User's Manual



Finite Element Analysis & Design Program

Version 10

Inter-CAD Kft.

Copyright	Copyright © 1991-2011 Inter-CAD Kft. of Hungary. All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, for any purposes.			
Trademarks	AxisVM is a registered trademark of Inter-CAD Kft. All other trademarks are owned by their respective owners. Inter-CAD Kft. is not affiliated with INTERCAD PTY. Ltd. of Australia.			
Disclaimer	The material presented in this text is for illustrative and educational purposes only, and is not intended to be exhaustive or to apply to any particular engineering problem for design. While reasonable efforts had been made in the preparation of this text to assure its accuracy, Inter-CAD Kft. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.			
Changes	Inter-CAD Kft. reserves the right to revise and improve its product as it sees fit. This publication describes the state of this product at the time of its publication, and may not reflect the product at all times in the future.			
Version	THIS IS AN INTERNATIONAL VERSION OF THE PRODUCT THAT MAY NOT CONFORM TO CORRESPONDING STANDARDS IN A RESPECTIVE COUNTRY AND IS AVAILABLE SOLELY ON AN "AS IS" BASIS.			
Limited warranty	INTER-CAD KFT. MAKES NO WARRANTY, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, REGARDING THESE MATERIALS. IN NO EVENT SHALL INTER-CAD KFT. BE LIABLE TO ANYONE FOR SPECIAL, COLLATERAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH OR ARISING OUT OF PURCHASE OR USE OF THESE MATERIALS. THE SOLE AND EXCLUSIVE LIABILITY TO INTER-CAD KFT., REGARDLESS OF THE FORM OF ACTION, SHALL NOT EXCEED THE PURCHASE PRICE OF THE MATERIAL DESCRIBED HEREIN.			
Technical support and services	If you have questions about installing or using the AxisVM, check this User's Manual first - you will find answers to most of your questions here. If you need further assistance, please contact your software provider.			

CONTENTS

1. NEW FEATURES IN VERSION 10	9
2. HOW TO USE AXISVM	11
2.1. HARDWARE REQUIREMENTS	
2.2. INSTALLATION	
2.3. Getting Started	
2.4. AxisVM User Interface	
2.5. Using the Cursor, the Keyboard, the Mouse	
2.6. Hot Keys	
2.7. QUICK MENU	
2.8. DIALOG BOXES	
2.9. TABLE BROWSER	
2.10. REPORT MAKER	
2.10.1. Report	
2.10.2. Edit	
2.10.3. Drawings	
2.10.4. Gallery	
2.10.5. The Report Toolbar	
2.10.6. Gallery and Drawings Library Toolbars	
2.10.7. Text Editor	
2.11. Stories	
2.12. LAYER MANAGER	
2.13. DRAWINGS LIBRARY	
2.14. SAVE TO DRAWINGS LIBRARY	
2.15. THE ICON BAR	
2.15.1. Selection	
2.15.2. Zoom	
2.15.3. Views	
2.15.4. Workplanes	
2.15.5. Geometric tranformations on objects	
2.15.5.1. I ransiate	
2.15.5.2. Kotate	
$2.15.5.4 \qquad \text{Scale}$	
2.15.6 Display Mode	
2.15.0. Display Wode	
2.15.7. Guidelines	
2.15.9 Dimensions Lines Symbols and Labels	
2.15.9.1 Orthogonal Dimension Lines	47
2.15.9.2. Aligned Dimension Lines	50
2.15.9.3. Angle Dimension	50
2.15.9.4. Arc Length	
2.15.9.5. Arc Radius	
2.15.9.6. Level and Elevation Marks	
2.15.9.7. Text Box	
2.15.9.8. Object Info and Result Text Boxes	
2.15.9.9. Isoline labels	
2.15.10. Renaming/renumbering	
2.15.11. Parts	
2.15.12. Sections	
2.15.13. Find	
2.15.14. Display Options	

4	A	XL/VM/()
2 15 15 Or	ations	66
2.15.15.1.	Grid and Cursor	
2.15.15.2.	Editing	
2.15.15.3.	Drawing	
2.15.16. Mo	odel Info	69
2.16. Speed	d Buttons	
2.17. INFO	RMATION WINDOWS	71
2.17.1. Inf	fo Window	71
2.17.2. Co	oordinate Window	
2.17.3. Co	olor Legend Window	
2.17.4. Pe	rspective Window Tool	73
3. THE MA	IN MENU	75
3.1. File.		75
3.1.1. Ne	ew Model	75
3.1.2. Op	pen	76
3.1.3. Sav	ve	76
3.1.4. Sav	ve As	76
3.1.5. Ex	port	77
3.1.6. Im	port	
3.1.7. Te	kla Structures – AxisVM connection	
3.1.8. Pa	ge Header	
3.1.9. Pri	int Setup	
3.1.10. Pri	int	
3.1.11. Pri	inting from File	
3.1.12. Mo	odel Library	
3.1.13. Ma	aterial Library	
3.1.14. Cr	oss-Section Library	
3.1.14.1.	Cross-Section Editor	
3.1.15. Ex	it	
3.2. EDIT		
3.2.1. Ur	ndo	
3.2.2. Re	do	
3.2.3. Sel	lect All	
3.2.4. Co		
3.2.5. Pa	ste	
3.2.6. Co	ppy / paste options	101
3.2.7. De	Plete	
3.2.0. 1a	Die Drowser	
2.2.9. Ke	port Maker	
$3.2.10.$ $5a^{\circ}$	ving urawings and design result lables	103 102
3.2.11. We	somble structural mombars	
3.2.12. As	eak apart structural members	
3.2.13. Div	novert surface loads distributed over beams	104 104
3.2.14. CO	nivert suitace toaus distributed over bearing	104 104
3.2.15. CO	INCS	104 104
331 Di	enlav	104
332 Or	spiray	104 105
333 La	ver Manager	105
334 Sto	y er mannager innen i	
335 G	idelines	
3.3.6 De	rsign Codes	
3.3.7. Ur	nits and Formats	
3.3.8. Gr	avitation	108
339 Pr	eferences	100
3310 La	ng11age	
3.3.11 Re	port Language	
3.3.12. To	olbars to default position	
	1	

3.4.	VIEW	116
3.5.	WINDOW	
3.5.1.	Property Editor	
3.5.2.	Information Windows	
3.5.3.	Background picture	
3.5.4.	Split Horizontally	
3.5.5.	Split Vertically	
3.5.6.	Close Window	
3.5.7.	Drawings Library	
358	Save to Drawings Library	120 121
36	HFI P	122
361	Contents	122
362	AxisVM Home Page	122
363	AxisVM Update	122
364	About	122
365	Release information	122
37	MAIN TOOL BAR	
371	New	
372	Open	
373	Open Sava	
3.7.5.	Dave	
3.7.4. 2.7.5	T IIIt	123 122
3.7.5.	Pada	123 122
3.7.0.	Lavor Monagor	120 100
5.7.7. 279	Layer Mallager	123 124
<i>5.7.</i> 0.	Jiones	
<i>5.7.9</i> .	Table browser	
3.7.10	Report Maker	
3.7.11	Drawings Library	
3/1/	Save to Firawings Fibrary	1/4
0.7.12		
4. THE	PREPROCESSOR	
4. THE 4.1.	PREPROCESSOR	125
4. THE 4.1. 4.2.	PREPROCESSOR	125
4. THE 4.1. 4.2. 4.2.1.	PREPROCESSOR	125
4. THE 4.1. 4.2. 4.2.1. 4.3.	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS	125
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1.	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System	
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2.	PREPROCESSOR	
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4.	PREPROCESSOR	
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5.	PREPROCESSOR	
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6.	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW COORDINATE WINDOW GRID CURSOR STEP	125 125 126 126 126 127 127 127 127 128 128 128 128
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7.	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW GRID CURSOR STEP EDITING TOOLS	125 125 126 126 126 127 127 127 127 128 128 128 128 128 129
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1.	PREPROCESSOR	125 125 126 126 126 127 127 127 127 128 128 128 128 128 129 129
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2.	PREPROCESSOR	125 125 126 126 126 127 127 127 127 128 128 128 128 128 129 129 130
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3.	PREPROCESSOR	124 125 125 126 126 127 127 127 127 128 128 128 128 129 129 129 130 130
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4.	PREPROCESSOR	124 125 125 126 126 127 127 127 127 128 128 128 128 129 129 130 130 130
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5.	PREPROCESSOR	125 125 126 126 126 127 127 127 127 128 128 128 128 128 129 129 129 130 130 130
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5. 4.7.6.	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance Constrained Cursor Movements Freezing Coordinates. Auto Intersect	$\begin{array}{c} 124\\ 125\\ 125\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 132\\ 132\\ 132\\ 132\\ 132\\ 132\\ 132\\ 132$
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.3. 4.7.4. 4.7.5. 4.7.6.	Save to Drawings Elotaly PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance Constrained Cursor Movements Freezing Coordinates Auto Intersect GEOMETRY TOOL BAR	125 125 126 126 126 126 127 127 127 127 127 128 128 128 128 129 129 130 130 132 132 133
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5. 4.7.6. 4.8. 4.8. 4.81	Save to Drawings Elorary PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW. GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance Constrained Cursor Movements Freezing Coordinates Auto Intersect GEOMETRY TOOLBAR Node (Point)	$\begin{array}{c} 125\\ 125\\ 125\\ 126\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 132\\ 133\\ 133\\ 133\\ 133\\ 133\\ 133\\ 133$
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5. 4.7.6. 4.8. 4.8.1. 4.8.1.	Save to Drawing's Elorary PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR	$\begin{array}{c} 121\\ 125\\ 125\\ 126\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 132\\ 133\\ 133\\ 133\\ 133\\ 133\\ 133\\ 133$
4. THE 4.1. 4.2. 4.2.1. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5. 4.7.6. 4.8. 4.8.1. 4.8.2. 4.8.3	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance Constrained Cursor Movements Freezing Coordinates Auto Intersect GEOMETRY TOOLBAR Node (Point) Line Arc	$\begin{array}{c} 124\\ 125\\ 125\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 132\\ 133\\ 133\\ 133\\ 133\\ 134\\ 134\\ 134\\ 134$
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5. 4.7.6. 4.8. 4.8.1. 4.8.2. 4.8.3. 4.8.4	PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance Constrained Cursor Movements Freezing Coordinates Auto Intersect Auto Intersect Second Presson Cursor Identification Cursor Identification Cursor Identification Cursor Identification Cursor Identification Cursor Identification Cursor Identification Cursor Identification Cursor Identification Constrained Cursor Movements Freezing Coordinates Auto Intersect Auto Intersect Auto Intersect Horizontal Division	$\begin{array}{c} 124\\ 125\\ 125\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 130\\ 132\\ 133\\ 133\\ 133\\ 134\\ 135\end{array}$
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3.1. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.3. 4.7.3. 4.7.4. 4.7.5. 4.7.6. 4.8. 4.8.1. 4.8.2. 4.8.3. 4.8.4.	Save to Drawings Elonary PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW. GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance. Constrained Cursor Movements Freezing Coordinates. Auto Intersect GEOMETRY TOOLBAR. Node (Point) Line Arc Horizontal Division Vertical Division	$\begin{array}{c} 124\\ 125\\ 125\\ 126\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 132\\ 133\\ 133\\ 133\\ 133\\ 134\\ 135\\ 135\\ 135\\ 135\\ 135\\ 135\\ 135\\ 135$
4. THE 4.1. 4.2. 4.2.1. 4.3. 4.3. 4.3.1. 4.3.2. 4.4. 4.5. 4.6. 4.7. 4.7.1. 4.7.2. 4.7.3. 4.7.4. 4.7.5. 4.7.6. 4.8. 4.8.1. 4.8.2. 4.8.3. 4.8.4. 4.8.5. 4.8.6	Save to Drawings Library PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance Constrained Cursor Movements Freezing Coordinates Auto Intersect GEOMETRY TOOLBAR Node (Point) Line Arc Horizontal Division Vertical Division Ouad/Triangle Division	$\begin{array}{c} 124\\ 125\\ 125\\ 126\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 131\\ 133\\ 133\\ 133$
$\begin{array}{c} \textbf{4. THE} \\ \textbf{4.1.} \\ \textbf{4.2.} \\ \textbf{4.2.1.} \\ \textbf{4.3.} \\ \textbf{4.3.1.} \\ \textbf{4.3.2.} \\ \textbf{4.4.} \\ \textbf{4.5.} \\ \textbf{4.6.} \\ \textbf{4.7.} \\ \textbf{4.7.1.} \\ \textbf{4.7.2.} \\ \textbf{4.7.3.} \\ \textbf{4.7.4.} \\ \textbf{4.7.5.} \\ \textbf{4.7.6.} \\ \textbf{4.8.1.} \\ \textbf{4.8.2.} \\ \textbf{4.8.3.} \\ \textbf{4.8.4.} \\ \textbf{4.8.5.} \\ \textbf{4.8.6.} \\ \textbf{4.8.7} \end{array}$	Save to Drawings Library PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR	$\begin{array}{c} 125 \\ 125 \\ 125 \\ 126 \\ 126 \\ 126 \\ 127 \\ 127 \\ 127 \\ 127 \\ 128 \\ 128 \\ 128 \\ 128 \\ 129 \\ 129 \\ 129 \\ 130 \\ 130 \\ 130 \\ 130 \\ 130 \\ 131 \\ 133 \\ 133 \\ 133 \\ 133 \\ 134 \\ 135 \\ 135 \\ 135 \\ 136 \\ 127 \end{array}$
$\begin{array}{c} \textbf{4. THE} \\ \textbf{4.1.} \\ \textbf{4.2.} \\ \textbf{4.2.} \\ \textbf{4.2.1.} \\ \textbf{4.3.} \\ \textbf{4.3.1.} \\ \textbf{4.3.2.} \\ \textbf{4.4.} \\ \textbf{4.5.} \\ \textbf{4.6.} \\ \textbf{4.7.} \\ \textbf{4.7.5.} \\ \textbf{4.7.1.} \\ \textbf{4.7.2.} \\ \textbf{4.7.3.} \\ \textbf{4.7.4.} \\ \textbf{4.7.5.} \\ \textbf{4.7.6.} \\ \textbf{4.8.1.} \\ \textbf{4.8.2.} \\ \textbf{4.8.3.} \\ \textbf{4.8.4.} \\ \textbf{4.8.5.} \\ \textbf{4.8.6.} \\ \textbf{4.8.7.} \\ \textbf{4.8.8.} \end{array}$	PREPROCESSOR	$\begin{array}{c} 121\\ 125\\ 125\\ 126\\ 126\\ 126\\ 126\\ 127\\ 127\\ 127\\ 127\\ 127\\ 128\\ 128\\ 128\\ 128\\ 128\\ 128\\ 129\\ 129\\ 129\\ 130\\ 130\\ 130\\ 130\\ 130\\ 131\\ 133\\ 133$
$\begin{array}{c} \textbf{4. THE} \\ \textbf{4.1.} \\ \textbf{4.2.} \\ \textbf{4.2.} \\ \textbf{4.2.1.} \\ \textbf{4.3.} \\ \textbf{4.3.1.} \\ \textbf{4.3.2.} \\ \textbf{4.3.} \\ \textbf{4.3.2.} \\ \textbf{4.4.} \\ \textbf{4.5.} \\ \textbf{4.6.} \\ \textbf{4.7.} \\ \textbf{4.7.1.} \\ \textbf{4.7.2.} \\ \textbf{4.7.3.} \\ \textbf{4.7.4.} \\ \textbf{4.7.5.} \\ \textbf{4.7.6.} \\ \textbf{4.8.} \\ \textbf{4.8.1.} \\ \textbf{4.8.2.} \\ \textbf{4.8.3.} \\ \textbf{4.8.4.} \\ \textbf{4.8.5.} \\ \textbf{4.8.6.} \\ \textbf{4.8.7.} \\ \textbf{4.8.8.} \\ \textbf{4.8.9} \end{array}$	Surve to Drawings Library PREPROCESSOR GEOMETRY THE GEOMETRY EDITOR Multi-Window Mode COORDINATE SYSTEMS Cartesian Coordinate System Polar Coordinates COORDINATE WINDOW. GRID CURSOR STEP EDITING TOOLS Cursor Identification Entering Coordinates Numerically Measuring Distance. Constrained Cursor Movements Freezing Coordinates Auto Intersect GEOMETRY TOOLBAR Node (Point) Line Arc Horizontal Division Vertical Division Quad/Triangle Division Line Division Remove node	124 125 126 126 126 126 127 127 127 128 128 129 130 130 131 132 133 134 135 136 137 138 139

