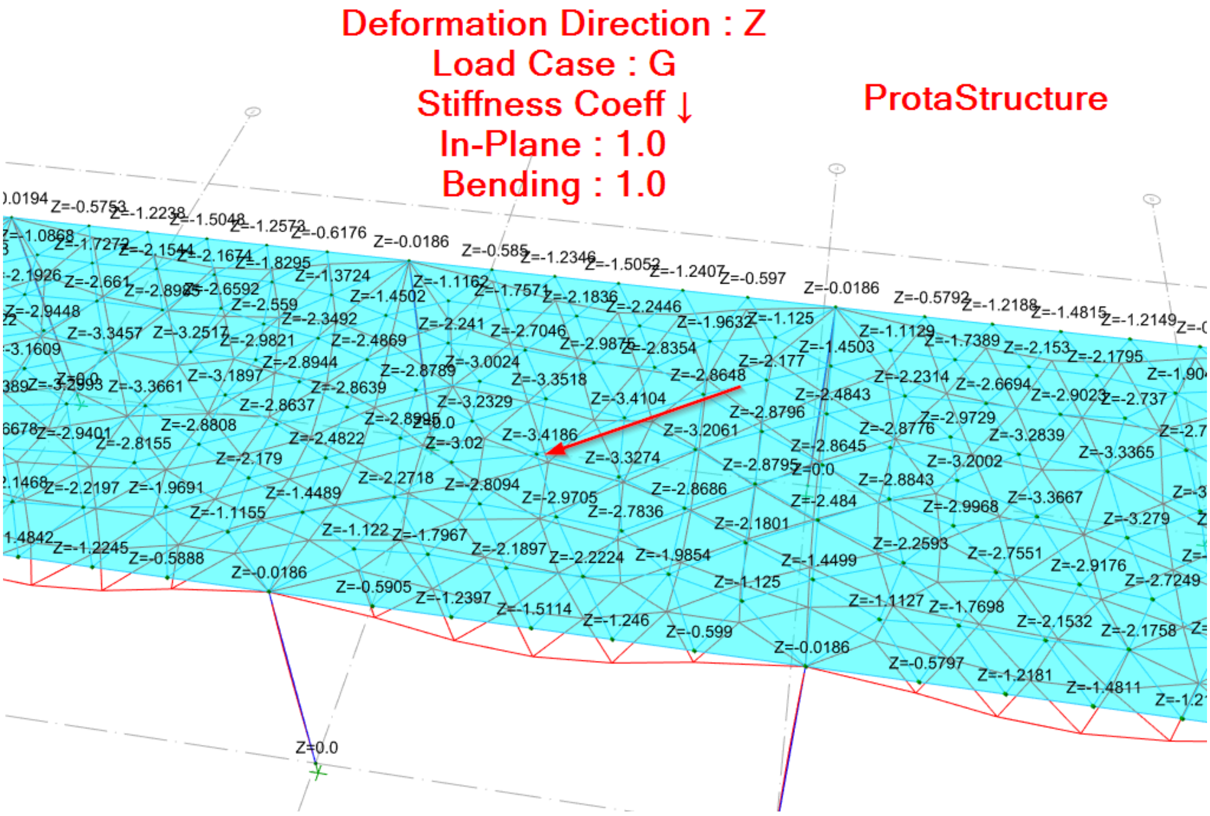


Deformation in Z Direction for Load Case G

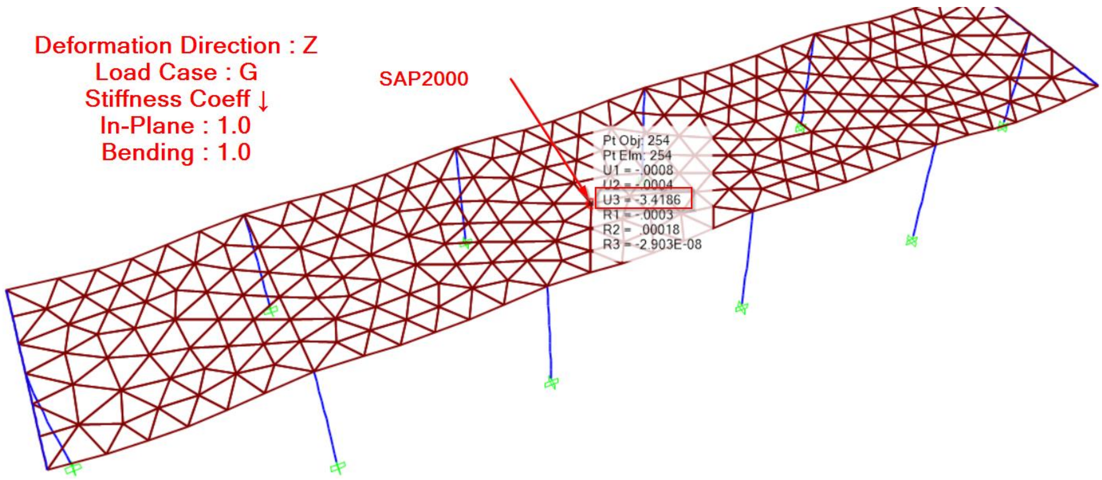


Deformation in Z Direction for Load Case G

Analysis Results of SAP2000 and Building Analysis Method

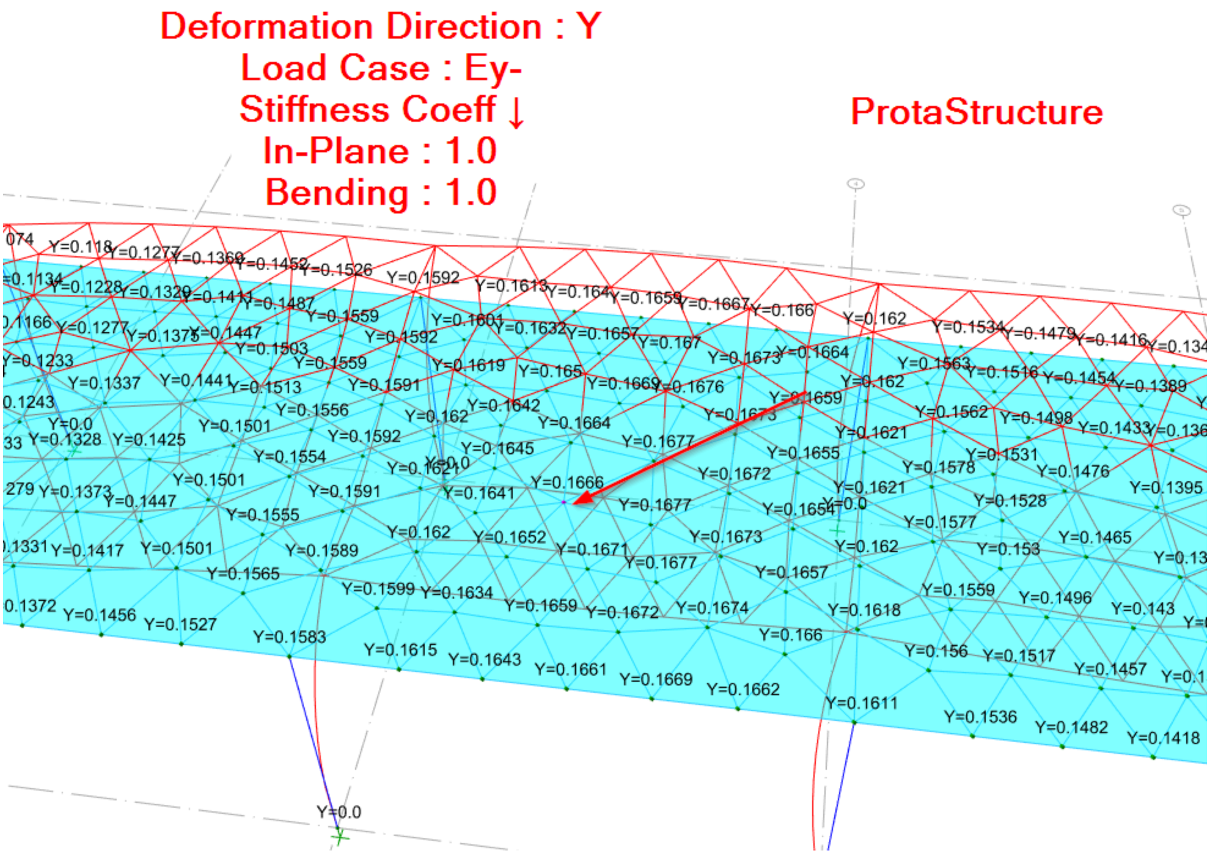


Deformation in Z Direction for Load Case G

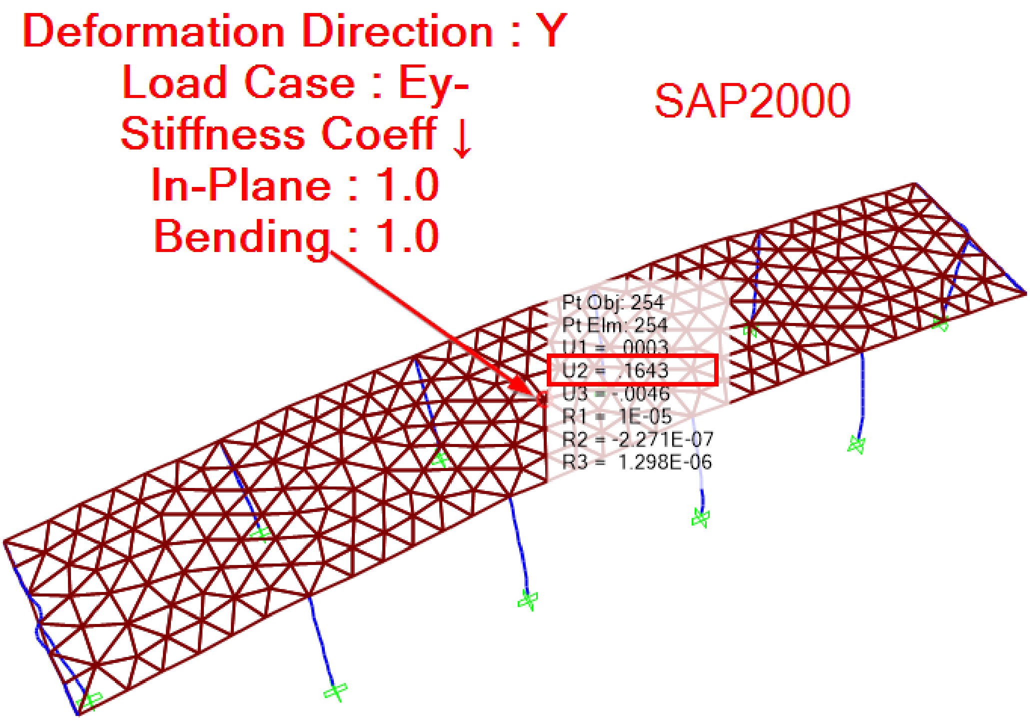


Deformation in Z Direction for Load Case G

