

TUTORIAL INTEGRATION MEMBER

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2021 SCIA nv. All rights reserved.

Table of Contents

Table of Contents	
Introduction4	
Input and use the integration member5	
Input integration member	5
Extra parameters	6
Integration relative to position of	6
Number of sections	6
Selection of members for integration	6
LCS	7
Results of the integration member	7
Design and check of an integration member8	
Settings of the integration member	8
Example	9

Introduction

The integration member is a tool that enables integration of results from one or more 2D and 1D members and their subsequent presentation in the form of the internal forces and deformations of 1D members. Meaning that it enables the presentation of a set of 7 components of 1D internal forces: N, V_y, V_z, M_x, M_y, M_z, V_r and a set of 6 components of 1D deformation: u_x , u_y , $u_z \phi_x$, ϕ_y , ϕ_z , U_{total}

For concrete applications – for example: lintels, deep beams, beam slab, core wall, etc. – you can use this representation to design the reinforcement and check the element.

This tutorial will explain how to input the integration member, explain its properties and how to get the results. It will also discuss an example how you can design and check an integration member used in concrete applications.

Input and use the integration member

Input integration member

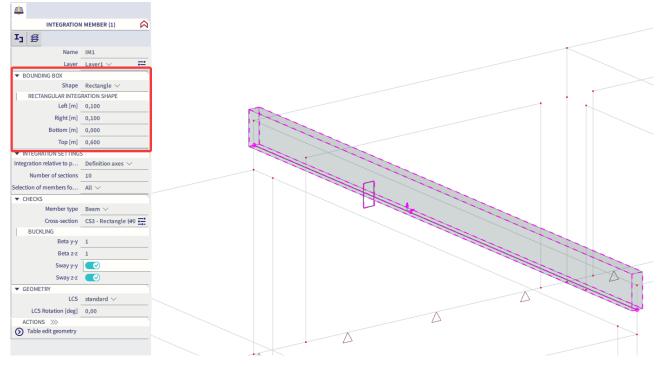
To input an integration member, you can simply type this into the SCIA Spotlight:

Integration member	?⊗
Integration member	

Another way is to find the functionality in the results toolbar:



You can input the integration member simply by clicking on the start and end point of the integration member. Next you need to check the bounding box. This bounding box defines which entities are considered in the integration:



Note that you have several options for the shape of the bounding box. You can use this to your advantages and use the optimal shape for your projects.

Extra parameters

After inputting the integration member, you can edit some other parameters to ensure the integration is correct:

	Integration member			×
	Name	IM2		^
4	Bounding box			
		Rectangle	~	
	Rectangular integration shape			
	Left [m]	0,100		
	Right [m]	0,100		
	Bottom [m]	0,000		
	Top [m]	0,600		
	Buckling			
	Beta y-y	1		
	Beta z-z			
	Sway y-y			
	Sway z-z			
	Member type		*	
		CS3 - Rectangle (400; 200)	×	
	Integration relative to position of		*	
	Number of sections			
	Selection of members for integration			
		standard	*	
	LCS Rotation [deg]			
	Layer	Layer1	×	
				~
		ОК	Cance	el

Integration relative to position of

This parameter has two options:

- Center of gravity: This is the default option. With this option, the theoretical center of gravity is detected in each section. Integration is then referred to the center of gravity.
- Definition axes: This option refers to the definition axes. In cases where the position of definition axes differs from the position of the theoretical center of gravity, values of bending moments M_y and M_z are influenced by normal force acting on eccentricity.

Number of sections

This parameter defines on how many sections evenly spaced along the length of the integration member results will be integrated and presented. It is recommended to check the accuracy of the results. This means you might need to adjust the number of sections.

Selection of members for integration

This parameter has two options:

• All: With this option, internal forces are integrated from all members that are fully or at least partly inside the bounding box.

• User defined: This option allows you to manually define a set of members that will be excluded from the integration. In case this option is selected, additional action buttons "Exclude members from integration" appears in the properties. That may be used for the definition of a set of excluded members.

Remark: in case 1D or 2D members are only partly inside of the bounding box, the results are integrated only from the part of such members that are inside the bounding box.