4.8.10.	Normal Transversal	
4.8.11.	Intersect plane with the model	
4.8.12.	Intersect plane with the model and remove half space	
4.8.13.	Domain Intersection	
4.8.14.	Geometry Check	
4.8.15.	Surface	
4.8.16.	Modify, transform	
4.8.17.	Delete	
4.9. F	INITE Elements	
4.9.1.	Material	
4.9.2.	Cross-Section	
4.9.3.	Direct drawing of objects	
4.9.4.	Domain	
4.9.4.1	COBIAX-domain	
4.9.5.	Hole	
4.9.6.	Domain operations	
497	Line Elements	149
498	Surface Elements	156
499	Nodal Support	150 159
4910	Line Support	162
4911	Surface Support	164 164
4912	Edge hinge	
4913	Rigid elements	
4914	Dianhragm	
4915	Spring	
4916	Can	
4917	Link	
4.9.17. 1018	Nodal DOF (Degrees of Freedom)	
4.9.10.	References	
4.9.19.	Creating model framework from an architectural model	
4.9.20.	Modify	
4.9.21.	Delete	180 180
4.7.22.		
4.10. L	Load Cases Load Croups	
4.10.1.	Load Combination	
4.10.2.	Nodal Loads	
4.10.5.	Concentrated Lead on Beam	
4 10 5	Point Load on Domain	
4.10.5.	Distributed line load on beam/rib	
4.10.0.	Edge Load	
4.10.7.	Domain Line Load	
4.10.0.	Surface Load	
4.10.9.	Domain Aroa Load	
4.10.10.	Curface load distributed even line along on to	
4.10.11.	Surface load distributed over line elements	
4.10.12.	Fluid Load	
4.10.13.	Deau Load	
4.10.14.	Fault In Length (Fabrication Error)	
4.10.15.	Thermol Lood on Line Flore onto	
4.10.10.	Thermal Load on Surface Florents	
4.10.17.	Inermai Loau on Surface Elements	
4.10.18.	Forced Support Displacement	
4.10.19.	finituence Line	
4.10.20.	Delsinic Loads	
4.10.2	0.1. Seismic calculation based on Eurocode δ	
4.10.2	0.2. Seismic calculation based on SWISS Code	
4.10.2	U.S. Seismic calculation based on German Code	
4.10.2	U.4. Seismic calculation based on Italian Code	

4.10.2	1. I ushovel loaus	
4.10.2	22. Tensioning	
4.10.2	23. Moving loads	
4.1	0.23.1. Moving loads on line elements	
4.1	0.23.2. Moving loads on domains	
4.10.2	24. Dynamic loads (for time-history analysis)	
4.10.2	25. Nodal Mass	
4.10.2	26. Modify	
4.10.2	27. Delete	
4.11.	Mesh	
4.11.1	. Mesh Generation	
4.1	1.1.1. Meshing of line elements	
4.1	1.1.2. Mesh generation on domain	
4.11.2	2. Mesh Refinement	
4.11.3	B. Checking finite elements	
5 ANI	AT VSIS	230
J. AN		······
5.1.	STATIC ANALYSIS	
5.2.	VIBRATION	
5.3.	DYNAMIC ANALYSIS	
5.4.	BUCKLING	
5.5.	FINITE ELEMENTS	
5.6.	MAIN STEPS OF AN ANALYSIS	
5.7.	ERROR MESSAGES	
6. THE	E POSTPROCESSOR	
61	CT ATTC	255
0.1.	Minimum and Maximum Values	
612	Animation	
0.1.2.	7 mmauon	
613	Diagram display	261
6.1.3. 6.1.4	Diagram display Pushover capacity curves	
6.1.3. 6.1.4. 6.1	Diagram display Pushover capacity curves	
6.1.3. 6.1.4. 6.1	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8	
6.1.3. 6.1.4. 6.1 6.1	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables	
6.1.3. 6.1.4. 6.1 6.1 6.1.5. 6.1.6	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements	
6.1.3. 6.1.4. 6.1 6.1 6.1.5. 6.1.6. 6.1.6.	Diagram display Pushover capacity curves .4.1. Capacity curves according to eurocode 8 .4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces	261 263 264 264 264 266 266 267 268
6.1.3. 6.1.4. 6.1 6.1 6.1.5. 6.1.6. 6.1.7. 6.1.8.	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces	261 263 264 264 264 266 267 268 270
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\end{array}$	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces Surface Elements Internal Forces	261 263 264 264 264 266 267 268 270 270
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\end{array}$	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces Surface Elements Internal Forces Surface Elements Internal Forces	261 263 264 264 264 266 267 268 268 270 270 270
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\end{array}$	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces Surface Elements Internal Forces Surface Elements Internal Forces Internal forces of line to line link elements and edge hinges	261 263 264 264 264 266 267 268 270 270 270 273 273
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\end{array}$	Diagram display Pushover capacity curves A.1. Capacity curves according to eurocode 8 A.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces Surface Elements Internal Forces Surface Element Internal Forces Internal forces of line to line link elements and edge hinges Truss/Beam/Rib Element Stresses	261 263 264 264 264 266 267 268 270 270 270 273 274 274
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.13\end{array}$	 Diagram display	261 263 264 264 264 266 267 268 270 270 270 273 274 274 274 274
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.13\\ 6.1.14\end{array}$	 Diagram display	261 263 264 264 264 266 267 268 270 270 270 270 273 274 274 274 274
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.14\\ 6.1.15\end{array}$	 Diagram display	261 263 264 264 264 266 267 268 270 270 270 270 273 274 274 274 276 276 277
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.2.\\ \end{array}$	 Diagram display	261 263 264 264 264 266 267 268 270 270 270 273 274 274 274 274 276 277 277
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ \end{array}$	 Diagram display	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ \end{array}$	 Diagram display	261 263 264 264 264 266 267 268 270 270 270 270 273 274 274 274 274 276 277 278 279 279 279
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ \end{array}$	 Diagram display	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\end{array}$	Diagram display Pushover capacity curves	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\\ 6.5.1.\\ 6.5.\end{array}$	 Diagram display	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\\ 6.5\\ 6.5.1.\\ 6.5\\ 6.5\end{array}$	Diagram display Pushover capacity curves	261 263 264 264 264 266 267 268 270 270 270 270 273 274 274 274 274 274 276 277 278 279 279 279 279 279 280 280 281
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.13\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\\ 6.5\\ 6.5.2.\\ \end{array}$	Diagram display Pushover capacity curves	261 263 264 264 264 266 267 268 270 270 270 273 274 274 274 274 276 276 277 278 279 279 279 279 279 279 279 279 279 280 281 281 283
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\\ 6.5.\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\end{array}$	Diagram display	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.\\ 6.5.1.\\ 6.5.\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5\\ 6.5\\ 6.5\\ 6.5\\ 6.5\\ 6.5\\ 6.$	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces Surface Elements Internal Forces Surface Elements Internal Forces Support Element Internal Forces Internal forces of line to line link elements and edge hinges Surface Element Stresses Surface Element Stresses Surface Element Stresses Influence Lines Surface Element Stresses Unbalanced Loads VIBRATION DYNAMIC Surface Reinforcement Surface Reinforcement Surface Reinforcement 1.1. Calculation based on Eurocode 2 1.2. Calculating based on DIN 1045-1 and SIA 262 Actual Reinforcement Actual Reinforcement 2.1. Reinforcement for surface elements and domains 2.2. Mesh-independent reinforcement	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.11\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.3.\\ \end{array}$	Diagram display Pushover capacity curves 4.1. Capacity curves according to eurocode 8 4.2. Acceleration-Displacement Response Spectrum (ADRS) Result Tables Displacements Truss/Beam Element Internal Forces Rib Element Internal Forces Surface Elements Internal Forces Surface Element Internal Forces Internal forces of line to line link elements and edge hinges Pruss/Beam/Rib Element Stresses Surface Element Stresses Surface Element Stresses Influence Lines Pruss/Beam/Rib Element Stresses Unbalanced Loads Pruss/Beam/Rib Element VIBRATION Pruss/Beam/Rib Element DYNAMIC Pruss/BuckLING R.C. DESIGN Surface Reinforcement 1.1. Calculation based on Eurocode 2 1.2. Calculating based on DIN 1045-1 and SIA 262 Actual Reinforcement 2.1. Reinforcement for surface elements and domains 2.2. Mesh-independent reinforcement 2.2.	
$\begin{array}{c} 6.1.3.\\ 6.1.4.\\ 6.1\\ 6.1.5.\\ 6.1.6.\\ 6.1.7.\\ 6.1.8.\\ 6.1.9.\\ 6.1.10\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.12\\ 6.1.13\\ 6.1.14\\ 6.1.15\\ 6.2.\\ 6.3.\\ 6.4.\\ 6.5.\\ 6.5.1.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.2.\\ 6.5\\ 6.5.3.\\ 6.5\\ 6.5.3.\\ 6.5\end{array}$	Diagram display Pushover capacity curves	261 263 264 264 264 266 267 268 270 270 270 273 274 274 274 274 274 274 276 277 278 279 279 279 279 279 279 279 280 280 281 281 283 284 284 285 284

AXIJVM1()

6.5.4	. Non-linear deflection of RC plates	
6.5.5	. Shear resistance calculation for plates and shells	
6.5	5.5.1. Calculation based on Eurocode 2	
6.5.6	. Column Reinforcement	
6.5	5.6.1. Check of reinforced columns based on Eurocode 2	
6.5	5.6.2. Check of reinforced columns based on DIN1045-1	
6.5	5.6.3. Check of reinforced columns based on SIA 262	
6.5.7	. Beam reinforcement design	
6.5	5.7.1. Beam Reinforcement Design based on Eurocode2	
6.5	5.7.2. Beam Reinforcement Design based on DIN 1045-1	
6.5	5.7.3. Beam Reinforcement Design based on SIA 262:2003	
6.5.8	. Punching Analysis	
6.5	5.8.1. Punching analysis based on Eurocode2	
6.5	5.8.2. Punching analysis based on DIN 1045-1	
6.5.9	. Footing design	
6.5.1	0. Design of COBIAX slabs	
6.6.	Steel Design	
6.6.1	. Steel beam design based on Eurocode 3	
6.6.2	. Bolted Joint Design of Steel Beams	
6.7.	TIMBER BEAM DESIGN	
7. AX	SVM VIEWER AND VIEWER EXPERT	
8. PRO	DGRAMMING AXISVM	
0 STF	P BV STEP INPUT SCHEMES	3/10
<i>.</i>		
9.1.	PLANE TRUSS MODEL	
9.2.	PLANE FRAME MODEL	
9.3.	PLATE MODEL	
9.4.	MEMBRANE MODEL	
9.5.	KESPONSE SPECTRUM ANALYSIS	
10. EXA	AMPLES	
10.1.	LINEAR STATIC ANALYSIS OF A STEEL PLANE FRAME	
10.2.	GEOMETRIC NONLINEAR STATIC ANALYSIS OF A STEEL PLANE FRAME	
10.3.	BUCKLING ANALYSIS OF A STEEL PLANE FRAME	
10.4.	VIBRATION ANALYSIS (I-ORDER) OF A STEEL PLANE FRAME	
10.5.	VIBRATION ANALYSIS (II-ORDER) OF A STEEL PLANE FRAME	
10.6.	LINEAR STATIC ANALYSIS OF A REINFORCED CONCRETE CANTILEVER	
10.7.	LINEAR STATIC ANALYSIS OF A SIMPLY SUPPORTED REINFORCED CONCRETE PLATE	
10.8.		0
	LINEAR STATIC ANALYSIS OF A CLAMPED KEINFORCED CONCRETE PLATE	
11 REI	LINEAR STATIC ANALYSIS OF A CLAMPED REINFORCED CONCRETE PLATE	

1. New features in Version 10

General

	New display style to help users with high resolution monitors	
	Architectural rendering	2.15.6 Display Mode
	Exporting SDNF file	2.1.5 Europe
	Export of parts or selected elements to AXS file	3.1.5 Export
	<i>New DXF import options (import of visible layers, creating parts using layer information)</i>	3.1.6 Import
	Automatically updated logical parts	2.15.11 Parts
	Renaming / renumbering elements	2.15.10 Renaming/renumbering
	Definition of stories	3.3.4 Stories
	IFC enhancements (improved processing of BREP and IFCB uilding Element Proxy)	3.1.6 Import
Ed	iting	
	Removal of intersections	4.8.9 Remove node
	Editing on stories	3.3.4 Stories
	Detachment of objects	4.8.16 Modify, transform
	Cutting multiple domains	4.8.11 Intersect plane with the model
	Cutting objects with a plane	4.8.12 Intersect plane with the model and remove half space
	New editing functions on pet palettes (detach, cutoff, tangential arc)	4.8.16 Modify, transform
	New constraints (point of intersection for two lines, dividing point betweeen to nodes)	2.15.8 Geometry Tools
	Structural copy & paste functions (customizable through Edit / Copy/paste options)	3.2.6 Copy / paste options

Structural copy & paste functions (customizable through Edit / Copy/paste options)3New functions in the COM server

Elements

Timber database with material parameters according to Eurocode5	6.7 Timber Beam Design
Rib definition with automatic eccentricity update	4.9.7 Line Elements
Nonlinear link elements (tension only / compression only)	4.9.17 Link

Loads

 Smart labeling of line loads
 A

 Polygonal or arced line loads
 A

 Polygonal, arced or complex polygonal surface loads
 A

 Edge loads can be defined on internal lines of a domain
 A

 Optimization of surface loads distributed over beams and ribs for multiple core processors
 A

Analysis

Pushover Analysis according to EC8

Analysis information can be reviewed any time using the Model information dialog New analisys engine optimized for multiple cores / threads can reach more memory than before Dynamic analysis **(module DYN)**

Increment function editor for nonlinear analysis

Results

Design of COBIAX slabs

Display of average support forces on line supports

Display of elastic hinges at beam ends New result tables (beam, rib, truss forces for different load cases)

Improved diagram display

Design

Cross-sections of Class 4 can be designed

Pad footing design according to Eurocode 7 calculating footing size and reinforcement (module RC4) Timber design according to Eurocode 5 (module TD1) 4.10.6 Distributed line load on beam/rib4.10.8 Domain Line Load4.10.10 Domain Area Load4.10.7 Edge Load

4.10.21 Pushover loads 6.1.4 Pushover capacity curves 2.15.16 Model Info 5 Analysis

5.3 Dynamic Analysis5.1 Static Analysis

6.5.10 Design of COBIAX slabs
6.1.10 Support Element Internal Forces
4.9.7 Line Elements
6.1.7 Truss/Beam Element Internal Forces
6.1.8 Rib Element Internal Forces
6.1.3 Diagram display

6.6.1 Steel beam designbased on Eurocode 36.5.9 Footing design

6.7 Timber Beam Design

2. How to Use AxisVM

Welcome to AxisVM!

AxisVM is a finite-element program for the static, vibration, and buckling analysis of structures. It was developed by and especially for civil engineers. AxisVM combines powerful analysis capabilities with an easy to use graphical user interface.

Preprocessing Modeling: geometry tools (point, lines, surfaces); automatic meshing; material and crosssection libraries; element and load tools, import/export CAD geometry (DXF); interface to architectural design software products like Graphisoft's ArchiCAD via IFC to create model framework directly.

At every step of the modeling process, you will receive graphical verification of your progress. Multi-level undo/redo command and on-line help is available.

Analysis Static, vibration, and buckling

Postprocessing Displaying the results: deformed/undeformed shape display; diagram, and iso-line/surface plots; animation; customizable tabular reports.

After your analysis, AxisVM provides powerful visualization tools that let you quickly interpret your results, and numerical tools to search, report, and perform further calculations using those results. The results can be used to display the deformed or animated shape of your geometry or the isoline/surface plots. AxisVM can linearly combine or envelope the results.

Documentation Documentation is always part of the analysis, and a graphical user interface enhances the process and simplifies the effort. AxisVM provides direct, high quality printing of both text and graphics data to document your model and results. In addition data and graphics can be easily exported (DXF, BMP, JPG, WMF, EMF, RTF, HTML, TXT, DBF).

2.1. Hardware Requirements

The table below shows the minimum/recommended hardware and software requirements, so you can experience maximum productivity with AxisVM.

Recommended con- figuration	at least 1 GB RAM at least 2 GB of free hard disk space CD drive XGA color monitor (at least 1024x768, 1280x1024 recommended) Windows 2000 / XP/ Vista / Windows 7 operating system Mouse or other pointing device Windows compatible laser or inkjet printer

Memory access To reach more memory is very important as it speeds up the analysis considerably. To enable advanced memory access is possible under Professional or Ultimate editions of Windows Vista and Windows 7 operating systems. Home Premium edition does not support this feature

If the computer has more than 4 GB of physical RAM, AxisVM10 can access memory over 4 GB on 32-bit operating systems.

To turn this function ot it is necessary to lock pages in memory:

After invoking the *Run* command from the *Start* menu type gpedit.msc. After clicking the OK button a Windows application named *Group Policy* opens. Find the following item in the tree on the left: *Computer Configuration / Windows Settings / Security Settings / Local Policies / User Rights Assignment*. Then find *Lock pages in memory* in the list on the right. Double click on this item. In the *Local Policy Sertings* dialog click the *Add* button then add the users or user groups who needs access to the memory above 4 GB. Close *Local Policy Settings* dialog then close *Group Policy* by clicking the Close icon in the top right corner.

User Account Control must also be disabled.

<u>Under Vista</u>: Launch MSCONFIG from the *Run* menu. Find and click *Disable UAC* on the *Tools* tab. Close the command window when the command is done. Close MSCONFIG and restart the computer.

<u>Under Windows 7</u>: Find *Start Menu / Control Panel / User Accounts*. Click on *Change User Account Control settings* link. Set the slider tothe lowest value (*Never Notify*). Click OK to make the change effective and restart the computer.