Apart from the deformations in the Z direction, deformations in the Y direction are also checked, and the results are shown below.



Deformation in Y Direction for Load Case Ey-



Deformation in Y Direction for Load Case Ey-

Conclusion

Slabs can be analyzed using several methods with different stiffness coefficients in ProtaStructure.

Different stiffness coefficients can be selected in Building Model Analysis and Finite Element Floor Analysis with different user interfaces. By default, stiffness modifiers follow the selected seismic and design code provisions, but users are free to modify them.

Analysis models are exported to SAP2000, and results are compared.

All values are tabulated and presented as follows.

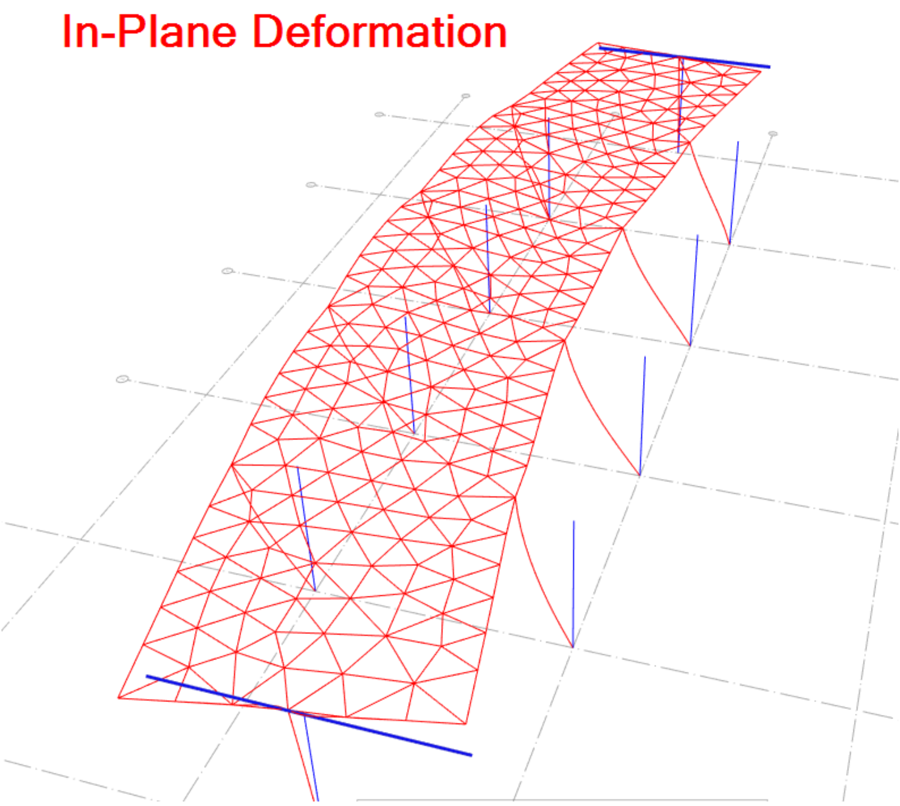
			Deformations (mm)		
			G	G_c	E_y-
			Direction Z		Direction Y
Slab Stiffness Mod.	In-Plane = 0.8 Bending = 0.7	Building Analysis Model	-4.83	-11.542	0.345
	Bending = 0.5	FE Floor Analysis Model	-7.681	-	-