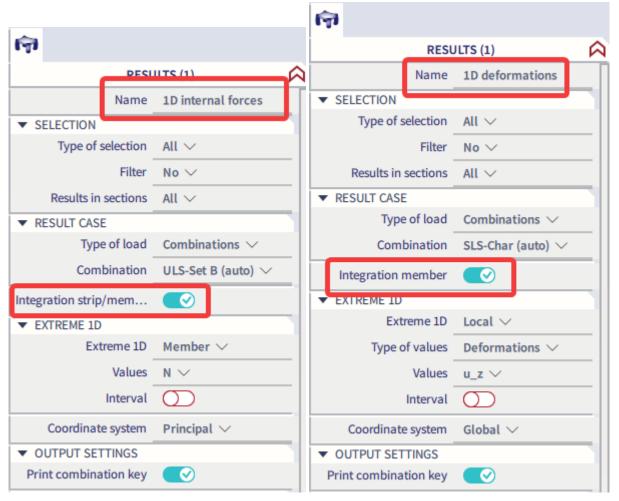
LCS

This property specifies the way the local axes of the integration member are determined. You can use LCS Rotation to define a rotation of the local axes of the integration member. The rotation is measured around the integration members longitudinal axis, i.e.X-axis.

Note that there are still some other parameters. These have a function when designing the reinforcement and will be discussed further on.

Results of the integration member

The results of the integration member are displayed as 1D member results. It is necessary to tick the checkbox "Integration strip/member" to see results on the integration member:



<u>1D internal forces</u>: Internal forces are integrated from all 1D and 2D members within the Bounding box. In case any object is only partly inside the Bounding box, only internal forces from that part which is inside the Bounding box are integrated.

<u>1D deflection</u>: For each section of the integration member, the average value of deflection of all members within the Bounding box is determined.

Design and check of an integration member

When modelling concrete structures, the integration member can be a useful tool for applications such as lintels, deep beams, beam slabs, core walls, etc. The modelling can be done as a 2D member, but by using the integration member you can get the results as a 1D member. On top of that you can design and check the reinforcement for these types of members.

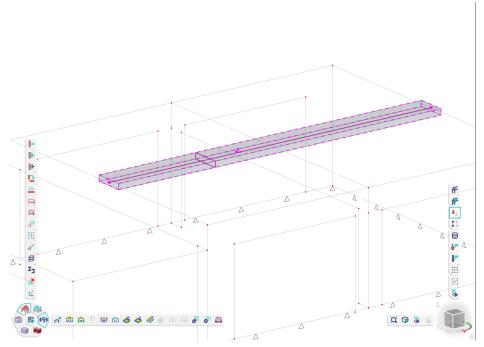
Settings of the integration member

It is necessary to define some extra properties that will be used for the design and check of the integration member:

- Group Buckling: This group contains properties which are necessary for the buckling calculation:
 - Beta y-y: definition buckling coefficient around the y-axis of the integration member. Default value is 1.0.
 - Beta z-z: definition buckling coefficient around the z-axis of the integration member. Default value is 1.0.
 - Sway y-y: definition if integration member is prone to sway (unbraced) around y-axis of the integration member. Default vaulue is loading from Setup according to type of material of integration member.
 - Sway z-z: definition if integration member is prone to sway (unbraced) around z-axis of the integration member. Default value is loading from Setup according to type of material of integration member.
- Member type: Definition of the type of integration member. Three options are supported:
 - o **Beam**
 - o Column
 - o Beam slab
- Cross-section: This property specifies which cross-section will be used for the design and check of integration members. The cross-section is presented graphically in the 3D window, if Draw cross-section is ON in View parameter setting.

Example

In this example we would like to design and check a beam slab. We start by inputting the integration member and setting the correct bounding box. We use a total width of 1m (0.5m left and 0.5m right) and a thickness of 0.2m (0.1m top and 0.1m bottom):