2.2. Installation

Software Protection	The program is protected by a hardware key. Two types of key are available: parallel port (LPT) keys and USB keys.		
(g-)	Plug the key only after installation is complete, because certain operating systems try to recognize the plugged device and this process may interfere with the driver installation.		
	Non-network drivers will be automatically installed. If you encountered problems you can install this driver later from the CD. Run the Startup program and select Reinstall driver .		
Standard Key	First install the program then plug the key into the computer.		
Network Keys	If you have a network version you must install the network key. In most cases AxisVM and the key are on different computers but to make the key available through the network the Sentinel driver must be installed on <i>both</i> computers.		

AxisVM Version 10 is shipped with a parallel port or USB Sentinel Super Pro dongle but earlier customers may have parallel port NetSentinel dongle.

a. Sentinel SuperPro dongle

- 1. Insert the AxisVM CD in the CD-ROM drive of the AxisVM server. Run [CD Drive]: \ Startup.exe. Select **Reinstall driver**. This type of network key requires at least a 7.1 driver. CD contains the 7.5 version of the driver.
- 2. Connect the key to the parallel or USB port of one of the computers. This way you select the AxisVM server.

To run AxisVM on any computer on the network SuperPro Server must be running on the server. If it stops all running AxisVM programs stop.

b. NetSentinel dongle

- 1. Insert the AxisVM CD in the CD-ROM drive of the AxisVM server. Run [CD Drive]: \ Sentinel \ English \ Driver\ setup.exe to install Sentinel driver.
- 2. Connect the key to the parallel port of one of the computers. This way you select the AxisVM server.
- 3. Copy the contents of the folder [CD Drive]: \ Sentinel \ English \ server \ Disk1 \ Win32 to a folder of the server's hard drive.
- 4. Run **NSRVGX.EXE** from that folder. This server program handles the network key and communicates with the applications on the network.
- To run AxisVM on any computer on the network NSRVGX must be running on the server. If NSRVGX stops all running AxisVM programs stop.

AxisVM runs on 2000 / XP / Vista / Windows 7 operating systems.

Insert the AxisVM CD into the CD drive. The Startup program starts automatically if the autoplay option is enabled. If Autoplay is not enabled, click the Start button, and select *Run*.... Open the Startup.exe program on your AxisVM CD. Select **AxisVM 10 Setup** and follow the instructions.

If the setup program cannot be launched or the following message appears: AUTOEXEC.NT -The system file is not suitable for running MS-DOS and Microsoft Windows applications, a Windows system file must be missing.

Installation under Vista Operating System:

- You need the latest Sentinel driver. You can download it: www.axisvm.eu / Support- Service Pack for AxisVM 10
- Click on the program icon with the Mouse right button after the installation of AxisVM program
- Choose the Properties menu item from the Quick Menu.
- Select the Compatibility tab on the appearing dialog and turn on the Run as administrator checkbox.

By default the program and the example models will be installed on drive C: in

C:\Program Files\AxisVM10 and C:\Program Files\AxisVM10\Examples

folders. You can specify the drive and the folders during the installation process. The setup program creates the AxisVM program group that includes the AxisVM application icon.

Installation

œ.

Starting AxisVM



Click the Start button, select Programs, AxisVM folder, and click the AxisVM10 icon. At startup a splash screen is displayed (**see...** 3.6.4 About) then a welcome screen is shown where you can select a previous model or start a new one.

Clearing the checkbox at the bottom turns the welcome screen off for the future. To turn it on choose the *Settings\Preferences\Data Integrity* dialog and check the *Show welcome screen on strartup* checkbox.

New Model			×
Select a view to	start with		
۲	Eolder	C:\Axis\hiba\	- 🗳
L→× Top View	<u>M</u> odel Filename:	Model 2	
₹ I II	Design Code	Eurocode 🗾	\bigcirc
Front View	Units and Formats	EU Units 💌	Change Settings
₹	<u>R</u> eport Language	English	
Perspective	— Page Header ———		
	Project		
	Analysis by Inter-CAD Kft		
	<u>C</u> omment An examp	ole comment line	
	Project Analysis by Inter-CAD An example comment Model: Model 2.axs	Kft. line	
		[OK Cancel

Upgrading It is recommended to install the new version to a new folder. This way the previous version will remain available.

Converting earlier Models created in a previous versions are recognized and converted automatically. Saving files will use the latest format by default. Saving files in the file format of one of the previous versions (6, 7, 8, 9) is possible but this way the information specific to the newer versions will be lost.

Steps of an analysis The main steps of an analysis using AxisVM are:

Creating the Model (Preprocessing)			
	\checkmark	/	
	Anal	ysis	
Static	Vibration	Dynamic	Buckling
(linear/nonlinear) (first/second-order) (linear/nonlinear)			
\checkmark			
Evaluating the Results (Postprocessing)			

Capacity

Practically, the model size is limited by the amount of free space on your hard disk. The restrictions on the model size and on the parameters of an analysis are as follows:

Entity		Maximum
Nodes		Unlimited
Materials		Unlimited
Elements	Truss	Unlimited
	Beam	Unlimited
	Rib	Unlimited
	Membrane	Unlimited
	Plate	Unlimited
	Shell	Unlimited
	Support	Unlimited
	Gap	Unlimited
	Diaphragm	Unlimited
	Spring	Unlimited
	Rigid	Unlimited
	Link	Unlimited
Load cases		Unlimited
Load combinations		Unlimited
Frequencies		Unlimited

Professional

Small Business

Entity	Maximum	
Nodes		Unlimited
Materials		Unlimited
Elements	Only trusses	500
	Truss+Beam+Rib *	250
	Rib on the edge of a surface	1000
	Any combination of	1500
membrane, plate or shell		
	Support	Unlimited
	Gap	Unlimited
	Diaphragm	Unlimited
	Spring	Unlimited
Rigid		Unlimited
	Link	Unlimited
Load cases		99
Load combinations		Unlimited
Frequencies (modal shapes)		30

* If there are beams or/and ribs in the structure

2.3. Getting Started

	Step-by-step input schemes are presented in the Section 9 . See Example 1 of Chapter 10 with a step-by-step input scheme in 9.2 Plane Frame Model
	There are three major steps in a modeling process:
Geometry	The first step is to create the geometry model of the structure (in 2D or 3D). Geometry can be drawn by hand or can be imported from other CAD programs. It is also possible to draw elements (columns, beams, walls, slabs) directly.
Elements	If you chose to draw the geometry first you must specify material and element properties, mesh the geometry into elements (assigning the properties and a mesh, to the wire-frame model), and define the support conditions.

Loads

16

In the third step you must apply different loads on the model.

The end result will be a finite element model of the structure.

Once the model is created it is ready for analysis.

In Chapter 7, the step-by-step modeling of a few typical structures are presented.

The following types of structures are shown:

- 1. Plane truss girder
- 2. Plane frame
- 3. Plate structure
- 4. Membrane cantilever
- 5. Seismic analysis

Understanding of these simple models will allow you to easily build more complex models. It is recommended that you read the entire User's Manual at least once while exploring AxisVM.

In Chapter 1 you can find the timely, new features of the version.

Chapter 2 contains general information about using AxisVM. In other chapters the explanation follows the pre- and postprocessor menu structures. Please consult this User's Manual every time you are using AxisVM.

2.4. AxisVM User Interface

This section describes the working environment of the full AxisVM graphical user interface. Please read these instructions carefully. Your knowledge of the program increases the modeling speed and productivity.

AxisVM screen

After you start AxisVM a screen similar to the following picture appears:



	The parts of the AxisVM screen are briefly described below.
Graphics area	The area on the screen where you create your model.
Graphics cursor	The screen cursor is used to draw, select entities, and pick from menus and dialog boxes. Depending on the current state of AxisVM, it can appear as a pick-box, crosshairs with pickbox, or pointer.
Top menu bar	Each item of the top menu bar has its own dropdown menu list. To use the top menu bar, move the cursor up to the menu bar. The cursor will change to a pointer. To select a menu bar item, move the pointer over it, and press the pick button to select the item. Its associated sub-menu will appear.
Active icon	The active icon represents the command that is currently selected.
Icon bar	The icons represent working tools in a pictorial form. These tools are accessible during any stage of work. The icon bar and flyout toolbars are draggable and dockable.
Coordinate	The window on the graphics area displaying the graphics cursor coordinates.
Color legend window	The window shows the color legend used in the display of the results. Appears only in the post-processing session.
Info window	The window shows the status of the model and results display.
Context sensitive help	Provides a help message that depends on the topic under process.
Property Editor	The Property Editor offers a simple way to change certain properties of the selected elements or loads.
Pet palette	Pet palettes appear when modifying geometry according to the type of the dragged entity (node, straight line, arc). See 4.8.16 Modify, transform
Speed buttons	Speed buttons in the bottom right provide the fastest access to certain switches (parts, sections, symbols, numbering, workplanes, etc.)
The model	With AxisVM you can create and analyze finite element models of civil engineering structures. Thus the program operates on a model that is an approximate of the actual structure.
	To each model you must assign a name. That name will be used as a file name when it is saved. You may assign only names that are valid Windows file names. The model consists of all data that you specify using AxisVM. The model's data are stored in two files: the input data in the filename . axs and the results in the filename . axe file.
	AvisVM charles if AVS and AVE files belong to the same version of the model

AxisVM checks if AXS and AXE files belong to the same version of the model.

2.5. Using the Cursor, the Keyboard, the Mouse

Graphics cursor



As you move your mouse, the graphics cursor symbol tracks the movement on the screen. To select an entity, an icon or menu item, move the cursor over it and click the left mouse button. The shape of the cursor will change accordingly (**see...** 4.7.1 Cursor Identification), and will appear on the screen in one of the following forms:



If you pick an entity when the cursor is in its default mode (info mode), the properties of that entity will be displayed as a tool tip.

	Geometry	node (point) coordinates, line length
	Elements	finite element, reference, degree-of-freedom, support
	Loads	element load, nodal mass
	Mesh	meshing parameters
	Static	displacement, internal force, stress, reinforcement, influence line ordinate
	Vibration	mode shape ordinate
	Dynamic	displacement, velocity, acceleration, internal force, stress
	R.C. Design	specific reinforcement values
	Steel Design	efficiency results and resistances
	Timber Design	utilization factor results and resistances
The keyboard	You can also use the	keyboard to move the cursor:
Arrow keys, 🐣	Moves the graphics of	cursor in the current plane.
[Ctrl] + Arrow keys, 个	Moves the graphics of set in the Settings dia	cursor in the current plane with a step size enlarged/reduced by a factor alog box.
+[Shift]) [√](→],	Moves the graphics $\alpha + n.90^{\circ}$.	cursor in the current plane on a line of angle $n{\cdot}\Delta\alpha$, custom αor
[Home] [End]	Moves the graphics of	cursor perpendicular to the current plane.
[Ctrl]+ [Home], [End]	Moves the graphic enlarged/reduced by	es cursor perpendicular to the current plane with a step size a factor set in the Settings dialog box.
[Esc] or ${\frown}$	Interrupts the comm	and and/or returns to an upper menu level.
[Enter]+[Space]	Selects an item from These are termed cor	a menu, executes a command, and selects entities. nmand buttons.
[Alt]	Activates the main m	lenu
[Tab]	Moves the focus from	n control to control in a dialog.
[+] [-]	Performs fast zoom current position of th set in <i>Settings / Optic</i> graphics cursor posit	in/out and pan. The zoom and pan parameters are defined by the ne graphics cursor in the graphics area, and by the magnification factor <i>ins / Zoom Factor</i> . Center of the fast zoom in/out is always the current ion.
[Insert] or [Alt]+[Shift]	Moves the relative o graphics cursor posit	rigin (i.e. the reference point of the relative coordinates) to the current ion.
ᠿ wheel	Roll forward to zoom Roll backwards to zo Press the wheel and Centre of zoom in an	n in om out drag to drag the drawing area ad zoom out is the current position of the cursor.
Hot Keys	Keyboard combination See 2.6 Hot Keys	ons to access frequently used functions faster.
A right button	Displays the Quick M	/lenu. See 2.7 Quick Menu

Depending on the menu your cursor is on, you may get the properties of the following entities:

2.6. Hot Keys

General Hot Keys

[Ctrl]+[W]	Zoom to fit
[Ctrl]+ [1]	X-Z view
[Ctrl]+ [2]	X-Y view
[Ctrl]+ [3]	Y-Z view
[Ctrl]+ [4]	Perspective view
[Ctrl]+ [P]	Print
[Ctrl]+ [A]	Select All (adds all entities to the selection list)
[Ctrl]+ [[]	View undo
[Ctrl]+ []]	View redo
[Ctrl]+[Z]	Undo
[Shift]+[Ctrl]+ [Z]	Redo
[Tab]	Move between graphics windows
[Ctrl]+[R]	Refresh drawing (redraw)
[Ctrl]+ [Q]	Exit
[Ctrl]+ [C]	Copy (to clipboard)
[Ctrl]+[V]	Paste (from clipboard)

Hot Keys in Tables

-	
[Ctrl]+[L]	Browse Libraries
[Alt]+[F4]	Exit
[Ctrl]+[Insert]	New line
[Ctrl]+[Del]	Delete line
[Ctrl]+[A]	Select all
[F5]	Jump to line
[Ctrl]+[D]	Default format
[Ctrl]+ [Alt]+[F]	Set column format
[Ctrl]+[R]	Set result display mode (for result tables)
[Ctrl]+[G]	Edit new cross-section (for cross-section tables)
[Ctrl]+[M]	Modify cross-section (for cross-section tables)
[F1]	Context sensitive help

- **[F9]** Add table to the report
- [F10] Report Maker

Hot keys in the Report Maker

[Ctrl]+[T]	Insert text
[Ctrl]+ [Alt]+[B]	Insert Page Break
[Ctrl]+[W]	Export to RTF file
[F3]	Report Preview
[Ctrl]+[P]	Print
[Ctrl]+[Del]	Delete

[Alt]	Go to main menu
[+]	Zoom in
[-]	Zoom out
[Ctrl]+ [O]	Open
[Ctrl]+ [S]	Save
[Del]	Delete entities/properties
[Ctrl]+ [D]	Switches
[Ctrl]+[L]	Labels
[Ctrl]+[Y]	Symbols
[Ctrl]+[E]	Reverse local x direction of line elements
[F1]	Context-sensitive help
[F7]	Set stories
[F8]	Weight Report
[F9]	Save drawing to the Drawings Library
[F10]	Report Maker
[F11]	Layer Manager

[F12] Table Browser

2.7. Quick Menu

🕆 right button

utton When the cursor is over the graphics area, by pressing the right mouse button a **quick menu** appears in accord with the current command in use.



2.8. Dialog Boxes

After selecting a function usually a dialog box appears on the screen. These dialog boxes can be used the same way as any other Windows dialog.

The dialog font can be changed by selecting the *Settings\Preferences\Fonts* dialog and clicking the font sample label *Dialog boxes*.

You can change the position of all dialog windows. The program saves the latest position and displays the dialog on the same position next time.

2.9. Table Browser

[F12]

AxisVM uses tables to display numerical information on the screen allowing changes in formatting. The tables operate in the same way independent of the content displayed. All the tables AxisVM creates are available through the Table Browser dialog box by clicking its button or pressing [F12].

The model data to be displayed in the Table Browser can be selected from the tree structure in the left side of the browser. If you use Table Browser while working in the pre-processor, input model data is displayed only. While working in the post-processor, the model results are also displayed.

Only the data of the current selection (if any) or of the active (i.e. displayed) part is listed by default.

The tree view on the left lists element / load data, result tables and libraries in a hierarchy and can also be used as a model overview.



Using the table A table can contain more rows and/or columns than can be displayed at the same time. It can be viewed in its entirety using the scroll bars and/or using the keyboard as follows: Arrow keys Moves the edit focus up and down, to the left and to the right, and scrolls the table along [✓][†] left button the rows or columns. Clicking an editable cell moves the edit focus to that cell. [Home] Moves the focus to the first cell of the row. Moves the focus to the last cell of the row. [End] [Ctrl]+[Home] Moves the focus to the first cell of the first row [Ctrl]+[End] Moves the focus to the last cell of the last row. [Page Up] Displays the previous page of rows. [Page Down] Displays the next page of rows. Moves the focus to the next (to the right) page of columns (only in tables where more [Ctrl]+ [→] columns can be displayed at the same time). Moves the focus to the previous (to the left) page of columns (only in tables where more [Ctrl]+ [←] columns can be displayed at the same time). [Enter] Ends the current editing in the edit box storing the data entered and moves the edit box a column to the right or to the first column of the next row. [Esc] Aborts the current editing in the edit box. \checkmark right button [Shift] While the [Shift] key is down all direction keys will select cells instead of moving the edit focus. You can also select cells by dragging the mouse. Clicking a fixed (topmost) cell of a column selects the column. Clicking a fixed (leftmost) cell of a row selects the row. Clicking the top left cell selects the entire table. Selected cells can be copied to clipboard as a table. If selection is within an editable column you can set a common value for the selected cells.

See... Set Common Value below



Browse Library

Loads cross-sectional or material data from a library. You can also save the current content of the table in a custom library.