Analysis Results from ProtaStructure

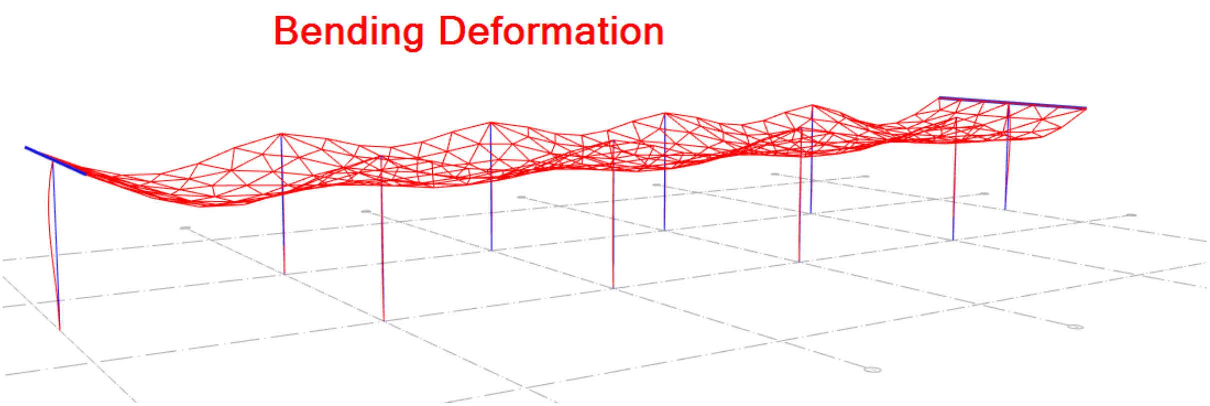
			Deformations (mm)		
			G	G_c	E_y-
			Direction Z		Direction Y
Slab Stiffness Mod.	In-Plane = 1.0 Bending = 1.0	PS - Building Analysis Model	-3.4186	-3.4186	0.1666
	In-Plane = 1.0 Bending = 1.0	SAP2000 – Building Analysis Model	-3.4186	-3.4186	0.1643
	Bending = 1.0	PS – FE Floor Analysis	-3.7853	-	-
	Bending = 1.0	SAP2000 – FE Floor Analysis	-3.7853	-	-

Comparison of Results

Based on the results, it is observed that stiffness and deformation have a negative correlation. In fact, as bending stiffness increases, deformation in the Z direction decreases. It is also valid for in-plane bending stiffness and deformation in the Y direction.



Demonstration of in-plane deformation



Demonstration of Bending Deformation

In these studies, it is observed that results from both ProtaStructure and SAP2000 are almost identical. The level of precision could change depending on the complexity and differences of models. All in all, the main lines to compare results are stated, and essential points for effective stiffness coefficients are underlined in this article.

Thank You…

Thank you for choosing the ProtaStructure Suite product family.

Our top priority is to make your experience excellent with our software technology solutions.

Should you have any technical support requests or questions, please do not hesitate to contact us at all times through globalsupport@protasoftware.com and asiasupport@protasoftware.com

Our dedicated online support center, together with our responsive technical support team, is available
to help you get the most out of Prota's technology solutions.

The Prota Team

ProtaStructure®
ProtaSteel®
ProtaDetails®
ProtaBIM®