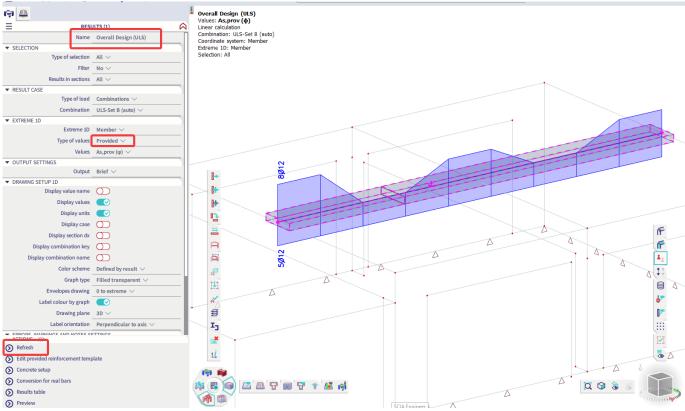


Next, we set the correct type of element, i.e. Beam slab, and choose the same cross-section. We also choose that the integration will be done according to the definition axes:

4	
INTEGRATION	MEMBER (1)
I]	
Name	IM1
Layer	Layer1 V
▼ BOUNDING BOX	
Shape	Rectangle 🗸
RECTANGULAR INTEGRATION SHAP	-
Left [m]	0,500
Right [m]	0,500
Bottom [m]	0,100
Top [m]	0,100
▼ INTEGRATION SETTINGS	
Integration relative to position of	Definition axes \checkmark
Number of sections	10
Selection of members for integration	All \checkmark
▼ CHECKS	
Member type	Beam slab \checkmark
Cross-section	CS4 - Rectangle (200; 1000) 🗸 📑
BUCKLING	
Beta y-y	1
Beta z-z	1
Sway y-y	
Sway z-z	
▼ GEOMETRY	
LCS	standard \checkmark
LCS Rotation [deg]	0,00
ACTIONS >>> Table edit geometry	

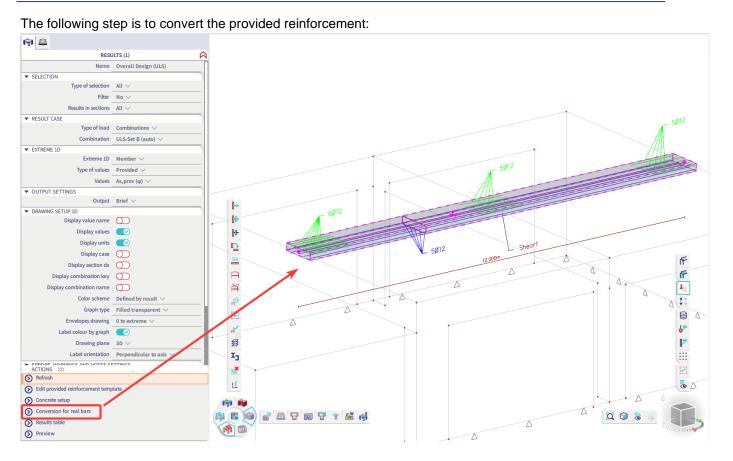
The next step is to check the 1D internal forces. If necessary, you can adjust the number of sections and the selected members for the integration.

Once the 1D internal forces are calculated, we can use these to design and check the reinforcement. Simply go to the 1D Reinforcement design and check the provided reinforcement:



Note that you can change the template of the provided reinforcement in the concrete setup according to the type of member:

Descrip	ntion	Symbol	Value	Default	Unit	Chapter	Code	Structure	CheckTy	
all>	۵ ۵	-	<all></all>	<all></all>	<	<all></all>	<all></all>		Design (X	
	n defaults	-un-	-ou-	-one /-		-our /-	-un-	oute 10	Designe X	1
	inforcement									
⊳	Beam / Rib									
	Beam slab									
	 Longitudinal 									
	Design of provided reinforcement						Independent	Beam slab	Design de	
	Rectangular section		BeamSlab	BeamSla			Independent	Beam slab	Design de	
	✓ Upper (z+)									
	Type of cover		Auto	Auto		4.4.1	EN 1992-1-1		Design de	
	Diameter	d _{s,u}	16	16	mm		EN 1992-1-1	Beam slab	Design de	
	Lower (z-)									
	Type of cover		Auto	Auto		4.4.1	EN 1992-1-1		Design de	
	Diameter	d _{s,I}	16	16	mm		EN 1992-1-1	Beam slab	Design de	
-	Detailing (det)									
	Column									
	Plate Wall / Deep beam									
	nimum cover									



Once the provided reinforcement is converted to practical reinforcement, you can use this reinforcement to perform the necessary concrete checks.