Import DBase File

.....

use File Imports a DBase file *name.dbf* into the current table. The program checks the values of the fields and sends an error message if an incompatible value is found.

Save As DBaseExports the current table into a Dbase file name.dbf. The field names are generated based onFilethe names of the columns. The fields will be of text type.

Save As HTML Exports the current table into an HTML file *name.htm*. This file can be imported as a table into Word or can be opened in web browser applications. Some formatting information of the columns will be lost.



Save As RTF Exports the current table into an RTF file *name.rtf* using the current template file. You can import this file into Microsoft Word or any other word processor which can import RTF files. **See...** 2.10.1 Report

New Cross- Creates a new cross-section data file *name.sec*. The table created will be placed together with *Section Table* the cross-sections of the same type.

You can store cross sections of any type in these tables. Type of the table determines only the position of the table in the Cross-section Library.

Cross- You can modify properties (table's name, cross.section type) of a user defined table.

SectionTable Properties

Delete Cross You can delete a user defined table. *Sectin Table*

Print

Prints all the information displayed in the table to the selected printer or to a file, with the page header and comment row previously set with the *File/Header* menu command.



Exits the table in the same way as the Cancel button (the changes are not saved).

Edit

Eile	<u>E</u> dit	Format <u>R</u> eport <u>H</u> elp	
	+	New Row	Ctrl+Ins
	\times	Delete Rows	Ctrl+Del
		<u>S</u> elect Table	Ctrl+A
	Ē	Design New Cross-Section	Ctrl+G
	宜	Modify Cross-Section	Ctrl+M
	✓	Automatic cross-section shape	update
		Delete unused cross-sections	
	Ð	⊆ору	Ctrl+C
		Paste	Ctrl+V
		Set Common Value	
		<u>G</u> o To	F5



Adds a new row to the list, and allows you to fill all the editable cells with data in a fixed order from left to right.

Delete Rows



[Ctrl]+ [Del]

Delete textures

Available only if materials are listed. Removes texture from the selected materials. Available in the popup menu.

Deletes the selected rows. Also available in the popup menu.





Selects the entire table. Clicking the top left cell does the same.

Design New Custom Crosssection



Modify Custom Cross-section



Automatic crosssection shape update

If this function is on changing section parameters in the table leads to the recalculation of geometry and cross-section parameters.

Starts the graphics Cross-Section Editor, allowing the modification of a custom cross-section

Starts the graphics Cross-Section Editor, allowing the input of a new custom cross-section.

Delete unused cross-sections Unused cross-sections will be deleted from the table.

previously created with the graphics Cross-Section Editor.



Copies selected cells to the Clipboard as a table. Also available in the popup menu.



Pastes table cells from the Clipboard overwriting cell values.
If any of the values is unacceptable Paste aborts.
V] If entire rows were cut or copied and the table allows inserting new rows you can also add clipboard data to the end of the table instead of overwriting the existing rows.

Set Common Value

Sets a common value for the selected cells within a column.
Example: you can set the Z coordinate of all nodes to the same value making the model absolutely flat. Available from the *Table Browser Menu / Edit / Set Common Value*.
Also available in the popup menu.

Go to Jumps to a specified row in the table.

[F5]

Format	Format Report Help
	Iurn on/off columns Ctrl+Alt+F
During model	Restore Default Format Ctrl+D Order of load cases
building	✓ Intermediate sections
	Show used cross-sections in boldface
Turn on/off columns	You can specify whether a column is visible or not, by setting the check boxes of the corresponding columns.
	The display format is set according to the settings in the Units/Settings dialogue window (See 3.3.7 Units and Formats).
[Ctrij+ [Aitj+ [F]	Many cells require the entry of a numeric value. When entering real numbers you can use the following characters:
	+ - 0 1 2 3 4 5 6 7 8 9 0 E
	and the standard Windows decimal separator specified in <i>Start / Settings / Control Panel / Regional Settings / Number / Decimal symbol</i> field.
	In some cases you cannot enter a negative number so the - key is deactivated while entering these kind of values. If an integer value is required you cannot use the decimal separator and E.
Format Defaults [Ctrl]+ [D]	Restores the default format of the entire table (column visibility and decimals).
Order of load	The display order of load cases can be customized.
cases	See 4.10.1 Load Cases, Load Groups
Intermadiate sections	After dividing or meshing beams or ribs with variable cross-section AxisVM builds up in- termediate cross-sections. This menu item is to turn on/off the display of intermediate cross- sections at the end of the list.
	Table Browser



Show used crosssections in boldface

	Name	Draw.	Process	Shape	h [cm]	b [cm]	tw [cm]	tf [cm]
3	IPE 360	·- <u>‡</u>	Rolled	I	36,0	17,0	0,8	1,3
4	IPE 80	- <u>Ť</u>	Rolled	I	8,0	4,6	0,4	0,5
17	IPE 180	•- <u>‡</u>	Rolled	I	18,0	9,1	0,5	0,8

After the *Delete unused cross-sections* command only the sections in bold will remain in the list.

The cross-names which are signed by bold letter will remain in the table if the *Delete Unused Cross-sections* switch is turned on.

In case of result query new items appear on the Format menu and the Toolbar.

Eile	Edit	For	nat	<u>R</u> eport	Help		
		\square	<u>T</u> ur	n on/off	columns	. Cti	rl+Alt+F
			Res	store Def	ault Form	at	Ctrl+D
			<u>O</u> re	Order of load cases		· N	
			Res	Result Display Options		s K	Ctrl+R
		~	Re	Results			Ctrl+T
		✓	Ext	Extremes		Ctrl+E	
			Pro	perty Filt	ering		Ctrl+Q

During result query Result Display Options

You can control finding the extremes for result components and set to show results (Result) and/or just the extremes (Extremes). See in detail: 6.1.5 Result Tables [Ctrl]+[R]

Results On/Off [Ctrl]+[T] Extremes On/Off

[Ctrl]+[E]

Display of results can be turned on / off.

Display of extremes can be turned on / off.

Property Filtering [CTRL]+[Q]	Property Filtering Materials C25/30 Cross-Section 40x40 D40	Range C Entire Model O Displayed Parts Image: Selection Image: Selection <
	7 matches found	Filtering by surface thickness OK OK

Property filtering helps you to select which elements to include in the table.

Report

t	<u>R</u> ep	ort <u>H</u> elp	
		⊆urrent Report	•
	1	Add table to report	F9
	1	Report Maker 🗟	F10

Current report You can set the current report. Tables will be added to this report. See... 2.10 Report Maker

Add table to Adds the current table to the current report. If the selected node in the treeview has subreport nodes (e.g. MODEL or Loads) all tables under that node will be added. If the current table is a result table and is set to display extremes only all sub-tables will display extremes only. See... 2.10 Report Maker.

8 M -[F9]

Report Maker Opens Report Maker.

<u>File Edit Fo</u>rma

[F10]

Help

<u>File Edit Format Report Help</u> Help On Current Table How To Use Table Browser

Displays info about the table. About Table

About Table

Displays info about the table browser operation.

Closes the table without saving the data.

Browser

OK Saves the data and closes the table.

Cancel

Ŧ Result tables also display the extremes (minimum and maximum values) of the data if you select this option in the Display Options dialog when you enter Table Browser. Displaying both the individual values and the extremes is the default setting.

2.10. Report Maker

訂 [F10] Report Maker is a tool to compile a full report of a project using report items (tables/drawings/pictures created by AxisVM and user-defined text blocks). Reports are stored in the model file (*.axs) and can be printed or saved as a Rich Text Format (RTF) file. RTF files can be processed by other programs (e.g. Microsoft Word).

Tables ex ported from Table Browser are automatically updated if the model has been changed or some of its parts were deleted.

Report Maker can handle several different reports for the same project. The structure of reports is displayed in a tree view on the left. The properties of the selected report item are shown on the right side of the window.



Table

Text

If a table is selected, its comment text, column titles and other properties are shown. Display of title, comment and columns can be turned on and off.

If a text block is selected the text is shown on the right. Click the button Edit text... to make changes.

Picture or Drawing

If a picture or drawing is selected it is shown on the right. Its size, alignment and caption can be set.



Drawings Library

By clicking the Drawings Library tab you can browse the saved drawings and add the selected ones to the report. Unlike the pictures in the Gallery these drawings are not graphics files, but view settings stored to recreate the drawing at any time. This way drawings will be automatically updated if we change and recalculate the model. **See in detail...** 3.5.7 Drawings Library, 3.5.8 Save to Drawings Library.

Gallery

By clicking the Gallery tab you can browse the saved pictures (BMP, JPG, WMF, EMF) located in a folder named *Images_modelname* and add the selected ones to the report. This folder is automatically created as a subfolder of the model folder.

See in detail... 2.10.4 Gallery

Fitture size in the report Eft to page Vidth [mm] Height [mm]	198,1 mm 💌	Center Normal Color	-
⊂ Scale ⓒ Fit in Window	C 1: 100	ок	Cancel

Click the *Settings*... button to change the caption, size, justification, rotation color mode or scaling of drawings.

You can save the current drawing on screen or the result tables in design modules with the function of *Edit\ Saving drawings and design result tables* in main menu. **See...** 3.2.10 Saving drawings and design result tables

One or more selected pictures in the Gallery can be inserted into a report by selecting menu item *Gallery/Add pictures to the report* or clicking the arrow button above the Gallery or by drag and drop.

In printed reports Report Maker automatically builds a table of contents and inserts it to the beginning of the report. Tables are listed according to their titles. Text blocks are listed only if they were formatted using one of the Heading styles in the Text Editor. Pictures are listed only if they have a caption.

2.10.1. Report

<u>R</u> ep	ort	Edit	Drawings	; <u>G</u> allery
氜	New report			
\times	Delete entire report			t
	<u>R</u> er	name.		
	<u>S</u> a۱	/e As `	тхт	
恊	E×p	oort as	RTF	Ctrl+W
	RŢ	= Opti	ons	
10	Rep	ort p	review	F3
9	Prir	nt		Ctrl+P
	E <u>x</u> i	t		



Creates a new report. Report names can be 32 characters long.

Delete entire report

Deletes the current report (i.e. the report which contains the selected item). Pictures used in the report are not deleted from Gallery.

[Del], [Ctrl]+[Del]

Rename Gives a new name to an existing report.

Save As TXT Exports the report into a ASCII text file. Drawings or pictures are not included.

Export as RTF 뛥

Saves the report as *name.rtf* using the current template. If you save the file to a folder different from the model folder all picture files used in the report are copied to an automatically created subfolder Images_modelname. It is necessary because pictures are only linked and not saved into the RTF document. To print the RTF report on a different machine make sure that picture files are also copied to a subfolder *Images modelname*.

Character and paragraph formatting of text blocks will be exported. The only exception is the character color. Tables will be exported as RTF tables. Table titles are formatted with Heading 3 style so it is easy to build a table of contents automatically using Microsoft Word. In Insert / Index and Tables or Insert / Reference / Index and Tables select the Table Of Contents tab of the dialog, set Formats to From template and Show levels to at least 3.

RTF Options AxisVM saves reports to RTF files using a template (the default one is *Template.rtf* in the program folder). You can use other templates as well. When changing a template you can create your own cover sheet and header/footer for the re-port. Read the text of the template file carefully before changing it.

Tables Cyridlines 1234 54 10 1234 54 20 1234 54 20 1234 54 40 1344 54 50	Format of drawings in RTF file C Embedded VMF C Link to BMP files C Link to JPG files

Format of drawings in RTF file can also be set.:

Embedded WMF: Drawings are embedded into the file. It improves portability but can result in huge file size.

Link to BMP, JPG: This option keeps the RTF file smaller as drawings are stored in external files. Drawings appear only if pictures are located in an *Images modelname* subfolder relative to the folder of the RTF file.

Gridlines of exported tables can also be turned on/off.

Displays a print preview dialog. You can set the zoom factor between 10% and 500% Report preview (Page Width and Full Page is also an option). Click the buttons or use the keyboard to move [F3] backward and forward between pages ([Home] = first page, [PgUp] = previous page, **[PgDown]** = next page, **[End]** = last page.



Q

Print A dialog to set printing parameters and print a report. The options are the same as the table printing options.

Exit Quits the Report Maker.

Rep

2.10.2. Edit

ort	Edit	Drawings Gallery		
	кŋ	<u>U</u> ndo		
	$\bigcirc I$	Redo		
	÷*	Report Builder	2	
		Insert folder	v	
		Insert text into report	Ctrl+T	
	+	P <u>ag</u> e break	Ctrl+Alt+B	
	Ŧ	Move up selected report item		
	÷	Move down selected report	item	
		Mo <u>v</u> e to	•	
		⊆opy to	•	
		Selection filter		
		Select subitems automatical	ly	
		Deselec <u>t</u> all		
		Select all items of the curre	nt report	
	×	D <u>e</u> lete	Ctrl+Del	
	×	Delete all report items		

Some of the functions in the *Edit menu* are also available in the popup menu after clicking right mouse button on a report item.

- *Undo* Undoes the effect of the previous command.
- *Redo* Executes the command which was undone.

Report Builder Report Builder creates complete structured reports based on several filter options set on the *Filter* tab. Load cases, result components, parts, element and load types can be selected and set the display of extremes or results in the tables

set the display of extremes or results in the tables. The rules of creating reports can be set on the Preferences tab. You can choose if you want to see different element types listed within a part or different parts listed within an element

to see different element types listed within a part or different parts listed within an element type, or if you want to see result components listed within a load case or load cases listed within a result component.

If we imported an architectural model it is also possible to filter for architectural objects and ask for separate tables for each architectural object.

The number of expanded levels (1-7) of the report tree on the right can be set with the leveladjustment bar.

The tree on the right side shows the report built using the criteria set in the left. Each report item can be turned on/off individually. The report sent to the report maker will contain the checked items only.



Filter lists only the user-defined parts. Logical parts do not appear in the list.

2	\boldsymbol{n}
- >	
-	~

Insert folder	Inserts a new folder into the tree, below the current item. The current folder name appears on the right side under the folder icon. The number of expanded levels (1-7) of the report tree can be set with the level-adjustment bar.
Insert text into report	Starts a built-in Text Editor to create a new text block. The formatted text will be inserted after the selected report item.
[Ctrl]+[T]	
Page break	Inserts a page break after the selected report item.
[Ctrl]+[Alt]+[B]	
Move up/down selected report item	Moves up/down the selected report item by one.
Move to / Copy to	Moves / copies the selected report item to the end of another report.
Selection filter	Determines which types of report items can be selected (report, table, drawing, picture, text, page break, folder).
Select subitems automatically	If you turn this checkbox on and select a folder all subitems will be selected automatically.
Deselect all	Deselects all selected items in the documentation.
Select all items of the current report	Every report item of the current report will be selected.
Delete X [Del], [Ctrl]+[Del]	Deletes the selected report item (text block, picture, table, page break). If the current selection in the tree is a report it deletes the entire report.
Delete all report items	Deletes all items from the current report but does not delete the report itself.

2.10.3. Drawings



Add drawings to
the reportInserts the selected drawing(s) from the Drawings Library into the selected report. Place of
insertion is determined by the selected item of the report tree. Effect of this function is the
same as that of the the button on the Drawings Library tab.

Format of See... 2.10.1 Report drawings in RTF file...

2.10.4. Gallery



Add pictures to the report	Inserts selected pictures into the current report.
Copy pictures to Gallery	You can copy bitmaps (.BMP, .JPG) and Windows Metafiles (.WMF, .EMF) to the folder Images_modelname.
Delete pictures from Gallery	Deletes selected pictures from the Gallery. Files are permanently deleted.
Delete unused pictures	Deletes pictures which are not used in the reports.
Sort by name / type / date	Gallery sorts pictures by filename / by type (.BMP, .EMF, .JPG, .WMF) or by date.
Reverse order	If checked pictures are sorted in descending order. Otherwise pictures are sorted in ascending order.

2.10.5. The Report Toolbar





2.10.6. Gallery and Drawings Library Toolbars

You can perform certain tasks faster using these small toolbars.





Inserts selected pictures or drawings into the current report. Place of the insertation is determined by the selected item in the report tree.

Copies pictures from other locations to the Gallery. This function is not available on the Drawings Library tab.

2.10.7. Text Editor

After selecting *Insert text to report* a formatted text can be created in a simple WordPad-like text processor.

File

Open The main purpose of this function is to load a Rich Text file written in Text Editor. If you open an RTF file created in another word processor it may contain special commands (e.g. tables, paragraph borders, Unicode characters) which are not supported this simple editor. As a result you may get a series of rtf control commands instead of formatted text.

Save Saves the text into an RTF file.

[Ctrl]+[S]

Exit	Quits Text Editor.
Edit	
Undo / Redo [Alt]+[BkSp] / [Shift]+[Alt]+[BkSp]	Undoes / redoes the last editing action.
Cut [Ctrl]+[X]	Cuts the selected text and places it to the Clipboard.
Copy [Ctrl]+[C]	Copies the selected text to the Clipboard.
Paste [Ctrl]+[V]	Pastes the content of the Clipboard at the current position.
Find [Ctrl]+[F]	You can search for any text in the document. You can search from the beginning or from the current position. You can search whole words only and turn on and off case sensitivity.
Find next [F3]	If a match was found you can get the next match with this function.
Select all [Ctrl]+[A]	Selects the entire text.

l Iser's	Manual
	manuar

Character	
Bold	Applies bold formatting to the selected text.
[Ctrl]+[B]	
Italic	Applies italic formatting to the selected text.
[Ctrl]+[l]	
Underline	Applies underline formatting to the selected text.
[Ctrl]+[U]	
Color	Sets the character color of the selection.
[Ctrl]+[Alt]+[C]	
Paragraph	
Left justify	Justifies the selected paragraphs to the left.
[Ctrl]+[L]	
Centered	Justifies the selected paragraphs to the centerline.
[Ctrl]+[E]	
Right justify	Justifies the selected paragraphs to the right.
[Ctrl]+[R]	
Bullet [Ctrl]+[Alt]+[U]	Places bullets before the selected paragraphs.

2.11. Stories



Ð

See in detail... 3.3.4 Stories

2.12. Layer Manager

See in detail... 3.3.3 Layer Manager

2.13. Drawings Library



喝

See in detail... 3.5.7 Drawings Library

2.14. Save to Drawings Library

See in detail... 3.5.8 Save to Drawings Library.

2.15. The Icon bar



If you choose Workplanes, Dimensioning - Model info a dialog will appear.

Dragging and docking the Icon bar and the flyout toolbars

The left-side icon bar and any flyout toolbar can be dragged and docked.

Dragging and docking of the Icon bar

If you move the mouse over the handle of the Icon bar (on its top edge), the cursor will change its shape (moving). You can drag the Icon bar to any position on the screen. If you drag the Icon bar out of the working area through its top or bottom edge the Icon bar becomes horiozontal. If you drag it to the left or right edge it becomes vertical.

If the Icon bar is horizontal you can dock it at the top or at the bottom. You can change the position and the order of docked toolbars by dragging. In the Cross-Section Editor and in Beam and Coumn Reinforcement dialogs the Icon bar cannot be docked. Closing a floating Icon bar restores its original position docked on the left.

Dragging and docking of flyout toolbars

You can also separate flyout toolbars from the Icon bar by dragging their handle. Closing or dragging them back to the Icon bar restores their original position. Floating flyout toolbars can be docked at the top or at the bottom.

The Icon bar and the flyout toolbars can be restored to their original position by selecting Settings\Toolbars to default position from the menu

2

2.15.1. Selection

Activates the selection mode and displays the selection icon's bar.



Lets you select a set of entities (nodes (points), lines, finite elements and loads) for processing. When you execute commands you can use the Selection icon to specify the entity set to which to apply the command to. If the Parts check box (See section 2.15.11 Parts) is enabled the selection will refer only to the active (visible) parts.

You can change the view settings or continue selection in another window pane during the selection process. These allow you to select elements in the most convenient view. The selected entities are displayed in magenta in the graphics area.

The selection process is considered finished when the **OK** button is pressed.

Selection methods with selection frame:

of various selection shapes are provided:

- dragging the selection frame from left to right selects elements entirely within the frame
- dragging the selection frame from right to left selects elements which are not entirely outside the frame

Select	Adds the currently selected entities to the set of selected entities.
Deselect	Removes the currently selected entities from the set of selected entities.
Invert	Inverts the currently selected entities' selection status.
All	Applies the current selection mode (add, remove, or invert) to all filtered entities.
Previous	Restores the previous selection set.
Selection of parts	Clicking the button and a part from the list will select elements of the chosen part.
Filter	Lets you specify filtering criteria to be used during selection. Check element types to select. Property filtering lets you apply further criteria (beam length, cross-section, material, surface thickness, reference).
Method	Selects entities using different methods (selection shapes). Rectangular, skewed rectangular, sectorial or ring selection shapes are available. In the followings examples of the application



- OK Ends the selection, retaining the selected set for use.
- Cancel Ends the selection, discarding the selected set.
 - æ If an entity is hidden by another entity you cannot select it by simply clicking on it. In such a case, you have to change view to select it.
 - The selected nodes are marked with a surrounding magenta rectangle. Sometimes it is G. necessary to double-select nodes. In this case these nodes are marked with an additional blue rectangle surrounding them.

Selections can also be made, without using the Selection Icon Bar. Pressing and holding the [Shift] button while selecting with the 🖑 will add entities to the selection and pressing and holding the **[Ctrl]** button while selecting with the $^{\circ}$ will remove entities from the selection. Double selections can be made by pressing and holding the [Alt] button while double clicking on the entities with the $^{-}$.
[•] During the selection we can modify the apperiance of the structure, we can switch over an other view or perspective observation.

2.15.2. Zoom



Displays the zoom icon bar.



Zoom in

(+)

Displays an area of the model drawing specified by two points (two opposite corners) on the graphics area defining a rectangular zoom region. As a result, the apparent size of the model displayed in the graphics area increases.

area specified by two points (two opposite corners) defining a rectangular zoom region. As a result, the apparent size of

Zoom out Displays the model drawing from the graphics area on the

the model displayed in the graphics area decreases.



Zoom to fit Scales the drawing of the model to fit the graphics area, so you can view the entire model.



Moves the drawing. Press and hold the left button of the *A* while moving the mouse, until the desired position of the drawing is obtained on the screen.

Quick Drag:

You can use the mid mouse button to drag the model drawing at any time (without the the *Pan* icon).

- 1. Click the Pan icon.
- 2. Drag the model to its new position.



 Φ This cursor shape indicates that you can pan the model.



After clicking this icon you can rotate the model around the centre of the encapsulating block of the model by dragging. During the rotation the following pet palette appears at the lower part of the screen:



Rotation methods in the order of icons:

Free rotation around the horizontal axis of the screen and the global Z axis.

Rotation around the global Z axis.

Rotation around the vertical axis of the screen.

Rotation around the horizontal axis of the screen.

Rotation around an axis perpendicular to the screen.

This cursor shape indicates that you can rotate the model.

Undoes / redoes the action of up to 50 view commands.



Undo view



Sets the parameters of the perspective display. The proper view can be set by rotating the model drawing around the three axes, and by setting the observation distance. Rotation angles can be set with a precision of 0.1 degrees. You can assign a name to each setting that you want to save for later use. Type a name into the combo and click on the icon on the left of the combo to save the settings. To delete a perspective setting choose it from the dropdown list and click on the Delete icon on the right side of the combo. Palette settings are stored.

- *Observation* Observation distance is the distance between the viewpoint and the centre of the *distance* encapsulating block of the model.
 - Rotation

After clicking on the rotate icon a pet palette appears as described earlier (Zoom*Rotate*).



Views, perspective

낝

Displays three projection views and the perspective view of the model, and allows you select the view that you want to display. Click the view you want to select.





2.15.4. Workplanes

Workplanes (user coordinate systems) makes it easier to draw on oblique planes. Consider a hole for a skylight on an oblique plane of a roof. The plane of the roof can act as a workplane so drawing can be performed in two dimensions. In case of workplanes altitudinal coordinate means the distance along the axis normal to the workplane.

All drawing/editing functions are available in workplane mode. Using multi-window mode a different workplane can be set for each window.

Global X-Y, These workplanes are parallel with a global *Global X-Z*, coordinate plane so their position is defined by a *Global Y-Z* single coordinate. Useful when drawing stories of a *workplanes* building.

General These workplanes are defined by an origin and two *workplanes* vectors for the local *x* and *y* axes.

- *Smart workplanes* These workplanes follow the local system of a truss, beam, rib or domain. The origin is the first point of the element, local *x* and *y* axes are parallel to the local *x* and *y* axes of the local system of the element.
 - Changing the local system of the finite element the workplane is also changing. Deleting the finite element you delete the workplane as well.

Clicking the workplane speed button the workplane can be selected from a list. Workplanes are also available from the main menu by selecting *View* \ *Workplanes* or from the popup menu by selecting *Workplanes*.

Workplanes	×	
Global model space	Global X-Y	
() Workplane_1 (Y = 6,901)		
Ø Workplane_2 (X = 3,634)	Glob <u>a</u> l Y-Z	
i⊟ General workplane	General	
	S <u>m</u> art	
	Delete	
Parameters		
Iype General workplane		
Origin(m): 6,391 8,049 Local ×(m): 1,000 -0,024 Local y(m): -0,027 -0,953	17,850 0,015 0,300	
Display Olobal (model space) Image: Constraint of the space of the s		
Show elements out of workplane grayed		
☐ <u>R</u> efresh All OK	Cancel	

Clicking the workplane speed button the workplane can be selected from a list. Workplanes are also available from the main menu by selecting *View* \ *Workplanes* or from the popup menu by selecting *Workplanes*.

Display options A workplane can be displayed in the global coordinate system or in its local system. After checking *Hide elements not in the workplane* only those elements are displayed that are in the workplane. After checking *Show elements out of workplane grayed* elements out of the workplane appears grayed.

Changing If you select a workplane from the tree, its parameters are displayed. Editing them and *workplane* clicking the **OK** button or selecting another workplane will change the parameters of the selected workplane.

Delete Deletes user defined workplanes.

Pick Up >> Lets you define workplane parameters (origin or axes) graphically.

2.15.5. Geometric tranformations on objects



2.15.5.1. Translate

Translate

Makes multiple copies of, or moves the selected geometric entities or loads, by translation along a vector. You must specify the translation vector (dX, dY, dZ), and the number of copies (N).

Translation options **Incremental:** makes N copies of the selected entities by the distance dX, dY, dZ.

Distribute: makes N copies of the selected entities along the distance dX, dY, dZ (by dX/N, dY/N, dZ/N increments).

Spread by distance: makes copies of the selected entities spread by distance *d* in the direction of the translation vector.



The number of copies depends on how many copies will fit into the length defined by the translation vector dX, dY, dZ.

Consecutive: makes N consecutive copies of the selected entities by different distances dX, dY, dZ.

Move: moves the selected entities by the distance dX, dY, dZ. Lines running into the moved nodes remain connected.

Detach: moves the selected entities by the distance dX, dY, dZ. Lines running into the moved nodes are detached.

None: No nodes will be connected.

Double selected: Holding the **[Alt]** key pressed you can double select nodes. These nodes will be connected.

All: All nodes to be copied will be connected.

Switches

Copy options **Copy elements:** You can specify the finite elements assigned to the geometric entities to be copied as well.

Copy loads: You can specify the loads assigned to the geometric entities to be copied as well.

Coads can be copied separately (without the elements).

Copy nodal masses: You can specify the nodal masses to the geometric entities to be copied as well.

Copy dimension lines: The dimension lines will be copied only if the nodes to which they are assigned are selected.

With guidelines All rulers will also be moved (useful when moving the entire model).

With DXF layer With this option checked the transformations will be performed on the objects of the DXF layer as well.

Visible layers only With this option checked only the visible layers will be transformed.

Steps of translating

^{ng} The translation consists of the following steps:

- 1. Click on the Translate icon
- 2. Select the entities or loads to be copied
- 3. Click **OK** on the Selection Window (or Cancel to interrupt the selection and translation commands)
- 4. Select your options from within the Translate Window.
- 5. Click **OK**
- 6. Specify the translation vector by its start and end point

The command can be applied in the 2-3-1-4-5-6 sequence as well.

^{There} If you have repetitive parts in your model, you should first create these (including the definition of finite elements, support conditions, loads, and dimension lines), and then make copies of them.

You can use any existing point when you have to specify the translation vector. Selected loads can be copied or moved to another load case if load case is changed to the target load case during the operation.

2.15.5.2. Rotate

Rotation

Makes multiple copies of, or moves the selected geometric entities or loads, by rotation around a center. In X-Y, X-Z or Y-Z views the rotation axis is normal to the current view plane. In perspective view rotation axis is always the Z axis.



additional translation h along the rotation axis (each copy will be shifted by this distance). Click the rotation center (OX, OY, OZ), the rotation arc start point and draw the cursor

You can specify the method of rotation.

Parameters depend on the method: rotation

angle α , the number of copies (N) and an

h [m] = 0 Rotation Nodes to connect (None Incremental Double selected Distribute O <u>A</u>I Spread by angle Consecutive C Move Copy elements O Detach Copy loads 🔽 Copy noda<u>l</u> masses Vith quidelines 🔽 Copγ dimension symbols Vith DXF Layer Visible layers only OK Cancel

Rotation options

angle.

Incremental: makes N copies of the selected entities by the cursor angle.

Distribute: makes N copies of the selected entities by cursor angle/N increments.

Spread by angle: makes copies of the selected entities spread by a given angle α specified in the dialog. The number of copies depends on how many copies will fit into the cursor angle.

Consecutive: makes N consecutive copies of the selected entities at different cursor angles.

- **Move:** moves the selected entities by the cursor angle. Lines running into the moved nodes remain connected.
- **Detach:** moves the selected entities by the cursor angle. Lines running into the moved nodes are detached.

x

Nodes to connect	See2.15.5.1 Translate
Switches	See21551 Translate

See...2.15.5.1 Translate

In perspective view, the centerpoint, start point and endpoint can be specified only using existing points or other identified 3D locations (i.e. a point on a line). In perspective view, cursor angle is determined by the global X and Y coordinates only.

2.15.5.3. Mirror

Switches

See... 2.15.5.1 Translate

Mirror	Makes a copy of, or moves the selected geometric entities or loads, by mirroring. Specify two points of the symmetry plane. The symmetry plane is always parallel to a global axis depending on what view you are in.		Stranger Carrier
Mirror options	Copy: reflects a copy of the selected entities over the mirror plane.	Mirror	X
	 Multiple: makes consecutive copies of the selected entities over different mirror planes. Move: moves the selected entities across the mirror plane. Lines running into the moved nodes remain connected. Detach: moves the selected entities across the mirror plane. Lines running into the moved nodes are deatched. 	Mirror Cogy Multiple Moye Detach Vith guidelines Vith DXF Layer Visible layers only	Iodes to connect ○ None ○ Double selected ○ All IV Copy elements IV Copy oldad IV Copy nodal masses IV Copy dimension symbols OK
Nodes to connect	See 2.15.5.1 Translate		
Switches	See 2.15.5.1 Translate		
	In perspective view , the mirroring is possibl Z axis.	le only across a plane	e parallel with the global

2.15.5.4. Scale Scale Makes multiple copies of, or moves the selected geometric entities, by scaling from a center. You must specify the scaling center, a point of reference and its new position after scaling X (coordinate ratios will determine the scaling factors). Scale options Incremental: makes N scaled copies of the Scale × selected entities by repeating the scaling N N = 1 times. Distribute: distributes N scaled copies of the selected entities between the original and the scaled image. Scale -Nodes to connect C Incremental C Distribute C Consecutive **Consecutive**: makes differently scaled copies © Non of the selected entities in consecutive steps. Consecutive • Resize Resize: redefines the selected entities by 🔽 Copy elements $\overline{\nabla}$ scaling. 🔽 Copy nodaį masses 🔲 With guidelines Copy dimension symbols 🔲 🗥 th DXF Layer ок Cancel Nodes to connect See... 2.15.5.1 Translate

2.15.6. Display Mode



Wireframe:

666

Displays a wireframe model drawing. In this mode the axis of the line elements and the mid-plane of the surface elements are displayed.



Displays a wireframe model drawing with the hidden lines removed.



Rendered:

Displays a rendered model drawing. The line elements are displayed with their actual cross-section and the surface elements with their actual thickness.

The elements colors are displayed corresponding to colors assigned to their materials. Rendered view is smoother and shows the details of thin-walled cross-sections.

Transparency In *View / Rendering options...* transparency of element types can be set. Element types are determined by geometry. Vertical line elements are considered to be columns, horizontal ones are handled as beams, horizontal domains as floors, vertical domains as walls.

-Transparency	Cpaque Tra	ansparent
Columns Beams Other line elements Beams with tendons	} }	0 0 0 0 50
vVall Slab Other surface elements	} }	
○ Rendering schematic model Color ✓ Show tendons		
🔽 Auto Refresh	ок	Cancel



Opaque





111

Rendering type Two rendering types are available:

- Schematic model

Turning on *Show tendons* a more realistic picture of tensioned beams is drawn. Tendon color can also be set here.

- Architectural model

Eh.

Instead of drawing the structural framework this rendering mode intersects connections getting closer to the final look of the model.

Render bolted joints in detail turns on detailed rendering of designed bolted joints. *Draw object edges* turns on/off object edges.





Schematic model

Architectural model

Texture. A rendered view using textures assigned to individual materials. Textures can be assigned to materials by clicking the *Texture* field in the table of materials or in the material database and choosing a texture from the library of textures. It contains predefined textures and let the user define custom textures as well. If more than one row is selected in the table texture will be applied to all selected materials.





Branches of the tree view on the left and the horizontal list above the texture thumbnails show the material types (brick, concrete, metal, stone, timber, other). The last type (custom) is for the user-defined textures. Textures of the current type are displayed as thumbnails. The selected texture appears in the preview window with a thick black frame.

Popup menu	After clicking the texture with the right mouse button a popup menu appears with the following functions:		No texture Add custom texture
	Removing the texture from the material Defining or deleting a custom texture Rotation settings	•	Delete custom texture No rotation Rotate left Rotate right
No texture	Removes the texture from the current material		
Add custom texture	24-bit True Color bitmaps (JPG or BMP) can be converted to textures of 64×64 , 128×128 or 256×256 pixels. If the bitmap was not rectangular it will be cropped into a rectangle.		
Delete custom texture	Predefined textures cannot be deleted from the library, only the assigment can be removed. User-defined textures in the <i>Custom</i> category can be deleted.		
Rotation settings	Textures are mapped to the elements according to their local coordinate it can lead to undesirable results (e.g. in case of brick walls). Texture rota problems without changing the local system of elements. By default tated. The other two options are <i>Rotate left</i> and <i>Rotate right</i> rotating the local tion is indicated in the table by a < or > character appearing at the end of	e sy itio tex oitr of t	rstem. Sometimes on can solve these tures are not ro- nap by 90°. Rota- he texture name.

2.15.7. Guidelines

Helps in editing the geometry of the model. Guidelines can be defined in the global coordinate system. This way an arbitrary grid can be created, intersections can be determined and distances can be set. The cursor identifies the guidelines. **See...** 4.7 Editing Tools

Ger The guidelines are displayed as blue dashed lines. The display of the guidelines can be enabled or disabled in the Display Options menu (or icon) in the Switches section.



- Places a vertical guideline at the current position of the cursor.
- Places a horizontal guideline at the current position of the cursor.



Places a vertical and a horizontal guideline at the current position of the cursor.



Places an oblique guideline at the current position of the cursor.

X

Places a pair of orthogonal oblique guidelines at the current position of the cursor.

In perspective view all the guidelines are displayed but only oblique guidelines can be placed. You can change the position of a guideline with the mouse by dragging it to a new position. You can remove (delete) a guideline by dragging it off the graphics area.

Guidelines can be entered numerically by coordinates. Clicking with the mouse on a guideline or selecting *Settings/Guidelines Setup* command from the main menu, the following dialog is displayed:



a: is the angle of the guideline's projection on the X-Y plane and the X axis.

b: is the angle of the guideline and its projection on the X-Y plane.

2.15.8. Geometry Tools



﴿◈◈╳๔≛₮₽

The icons of Geometry Tools allow you to lock the direction of drawing a line.



Begin to draw a line. Click the *Perpendicular* or *Parallel* icon then click an existing line or click two points to define the direction. The cursor will move perpendicular or parallel to this baseline.



Perpendicular to a plane

Begin to draw a line. Click the *Perpendicular to a plane* icon then click the domain defining the plane. The cursor will move perpendicular to the plane. The plane can also be defined by clicking three points.

These icons can be conveniently used while editing the geometry of the model or defining section planes.

🔺 Line towards a midpoint

Begin to draw a line then click startpont and endpoint of another line. Midpoint will determine the direction.

Bisector

Begin to draw a line then click the two legs of an angle. Bisector will determine the direction of the line.

Point of intersection

Begin to draw a node or a line then click the icon, click the two lines or their start and endpoint. A node or line point is created at the point of intersection. Any of the lines (or both) can be an arc. In this case there may be more than one point of intersection. If so, calculated points are marked with small circles. The required point has to be selected by clicking.



Dividing point

Begin to draw a node or a line then click the icon and click the two nodes. Specify the division by ratio or by distance in the popup dialog. A node or line point is created.



Point constraint operation

The action for *Point of intersection* and *Dividing point* can be set here. Two options are available: creating a node or moving the relative origin to the position calculated.

2.15.9. Dimensions Lines, Symbols and Labels

This group of functions lets you assign associative orthogonal and aligned dimension lines or strings of dimension lines to the three dimensional model, as well as angle, arc length, arc radius, level and elevation marks, labels for result values. Click on the Dimensions icon to display the Dimension Toolbar. That will allow you to select the proper dimension tool. Click on the left-bottom icon of the Dimension Toolbar to set the parameters of the selected tool.



You can change the position of dimension lines or labels at any time by dragging them to their new position. If the dimension lines were associated with the model their position and dimension will be continuously updated as you modify the geometry of the model.

2.15.9.1. Orthogonal Dimension Lines



Associative orthogonal dimension lines or strings of dimension lines, parallel with the global X, Y, or Z axes can be assigned to the model by following the next steps:

- 1. Click on dimension line start point and on the end point. If these points are connected by a line you can just click on the line.
- dx[×] dy[×] dz[×]
- 2. Move the mouse. The position of the dimension line depends on the direction in which you moved the mouse. There is one exception: when the segment is not parallel with any global plane and the editing is in the perspective view. In this case you have to select the direction dX, dY, or dZ from the toolbar.
- 3. Click the left mouse button to set the final position of the dimension line.

To insert a string of dimension lines, click on the points in the corresponding order or on the lines if any. Steps 2 and 3 are the same as for the individual dimension lines. A string of dimension lines can be selected at once if you click on one of them while depressing the Shift key. It allows you to move it as a group. To change the position of a group segment individually select it using the selection rectangle and drag it to its new position. As a result this dimension line will be removed from the group (it can be moved individually).



created automatically.



An example of smart dimension lines

If the dimension line is assigned to the points of a model, it will always behave in an associative way (e.g. will move with the model when the model is changed or resized or moved).



Orthogonal and Aligned Dimension Line Settings



- *Tick mark* Lets you set the tick marks of the dimension lines. You can select from nine predefined symbols.
 - *Color* Lets you set the color of dimension lines individually. You can get the color from the active layer. The dimension lines, marks, and texts are placed on the Dimensions layer by default but you can change it any time.
 - *Sizes* Lets you set the drawing parameters of the dimension line.

Dimension style/ Lets you to set the type and thickness of a dimension or extension line. You can choose a *Extension style* predefined value or get it from the active layer. You can turn on/off the display of extension lines.

- *Label orientation* Lets you set the orientation of the text labels of the dimension lines (Always horizontal, Always vertical, Auto horizontal/vertical, or Aligned to dimension line) inside or outside the dimension line.
- *Use defaults* Lets you restore the default setting.

Apply font to all Apply the same font to every dimension line. *symbols*

Save as default setting Apply to all dimension lines

ult Lets you save the current setting as default setting.

l Applies the current setting to all existing orthogonal or aligned dimension lines to ensure a uniform look.

Lets you select/define/set layers where the dimension lines will be placed. If there are no layers defined when you start defining dimension lines, a Dimension layer will be



automatically created. **See...** 3.3.3 Layer Manager

☑	-	
☑	-	

measurement

Text Parameters

Orthogonal dimension line	X
Settings Text parameters	
dX =2,134	Measured value Display unit of measurement Arial 14 pt Suffix
Use defaults Apply font to all symbols Save as default setting Apply to all dimension lines	Dimensions 💽 😅

Allows to you to define the settings of the text on the dimension lines.

- *Measured value* Allows you to place the measured value on the dimension line, using the current prefix and suffix settings. By clicking the Units and formats button the number format can be set in the Dimensions section of the *Settings / Units and Formats* dialog box.
- Display unit of Display of the unit of measured value.

Units and For- To change the current font parameters click the button below the *Units and formats...* button. *mats...*

Prefix Sets the prefix used with the text on the dimension lines. You can choose from the following options: Auto (dX, dY, dZ, dL = [depending on the direction]) Auto (DX, DY, DZ, DL = [depending on the direction]) User defined (this option will require you to enter the prefix).

Suffix Sets the suffix used with the text on the dimension lines.

2.15.9.2. Aligned Dimension Lines

×

Assigns aligned dimension lines or a string of dimension lines to the model.



 $\overleftarrow{\not \in_x} \not \in \breve{v} \not \in$

The steps are the same as the steps of creating an orthogonal dimension line. **See...**2.15.9.2 Aligned Dimension Lines

The plane of the parallel dimension line is determined automatically. There is one exception: when the segment is not parallel with any global plane and the editing is in the perspective view. In this case you have to select the direction X, Y, or Z from the toolbar. The plane of the section line will be defined by the segment and the selected global axis.



Sets the dimension line settings (See... 2.15.9.1 Orthogonal Dimension Lines). For aligned dimension lines the automatic prefix is always dL = or DL =.

An example of associative dimension lines (orthogonal and aligned):



2.15.9.3. Angle Dimension

- Associative angle dimensions, as the symbol of the angle between two segments, can be assigned to the model in the following steps:
 - 1. Click on start point and on the end point of the first segment. If the points are connected by a line you can just click on the line.
 - 2. Click on start point and on the end point of the second segment. If the points are connected by a line you can just click on the line.
 - 3. Move the mouse. The position and radius of the angle dimension will be determined by the mouse movement. Based on the position of the mouse, the angle, supplementary angle or complementary angle dimension can be entered.
 - 4. Click the left mouse button to set the angle dimension in its final position.





By clicking the Units and formats button the angle number format can be set in the Dimensions section of the *Settings / Units and Formats* dialog box.

2.15.9.4. Arc Length

Creates arc length dimension symbols in your model.

To assign this symbol to a full circle click any point of the circle and drag the symbol. To assign this symbol to an arc click any point of the arc and drag the symbol. To assign this symbol to a part of an arc click any endpoint of the arc, click the middle point of the arc and drag the symbol.



2.15.9.5. Arc Radius

1.2

 \mathbb{R}

Creates arc radius dimension symbols in your model. To assign this symbol to an arc click any point of the arc drag the symbol.

2.15.9.6. Level and Elevation Marks

1.2 Creates associative level and elevation marks in your model. By clicking the Units and formats button the number format can be set as the unit of Distance in the Geometry section of the *Settings / Units and Formats* dialog box. This is the unit and format used in the Coordinate Window.

See... 3.3.7 Units and Formats



Level marks can be placed in top view, by clicking on the desired point. The top view is defined as the view in the direction of gravity (You can change it in the *Settings / Gravitation* dialog). **See...** 3.3.8 Gravitation



Elevation marks can be placed in front view, side view, or in perspective, by following the next steps:

- 1. Click on the point you want to mark.
- 2. Move the mouse in the direction you want to place the elevation mark, and click to set the symbol in its final position.



_	
	- 2
₽.	-
₽.	_

Sets the level and elevation mark parameters.

Level and elevation marks	x
Settings Text parameters Level	Color By layer
$\frac{12}{Mark size} \xrightarrow{12} \frac{12}{12} \xrightarrow{12} \frac{12}{12} \xrightarrow{12} 1.2$ $\frac{Mark size}{h \square \mathbf{v}^{1.2} h [nm]} = 2.0$	
Use defaults Use defaults Apply font to all symbols Save as default setting Apply to all symbols	Dimensions

Level Selects the level mark symbol, and sets its size and format.

Elevation Selects the elevation mark symbol, and sets its size and format.

2.15.9.7. Text Box

Α

Creates an associative text box in your model.

You can enter multiline text in a text box. The text will use the same text formatting within a text box.



You can create a text box in the following steps:

- 1. Enter the text in the Text box parameters window, or in case of a single line text enter it directly into the edit field of the Toolbar.
- 2. Click on the point to which you want to assign the text box.
- 3. Move the mouse to the desired position and click to set the text box in its final position.

Text box parameters	Í
Settings	
Text	
Color	
E By layer	
• A d[mm] - 20	Arial 8 nt
d 4 4 1000	Analo pr
Use defaults	Dimensions 🗨 🖨
Apply font to all text box	,
Save as default setting	
_	OK Cancel

- *Color* Sets the color of the text, frame, and extension line. You can get the color from the layer.
- *Text box* These switches set the drawing parameters of the text box, frame, and extension line, the transparency and alignment of the text, and the *d* distance of the extension line from the reference point (to which the text box is assigned to).
 - *Font* Sets the text font, style and size.

You can reload and change default settings, apply text box or font parameters to all existing text boxes

- Active Links Active links can be placed in text boxes to attach any external information tot the model. If the text contains a file reference or a link to a web page clicking the text box launches the application associated to the file or URL instead of opening the above dialog. To change the text select text box first (e.g. Shift+click) then click into the box.
 - File reference
 A file reference is made of the -> characters and a file name. E. g.:

 ->C:\MyModel\Reports\Details.doc

 If no full path is specified AxisVM starts from the folder of the model. So if our model is in

 C:\MyModel we can enter: -> \Reports\Details.doc

 Clicking the text box starts the application associated to the file type. This way we can attach pictures, movies, sounds, Excel tables or other documents to any part of the model.
 - URL Supported protocols and link formats are: http://..., ftp://..., https://..., file://..., www. ... Clicking the text box the default web browser launches and opens the web site or file. If the text contains more than one URL, the first one is used.

2.15.9.8. Object Info and Result Text Boxes

1.2

Object info textElement or load properties appear in the text box depending on the current tab (Geometry,
boxboxElement or Loads). Information text box parameters can be set in a dialog:

ettings		
Object info text box display optio	ns I⊄ Element I⊄ ∀alue	I Properties I Unit
Color 🛛 🗖 By layer	Text box	
A d [mm] = 0		Arial 8 pt
<u>⊔</u> se defaults		Object info text boxes 💽 🗲
Apply font to all text box Save as default setting		OK Cancel

Result labels

When displaying results the cursor determines the value of the current result component
 on nodes, mid-side nodes, surface centers, or intermediate points of beams or ribs and shows it as a tooltip. The text of the tooltip is automatically entered in a text box.
 The steps of result labeling are similar to creating a text box.

The result text box is visible only when the selected result component is the same as the one that was selected when the result text box was created. For example an *My* result text box is displayed only when the *My* component is selected as the current result component.

Result text box options can be set in a dialog box:

Text box parameters	x
Settings	
Result text box display options	
Element Case Component Unit	In this load case only In all load cases For this result component only For all result components
	Element Case Case Unit
Color	Text box
By layer	*a `a ` <u>a</u> <u>ă</u> <u>ă</u> <u>ă</u> <u>ă</u>
A d [mm] = 0	Arial 8 pt
<u>U</u> se defaults	Result text boxes 🗾 🥩
Apply font to all text box Save as default setting Apply parameters to all text box	OK Cancel

In this load case only

Result label is visible only in the load case in which it was created.

In all load cases

Result label remains visible regardless the load case. The actual values will be updated on changing the case.

For this result component only

Result label is visible only if its result component is displayed.

For all result components

Result label remains visible regardless the displayed result component.



Result label text options :

Element:	Include element type and number.
Component:	Include result component name.
Case:	Include name of the load case, combination
	or description of the critical combination.
Unit:	Include unit name.



Below the button of Use defaults three checkboxes helps to customize the text box:

Apply font to all text box

After clicking the **OK** button only the font of all text boxes will change.

Save as default setting

New text boxes will appear using the current settings as default.

Apply parameters to all text box

After clicking the **OK** button parameters of all text boxes will be set to these values.

Layer Manager

[F11]

Lets you create new layers or modify existing ones.

This function is also available from the menu as *Settings\Layer Manager*. **See...** 3.3.3 Layer Manager

2.15.9.9. Isoline labels

- `1.2^{∠'} `1.0^{∠'}
- Lets you place a series of labels to isolines.
 - 1. Click to the Isoline labels icon
 - 2. Enter two points defining a line segment
 - 3. The labels are placed at the intersections of the segment and the isolines



2.15.10. Renaming/renumbering

Nodes, trusses, beams, ribs and domains of the model can be renumbered and renamed (their numbering follows creation order by default). To rename and renumber nodes or elements select them first then click the function icon on the Icon bar on the left.

Nodes (62)	Trusses
Trusses (16)	Start at 1
Beams (48)	Name T_
	Restore original numbers
n name strings eleme	nt number is represented as an underscore (_)

The list on the left shows the number of selected nodes and elements. Choose what you want to rename / renumber.

- *Start at* Enter the starting number. Selected elements will be renumbered in an order determined from their position. Renumbering can have effect on elements not selected as two nodes or elements cannot have the same number.
- *Name* In name strings element number is represented as an underscore (_). For example: if starting number is 1, and the *Name* field contains *T*_, the names of the selected elements will be *T1*, *T2*, *T3*, ... If only one element is selected it is not necessary to include _ in the *Name*. Otherwise it must be included as elements must have different names. If the *Name* field is empty, the name will be the number itself.

Restore original If *Restore original numbers* is checked clicking the OK button restores the original numbers of the selected elements and clears their names. Element type must be selected from the list on the left.

To turn on/off the display of numbers / names of elements open the Display Options dialog (**see...** 2.15.14 Display Options) or use the speed button (**see...** 2.16 Speed Buttons)

2.15.11.Parts

47

Lets you create sets of structural elements called parts. Working with parts makes the pre- and postprocessing easier.

AxisVM allows you to display one or more parts, called active parts, at the same time. In addition, if the Parts check box is enabled the commands will only affect or refer to the entities of the active parts. The name of the current part is displayed in the Info window. If more than one part is turned on n parts is displayed, where n is the number of active parts.

There are two types of parts: user-defined parts and logical parts. *User defined parts* are created by the user selecting elements belonging to the part. *Logical parts* are created automatically by the program sorting the elements into categories by different criteria (material, cross-section, thickness, element type, story, etc.).

You can activate an existing part by clicking its name in the list box.

Parts can also be activated without opening this dialog box by simply clicking the Parts speed button (at the bottom of the screen).

Depth of the tree expansion can be set by clicking on the numbers on the right hand side of the window.



New Creates a new user-defined part (a set of model entities).

You must assign a name to each new part. You must then define the new part by selecting entities (using the Selection Icon Bar if necessary) in the active display window.

Modify 🖓

Lets you modify the selected user-defined part. When the selection menu appears, the entities of the model that are in the part are displayed as selected.



Lets you delete the selected user-defined part from the list. This command will not affect the model.

AXL/VM/()

Logical Set Operations

Creates a new part by performing logical set operations on the user-defined parts of a model. You have to specify the set operations. To enter the name of a part, double click on the respective name in the list. Use the % symbol to include the entire model.

For example: %-Columns will create the part that will include the entire model less the part named Column.

Clicking on the Create button, you can enter in the Name field the name of the newly created part. If you want to use the +, -, , (,) characters in the name of a new part, you need to put the name between "" marks (example: "floor +12.00").

Logical operations	X
Parts	
Base - 3.00	
Base + 6.00	
Part1	
Part4	
Part5	
Part6	
Part/	
Parto Douto	
Double click on a list item to copy it to the expression. % is the logical symbol of the entire model. Operators: + (union), - (difference), * (intersection) and () brackets.	
Part1+Part9+"Base + 6.00"+"Base - 3.00"	
Name: MyNewPart Create	
Close	e

Creating new folders Creating folders offer a way of sorting user-defined model parts. Parts can be moved and rearranged by dragging them to a new position. **[Ctrl]** and **[Shift]** allows multiple selection in the usual way.

Turning folders on/off turns on/off the parts within the folder.

Logical parts

This dialog is to set criteria for creating logical parts.

Architectural objects are defined by their geometry. Vertical beams, ribs and trusses are considered to be columns, horizontal ones are considered to be beams. Domains in horizontal plane are slabs, domains in planes perpendicular to horizontal planes are considered to be walls.

If we defined stories we can create logical parts by stories.

Logic	al parts		x
	🔽 📴 Material		
I	 ✓ Structural members by cross-sec □ By element type (trusses, bea □ Øy enchitectural object type (context 	tion ms, ribs) olumns, beams)	
	Domains by thickness	ora⊓es, shells) labs, walls)	
	🔽 By stories		
∏ Sa	a⊻e as default		
		ок	Cancel

Display Display switches work in the following way:

switches

Turns on or off all the parts in the list.

Parts

All

If it is on only the parts checked in the list are displayed. If it is off the entire model is displayed.

Logical parts

Turns on/off display of logical parts.

- *When working on parts, only the data of the active parts will appear in the tables by default.*
- Auto Refresh

If it is on turning on or off parts will immediately cause a redraw. If it is off the screen is updated only after clicking the **OK** button.

Refresh all

If it is on parts will be turned or on off in all window panes in multi-window mode. If it is off part settings will be updated only in the active panel.

Show non-visible parts grayed

If it is on the entire modell wireframe is also displayed in gray to help identification of model parts.

58

_

2.15.12. Sections

Lets you create section lines, planes and segments through any surface model, that can be used to process the results (displacements, internal forces, etc).

If a truss, rib or beam is within an active section plane and the result component has values on these elements a diagram is displayed on these line elements too.



The dialog works similar to the Parts dialog.

Section lines, planes and segments can also be turned on and off using a speed button at the bottom toolbar.

If the result display mode is Section result diagrams are displayed only on section lines, planes and segments.

To reduce the complexity of drawings display of individual sections lines, planes or segments can be controlled to appear only in a certain load case and/or for a certain result component. Section segments, planes and lines are automatically sorted into three different folders (type groups).

Items cannot be dragged into another type group.





Section segment groups can be created to make it easier to turn on/off several section segments together.

Click *New section segment group*, enter a name for the group (*name*) and define any number of section segments. End definition by pressing **[Esc]**. Section segments will be numbered (*xx*) and get into the *name* folder as *name_xx*.

Creating new folders Creating folders offer a way of sorting sections. Segments can be moved and rearranged by dragging them to a new position within its own type group. **[Ctrl]** and **[Shift]** allows multiple selection in the usual way.

Turning folders on/off turns on/off the segments within the folder.

New section segment

To define the segment enter two points of a domain or on domains in the same plane. Setting the radio buttons you can control how the internal forces diagram will be displayed. Left or right segment width can also be specified.

Diagrams are usually displayed perpendicular to the element plane but checking the option *Draw diagram in the plane of the elements* rotates the diagram into the plane. In the *Display Parameters* dialog this parameter can be turned on/off for all section segments.



Display of the resultant integrated values



Display of the average values

New section plane Click *New section plane* and assign a name to the section. This type of section is based on a plane. Click or enter two points to set the section plane. Then click OK in the Selection Icon Bar to save. In perspective view you have to click or enter three points to set the section plane. Section planes are displayed as rectangles of dotted lines. You can enable/disable the display of section plane rectangles.

Section planes are useful when you want to display results only along a certain line through the entire structure.



New section line Click New section line and assign a name to the section. You then have to select surface edges or beam elements that define the section line. Then click OK in the Selection Icon Bar to save. Section lines can be discontinuous.



The checked section lines, planes and segments are active. You can use Auto Refresh and Refresh All checkboxes, New, Modify and Delete buttons the same way as in the *Parts* dialog.

The tracelines of the section lines are not correlated with the directions of the result components displayed.

2.15.13.Find



Finds the entity having a specified index, and moves the cursor over it. If *Select element* is turned on the element found will also be selected (displayed in purple).

Node Deam Deam	C Nodal Support C Line Support C Surface Support C Spring C Link C Edge Jinge C ARBO/CRET element	
C Rib C Strugtural Member C Surface C Domain		
CReterence	C Rigid	

2.15.14. Display Options



Symbols

Enables/disables the display of the symbols.

Graphics Mesh

Symbols

Enables the display of the inner mesh lines.

 \mathcal{G} When disabled only the outlines are displayed.

Node

Enables the display of the nodes (small black rectangles).

Surface center

Enables the display of the center point (selection point) of the surface elements.

G Color codes: plate = red, membrane = blue, shell = green.

Center of circle

 $G_{\mathcal{A}}$ Enables the display of centers of circles as a small cross.

Domain

Enables the display of the domain's contour.

- $G_{\mathcal{A}}$ The color of the domain is the same as of the surface type.
 - Color codes: plate = red, membrane = blue, shell = green.

Nodal support

Enables the display of the nodal supports.

- $G_{\mathcal{A}}$ Nodal supports appear as thick axes.
 - Color codes: axial displacement=yellow, axial rotation= orange.

Edge support

Enables the display of the edge supports.

 \mathcal{G} Edge supports appear as a thick edge.

```
Color codes: axial displacement=yellow, axial rotation= orange.
```

Surface support

Enables the display of the surface supports.

 Gev Surface supports appear as a light brown hatch .

Links

Enables the display of link elements.

GAN Node-to-node link elements are displayed as solid green lines with an arrowhead showing the location of the link.

Line-to-line link elements are displayed as solid green lines with an arrowhead showing the location of the link and dashed green lines at the line endpoints.

↔ Enables the display of rigid bodies. They appear as thick black lines.

Diaphragm

← Enabled the display of diaphragms as gray dashed lines.

Reference

Enables the display of the references.

G Red vector, crosshairs or triangle.

Cross-section shape

Enables the display of the shape of the cross-section of the truss/beam/rib elements.

 \mathcal{A} The user-defined cross-sections will be displayed as rectangles that circumscribe the shape of the cross-sections.

End releases

4

Enables the display of the end release and edge hinges.

End release:

\checkmark	Blue circle:	hinge / roller
	Blue circle + cross	semi-rigid hinge
	Red circle:	spherical hinge
	Solid blue circle:	plastic hinge
da	hinger	

Edge hinges:

 G_{C} Circles on the edges.

Structural members

Enables the display of the structural elements.

An orange line along the member and the number of the member.

Reinforcement param.

Enables the display of brown stars at surface centers where reinforcement parameters are assigned.

Reinforcement domain

Enables the display of mesh independent reinforcement domains as dashed brown outlines. Top and bottom x and y reinforcements are also displayed. Two vertices of the polygon are connected to the center by brown lines.



Mass

Enables the display of the symbol of the concentrated masses.

 G_{\bullet} Double red circle.

Story center of gravity

Enables the display of center of gravity of each story. AxisVM converts loads of load cases used to calculate the vibration shapes for seimic analysis into masses then calculates the center of gravity for each story. The centers are displayed as black +s in black circles with a label Gm*i* where *i* is the level number.

Story shear center

Story shear center is determined form wall sections at the story level. The method to determine shear center of thin walled cross-sections is used.

Enables the display of shear center of each story. AxisVM calculates story shear centers by finding wall sections and using the same method as for thin-walled cross-sections. The centers are displayed as red +s, with a label S*i*, where *i* is the level number.

ARBO-CRET elements

Aschwanden ARBO-CRET elements placed into the model.

A schematic drawing of the element is displayed.

COBIAX elements

COBIAX elements placed into the model.

↔ Void formers are displayed as circles in wireframe mode and balls in rendered view.

Object contours in 3D

Displays static model with a 3D wireframe look.

Local Systems Enables the display of axes of the elements in the local coordinate system.



Beam element local coordinate system





Surface element local coordinate systems

Loads Display of load symbols can be set separately for each load type (concentrated, distributed along a line, distributed on surface, temperature, self weight, moving load, miscellaneous (length changing, tension / compression).

To display of surface loads distribution to beams (see the diagram on the right) check *Load distribution*. To display the derived beam loads check *Derived beam load*.



Derived beam loadDisplaying of derived beam loadsMoving loadIf this option is turned on all phases of moving loads are displayed in gray. If this option is
turned off the moving load is displayed only in the position determined by the current load
case.Auto RefreshIf it is turned on any change in settings will make the active panel redrawn immediately.

- *Refresh All* Changes will affect all panels in multi-window mode.
- *Save as default* Saves the current symbol display option settings as default for new models.

Labels



÷

Numbering

Displaying the number of nodes, elements, materials, cross-sections, references.

For meshed line elements checking *Use finite element numbers* displays the number of finite elements instead.

ŝ

Q

Ê

Ē

3

ê

.

h

۰.

Checking/unchecking *Labels on lines seen from axis direction* turns on/off labels on lines seen from the direction of their axis (seen as points).

Properties Enables the display of the name and values of materials properties, cross-sections, element lengths or thicknesses, load values, masses.

If the Units option check-box is enabled, the labels will include the units as well.

Actual Enables labeling for top and bottom reinforcement in *x* and *y* directions independently and *reinforcement* sets the labeling mode.



2.15.15.Options



Allows the selection of the options for the settings of the grid, cursor, editing, drawing parameters, and design code.

2.15.15.1. Grid and Cursor

Grid The grid consists of a regular mesh of points or lines and helps you position the cursor to provide a visual reference. Depending on its type the grid is displayed as:

Dot grid – axes are displayed with yellow crosses, points in gray

Grid lines – axes are displayed in yellow, lines in gray.

Options Grid Cursor Editing Drawing	×
Grid ↓ Display ΔX (m) = 1,000	Cursor Step ✓ Mouse Snap ΔX (m) = 0,100
ΔY (m) = 1,000 ΔZ (m) = 1,000	$\Delta Y [m] = [0,100]$ $\Delta Z [m] = [0,100]$
Ord Crid Ord Crid Orid Lines	
Auto Refresh	OK Cancel

You can set the **grid parameters** as follows:

Display

Displays the grid if the check-box is enabled.

$\Delta X, \Delta Y, \Delta Z$

Sets the spacing of the dots/lines of the grid in the direction X, Y or Z.

Type

Sets the type of the grid.

Cursor Step Allows to choose coordinates of an invisible dot mesh (not the grid).

You can set the **cursor step parameters** as follows:

Mouse Grid

Restricts the movement of the mouse cursor to an invisible grid specified by the cursor step values below.

$\Delta X, \Delta Y, \Delta Z$

Restricts the cursor movement to regular intervals. Each time you press a cursor movement key the cursor moves in the corresponding direction (X, Y or Z) one step (ΔX , ΔY or ΔZ respectively).

Ctrl x

Sets the value of a factor that increases or decreases the cursor step size if you press the **[Ctrl]** key when you move the cursor. This allows you to achieve adequate positioning accuracy.

The cursor step is ignored if you position the cursor on a line not parallel to global coordinate axes. In such a case, the cursor will move along the line.

If the editing tolerance is greater than the cursor step, the mouse will follow an invisible grid specified by the editing tolerance.

When using with constraints, the cursor step is applied in the constrained direction with the DX value.

See... 4.7.4 Constrained Cursor Movements

If the grid step and the cursor step is set to the same value, nodes will be placed snapped to the grid.

2.15.15.2. Editing

Constraint Angle During the model editing the movement of the cursor can be constrained.

Using the **[Shift]** key while moving the cursor, the movement direction can be set. In this case the constrained movement of the cursor will be based on two types of angles (for other type of constrained movements **see...** 4.7.4 Constrained Cursor Movements).

Options	×
Grid Cursor Editing Drawing	
Constraint Angle Δα. [*] = 1500 Custom α. [*] = 0	Auto ✓ Intersect ✓ Part Management ✓ Refresh
Editing Tolerance $\delta[m] = 1E-3$	Cursor Identification
Plane tolerance C Relative (%) = 5 C Absolute (m) = 1E-2	Polar Coordinates © Cylindrical © Spherical
☑ <u>A</u> uto Refresh	OK Cancel

Auto Sets commands that are applied automatically if the corresponding check-box is enabled. **Intersect** :

Sets the line intersection handling. At intersection points of lines a node will be generated and lines will be bisected. If surfaces are intersected by lines, they will be split, and the resulting elements will have the same material and cross-sectional properties as the original.

Part management :

Any entity drawn or modified after the check-box is enabled will be associated with all of the active parts.

Refresh :

Sets the display refresh mode to automatic.

Editing Tolerance If two nodes are closer than the value set as the editing tolerance, they will be merged in the case of a mesh check. This value is also used when comparing surface thickness or beam length.

Cursor **h** identification



The element under the cursor is identified if it is within an adjustable cursor identification distance. The unit for cursor identification distance is pixels.

If more than one element is within this range the closest one will be identified. **See...** 4.7.1 Cursor Identification

Plane tolerance Nodes of domains and surfaces must be in plane. If a node of a domain or surface deviates from this plane more than the given value the element will be deleted. Plane tolerance can be specified in two ways:

Relative [‰]per thousand of the biggest extension of the
element polygonAbsolute [m]a given value

Auxiliary Cylindrical or spherical. *coordinates* **See...** 4.3.2 Polar Coordinates

2.15.15.3. Drawing

Load symbol Sets the display size of the load symbols. This *display factors* factor is applied when the checkbox in the Symbols icon / Graphics Symbols / Load is enabled. These values do not affect load values.

Force

Sets the display size of the symbol of concentrated force loads.

Moment

Sets the display size of the symbol of concentrated moment loads.

Line / surface load

Sets the display size of the symbol of line / surface loads.

Options	X
Grid Cursor Editing Drawing	
Load Symbol Display Factors	
Eor	ce 🚺
Mome	nt 1
Line Los	ad 1
Surface Los	ad 1
Contour Line Angle	Zoom Factor
φ["] = 9,00	1,3
Auto Refresh	OK Cancel

Contour line Sets the display of the inner mesh lines (between adjacent surface elements). The common *angle* edge of two or more surface elements is displayed if the angle enclosed by the normal to the planes of the elements is larger than the value set here.



Zoom factor Sets the scale of magnification/reduction of the zoom commands associated to the [+] and [-] keys.

2.15.16. Model Info

٢

Shows the main parameters of the model.

Clicking the *Analysis parameters* button certain parameters of the latest analysis (memory usage, running time) can be studied. This information is available only if the model was analysed by Version 10.

Model Information	×			
O3D_2DE_02_10				
Version: 10.0 Result file exists		Analucic naramotors		X
Model file:	2.85 M	Anarysis parameters		<u>~</u>
Created:	1.10.06-11.55		Linear analysis	
Modified:	11.12.09-14.13		Eniour unuyere	
Result file:	13.4 M	humi	her of Equations:	20262
Parts:	16	Tana Sana Sana Sana Sana Sana Sana Sana	diana Maranan	20203
Nodes:	1411	Equa	auons memory.	17310
Midside nodes:	2668	Solve	er block size:	45.8 M
Lines:	2848	Larg	est available memory block:	165 M
Beams:	174	Anal	ysis block size:	155 M
Ribs:	309	Avai	lable physical memory:	871 M
Surfaces.	1276	Total	l physical memory:	1.999 G
Supports:	12/6	Mode	el optimization:	0:00:08
Materials:	40	Mode	el Verification:	0:00:01
Cross-Section:	9	Anal	ysis:	0:02:10
Load Cases:	10	s	tiffness matrix evaluation:	0:00:27
Load Groups:		D	ecomposition of the system of equations (2):	0.00.33
Load Combinations:	1	B	acksubstitution	0.00.26
			alculation of internal forces:	0:00:15
		Pagi	# File Generation:	0:00:00
		11630		0.00.00
A = = b == 1 = = = = = = =				
<u>A</u> naiysis param	eters >>			
Dimensions:	Close		Save	Close
56.3 × 30.0 × 25.3m		I	2410	

2.16. Speed Buttons

The quick switches toolbar allows you to change the display settings without entering *the Display Option/Symbols* or Options dialog. The icons are located in the bottom right corner of the graphics area.

	C Story 5 C Story 4 © Story 3 C Story 2 C Story 1 C Ground floor
X	Auto Intersection
Þ	Mouse Snap
	Stories
	Parts in tree view
(F	Display Parts of the selected elements
$ \leq $	Workplanes
tt	Section Lines & Planes & Segment
	Display Mesh
Ħ	Display Loads Symbols
ŧγ	Display Symbols
,t_→	Display Local Systems
12	Numbering
わ	Background Layer
A)	Background Layer Detection

Some of these settings are available also from Display and Service icons.

2.17. Information Windows

The information windows are situated in the graphics area. You can move these windows on the screen by clicking title bar, holding down the left mouse button, and dragging it to a new location on the screen.

2.17.1. Info Window



Shows information about the display of the results such as: active part(s), current perspective setting, type of analysis, current design code, current load case or load combination, solution errors, current result component. For the explanation of E(U), E(P), E(W), E(EQ) parameters **see** 5. Analysis and 5.1 Static Analysis

If more than one part is activated a list of active parts is displayed provided that the number of parts does not exceed a limit. This limit can be set by right clicking the info window and clicking the *Settings* menu item.

2.17.2. Coordinate Window

X	X[m]:	1.056		r[m] :	1.275
	V[m]:	25.000	4	a[°]:	34.06
L L	L Z[m]:	0.714	u	h[m]:	25.000
	L[m]:	25.032			

See... 4.4 Coordinate Window

2.17.3. Color Legend Window

Displays the color legend corresponding to the result component being displayed in the postprocessor. You can resize the window and change the number of levels simply by dragging the handle beside the level number edit box or entering a new value. Colors will be updated immediately.

You can set the color legend details in the color legend setup dialog box. To open this dialog box simply click the color legend window.

Color Legend setup

eZ Color Legend Setu	p	x	j 📃 🛛
e2 Color Legend Setur 0,107 2,252 4,811 -6,970 -9,329 -11,688 -14,047 -16,406 -18,764 -21,123 -23,482 -23,482 -25,841 -28,200 -30,559 -32,918	Levels Limits Min, Max of Model -32,918 0,107 Min, Max of Parts -32,918 3,002 Absolute Max. of Model -32,919 32,919 Agsolute Max. of Model -32,919 32,919 Agsolute Max. of Parts -32,919 32,919 Qustom Auto Interpolate A = [Quate As. Hatching for out of range [Opaque] Trans Display V Auto Refresh	el s values parent	ez [mm] 0,107 -2,252 -4,611 -6,970 -9,329 -11,688 -14,047 -16,406 -18,764 -21,123 -23,482 -25,841 -28,200 -30,559 -32,918 15
	✓ Auto Refresh Refresh All OK	Cancel	

Setting criteria for the interval limits: Limits

Min/max of model

Sets the lower and upper limit values to the minimum and maximum values of the entire model. Intermediate values are interpolated.

Min/max of parts

Sets the lower and upper limit values to the minimum and maximum values of the active parts. Intermediate values are interpolated.

Abs. max of model

Sets the lower and upper limit values to the maximum absolute value of the entire model with the respective negative and positive signs. The intermediate values are interpolated.

Abs. max of parts

Sets the lower and upper limit values to the maximum absolute value of the active parts with the respective positive and negative signs.

The intermediate values are interpolated.

Custom

Click an item of the list on the left to edit its value. If you are in editing mode you can navigate through the list by UP and DOWN keys and edit the current item. When you click OK the series of interval values must be monotonically decreasing from top to bottom.

Auto Interpolate

If Auto Interpolate is checked the series will be recalculated each time you enter a new value. If you enter a new top or bottom value the recalculated series will be linear between top and bottom values. If you enter a new value at a middle interval the recalculated series will be bilinear, i.e. linear between the top and the new value and between the new and the bottom value but steps may differ.

By step value

Color values are determined by the given step Δ . When entering a new level value the other levels will be recalculated using the step. Switching from other crieria the array starts from the lowest value and using the latest step value.

You can save the settings of the scale using the *Save As* button. To review saved settings click the ... button.
Hatching for out Hatching for out of range values can be set to *Opaque* or *Transparent*. *of range values*

Standard interval limit settings are also available directly from the color legend window popup menu. To activate popup menu click right mouse button on the window.

Calculate

When displaying reinforcement values click *Custom* and *Calculate* to get the amount of reinforcement from rebar diameters and distances for the selected list item.

 When displaying actual reinforcement schemes AxisVM does not assign color to numerical values but to different rebar configurations. It can be set to display all schemes or just those within the active (visible) parts.

Reinforceme	nt	×	
Ø (r	nm] = 12	•	
Spacing [r	nm] = 200	┓	
		Add	
Diameter	Distance	Quantity	
Ø 12	250 452		
Ø 12	200 565		
I			
	A _s [mm²/m]	= 1018	
	ок	Cancel	

2.17.4. Perspective Window Tool



See... 2.15.3 Views

This page is intentionally left blank.

3. The Main Menu

3.1. File



The menu commands are described below.

3.1.1. New Model

Ρ

New Model				x
Select a view to	start with			
	<u>F</u> older	C:\Program Files\AxisVM10\Peldak\	- 2	2
<u>⊥</u> op View	<u>M</u> odel Filename:	Model 1		
ĭ∰.	<u>D</u> esign Code	Eurocode	\bigcirc	
Front View	Units and Formats	EU Units 💌	Change Settings	
ĺŁĭ∰	<u>R</u> eport Language	English		
Perspective	— <u>P</u> age Header ———			_
	Project			
	Analysis by InterCAD			
	Comment			
	Project Analysis by InterCAD Model: Model 1.axs			
		[OK Cancel	

Creates a new untitled model. Use this command to start a new modeling session. If you have not saved the current model, a prompt appears asking if you want to save it first. Refer to the *Save* and *Save As* commands for more information on how to save your current model.

You must specify a name for the new model. You can select the appropriate Standard and system of units. You can enter specific information in the Heading section, that will appear on each printed page.

A new model uses the default program settings.

3.1.2. Open

[Ctrl]+[0] Loads an existing model into AxisVM. If you have not saved the current model, a prompt appears asking if you want to save it first. Refer to the *Save* and *Save As* commands for more information on how to save your current model.

Selecting this command will bring up the Open dialog box.

If the folder name appearing in the dialog box is what you want, simply enter the file name in the edit box or select it from the list box. If the directory is not what you want, select the drive and directory names along with the file name.

AxisVM saves your model data in file names appearing as Modelname.AXS (input data), and Modelname.AXE (the results). Both file contains the same identifier unique for each save which makes it possible to check if AXS and AXE files belong to the same version of the model.

	Open an Axis¥M Model
Current drive	Look in: 🔁 examples 💽 🗢 🖻 💣 🖽 -
Model data files in the current folder	W OSZLOPVASALAS.axs SteelFrame2.axs History OSZLOPVASALAS2.axs TK1-5T-1.AXS History RT1-ST-1.AXS VL1-ST-I.AXS Desktop RT2-ST-1.AXS VL1-ST-I.AXS Desktop SteelFrame.axs VL1-ST-I.AXS My Computer SteelFrame.axs VL-RZ-1.AXS My Network P SteelFrame_spanyol_vbeton.axs VL-RZ-1.AXS File name: VL1-ST-I.AXS VL-RZ-1.AXS
M. 1.1. (.	Files of type: AxisVM Models (*.axs) Cancel C:VAXISVAxisVM7_angolvexamplestVL1-ST-I_Recuperado1.axs Preview
Model info	Version 7.0
	Ho result file found
	Model file: 641 k
	Ureated: 16.02.04-17.01 Model display
	Design Code: Funccide
	Nodes: 446
	Trusses: 324
	Beams: 662

3.1.3. Save

[Ctrl]+[S] Saves the model under the name displayed at the top of the AxisVM screen. If you have not saved the model yet, the Save As dialog box automatically appears prompting you to enter a name. Use the Save As command if you are changing an existing model, but want to keep the original version.

If you enable Create Backup Copy check box in the *Settings / Preferences / Data Integrity / Auto Save* a backup file of your previous model will be created.

3.1.4. Save As

Names and saves the model. Use this menu command to name and save a model if you have not saved the model yet, or if you are changing an existing model, but want to keep the original version.

Selecting this menu command will bring up the Save As dialog box.

- *Converting* Models created with previous AxisVM versions (if applicable) will be converted into the *models* current version file format when you open them for the first time.
 - The File / Save As / File Format command lets you save the model in earlier formats.

3.1.5. Export

~	
DXF file	Saves the geometry of the model to a DXF file format for use in other CAD programs. The geometry is saved with actual dimensions, in a <i>Modelname</i> . DXF file. Selecting this menu command will bring up the Export DXF dialog box, that lets you specify the units of measurement in the exported file.
	Three different formats are available for DXF output. - AutoCAD 2000 DXF file - AutoCAD R12 DXF file - AutoCAD reinforcement design file
Tekla Structures file	 Two different file formats are available: Tekla (TS) Structures ASCII file (*.asc) Saves the geometry of the model into a file format that is recognized by Tekla Structures. The file includes the coordinates of i and j-end nodes, the cross-sectional properties and the reference point of truss and beam elements. Tekla (TS) DSTV file (*.stp) Saves the data of the truss and beam elements (endpoints, material, cross section, reference) as a standard DSTV file. This file format is supported by several steel designer CAD software.
Bocad file	Saves the geometry of the model into a file format that is recognized by the Bocad software. The file includes the coordinates of i and j-end nodes, the cross-sectional properties and the reference point of truss and beam elements.
StatikPlan file	For StatikPlan AxisVM exports a DXF file including the contour of the reinforced concrete plate, the calculated reinforcements as isolines and the result legends on different layers.
PianoCA file	Generates a *.pia interface file for PianoCA. It includes the data, supports, loads and the calculated results of the selected beam elements.
IFC 2x, 2x2, 2x3 file	Exports an IFC file describing the model with achitectural objects (walls, slabs, columns, beams). IFC files can be imported in ArchiCAD, AutoDesk ADT, Revit, Nemetscheck Allplan, Tekla-Xsteel and other architectural programs.
CADWork file	Creates a DXF file to use in CADWork reinforcement detailing software. Only selected domains will be exported. As CADWork works in 2D, selected domains must be in the same plane. Each domain in the DXF file is transformed to a local X-Y coordinate system, Z coordinate represents the calculated amount of reinforcement.
SDNF 2.0, 3.0 file	Saves the model in SDNF (Steel Detailing Neutral Format) file readable by steel detailing products (Advance Steel, SDS/2, Tekla Structures, PDMS).
AxisVM Viewer	Saves the model in AxisVM Viewer format (*.axv). See 7. AxisVM Viewer and Viewer Expert

AXIJVM1()

AXS file	The following groups of elements can be exported: the entire struc- ture, displayed parts or selected elements. To select export options similar to those of the Copy options (see 3.2.6 Copy / paste options) click the <i>Settings</i> button of the <i>Export</i> dialog.	Export to AxisVM file Elements to export Entire model Displayed parts Selection Copy associated objects Supports Selected supports Loads Selected loads Dimensions Selected dimensions Reinforcement domains Selected reinforcement domains Copy load cases of the loads copied Copy all load cases Copy all load combinations Copy all load groups OK Cancel
Export Selected Only	Exports only the elements that are in the	ne current selection set.
Coordinate units	The coordinate units of the exported fil	le can be selected here.

3.1.6. Import



78

AutoCAD *.dxf

- Imports a geometry mesh from a DXF file (drawing interchange file) exported in AutoCAD 12, 13, 14 and 2000 format into AxisVM. The layers of the imported file are loaded into the Layer Manager. **See**...3.3.3 Layer Manager
 - If the file date of the imported file has changed, the Layer Manager will ask if you want to update the layers.
 - Selecting this menu command will bring up the Import DXF dialog box.
 - The ellipses will be converted to polygons only if you load them as active mesh otherwise they remain ellipses.

Import Model

	Coordinate Unit: m
	Maximum Deviation From Arc [m] = 0.050
	Geometry Check Tolerance [m] = 0.005
Import As	Place
Actual Nodes & Lines	Select base plane
C Background Layer	z 💽 Plane X-Y
Import Mode	Y C Plane X-Z
C Quorierito	x Plane Y-Z
G Add	Place

The default unit is meter [m].

Parameters Input units

You need to specify the length unit used in the imported DXF file.

Maximum deviation from the arc [m]:

Importing a DXF file as an active mesh, ellipses will be converted to polygons based on this value.



Geometry check tolerance

When you import a DXF file as an active mesh, AxisVM checks for coinciding points (nodes) and lines in your model, and merges them.

You can specify the maximum distance to merge points. Points that are closer together than the specified distance are considered to be coinciding. The coordinates of the merged points (nodes) are averaged.

You must always set this to a small number relative to your model dimensions.

Import As You must specify whether you wish to use the imported DXF file as an active mesh or as a background layer.

Active mesh (nodes&lines)

The imported geometry is considered as if it were created with AxisVM commands. DXF layers can be used to create parts.

Background layer

The imported geometry is used as a background layer that is displayed but is inactive as a mesh. Import a DXF file as background layer when you want to create the model based on architectural plans or sections. You can use the entities in the background layer as a reference during editing your model.

- *Import Mode* You can choose between overwriting the former geometry or adding a new geometry to the former one
 - *Place* Lets you specify the plane of the DXF layer (X-Y, X-Z, or Y-Z). The Place button allows to graphically position the imported DXF drawing in your model space.
- IFC 2.0, 2x and 2x2, 2x3 *.ifc file Imports objects from an architectural model saved as an IFC file. Imported objects can be displayed as a 3D background layer or can be converted to a native model by assigning materials, cross-sections etc. to them. Existing architectural models are always overwritten by the new one.

You can import object based architectural models from ArchiCAD, AutoDesk Architectural Desktop, Revit Structure, Revit Building Nemetscheck Allplan, Bocad and Xsteel. Programs.

Import IFC file	×
Import Method	Import Method
Overwrite	C Static Model
C Update	Architectural model objects
Arc resolution	
0	Maximum Deviation From Arc [m] = 0,050
•	By angle [*] = 5
Joining objects	
If objects are closer than	
	ε[m] = 0,010
	OK Cancel

Importing IFC files can extract the static model (if available) or the architectural objects overwriting or updating the existing information within the AxisVM model.

Static model From IFC version 2x3 it is possible to export details of the static model (nodes, topology, supports, loads, load combinations). The *Static model* option is available only if the file contains this information. If it describes architectural objects (columns, beams, walls, slabs, roofs) only the static model can be created automatically in AxisVM after importing the file.

ArchitecturalThis option can overwrite or update existing architectural model information in the AxisVMmodel objectsmodel. AxisVM can read columns, beams, walls, slabs, roofs.See... 4.9.20 Creating model framework from an architectural model

When exporting a model from ADT (Architectural Desktop) turn off the automatic intersection of walls before creating the IFC file.

AxisVM *.axs	Imports a model from an existing AxisVM file into AxisVM, and merges it with the current model					
	During the merging process, the Geometry Check (See Section 4.8.14 Geometry Check) command is automatically applied. If there are different properties assigned to the same merged elements, the properties of the current model will be retained. Load groups and combinations if any, are appended to the existing ones as new groups and combinations, and the load cases as new cases. If no load groups or combinations are defined in the imported model, the load cases will be appended to the existing ones as new cases. If the same case exists in both models, the loads will be merged.					
	If both models contains loads that are limited to one occurrence (e.g. thermal) in the same load case, the load in the current model will be retained. The Section Lines/Planes Parts with the same name are merged, otherwise they are appended.					
	When importing an AxisVM file the following dialog is displayed: Importing Shell_1.AXS Geometry Check Tolerance [m] = 0,001					
	Place OK Cancel					
	Use the Place button to graphically position the imported model in your model's space.					
Stereo Lithography *.stl file	Reads the triangular mesh describing the surface of a model from a file in STL format (binary or text). Multiple nodes and degenerated triangles are filtered out. Import can be transferred to a background layer as well.					
Bocad interface *.sc1 file	Opens a data file created by Bocad steel construction software (*.sc1) and imports beam cross-sections and geometry.					
Glaser -isb cad- *.geo file	Imports *.geo files exported by Glaser -isb cad- describing beam or surface models.					
SDNF file (Steel Detailing Neutral Format)	Imports a file exported in Steel Detailing Neutral Format used in data exchange between steel detailing programs.					

3.1.7. Tekla Structures – AxisVM connection

Setup

The connection between the two software is made through a COM server enabled to run AxisVM. To make the connection work first the COM server must be registered within the operating system (in the Registry) then Tekla Structures must be notified that a compatible server is available.

AxisVM setup automatically performs these registering operations, however if Tekla Structures is not installed the second registration cannot be completed. Therefore after installing Tekla Structures the registration has to be started again by running two batch files from the AxisVM program folder:

!REGISTER_AXISVM.BAT !REGISTER_TEKLA.BAT

If connections fails any time it is recommended to run this registration again.

Connection

After a successful registration the model built in Tekla Structures can be transferred to AxisVM in the following way: click *Analysis & Design models...* in the *Analysis* menu then click the *Properties* button to set AxisVM AD Engine as the *Analysis engine*.



Dadose e roed 115 0 Part | Durier priste 1 0-1 Depuis entre BRSart | 21 @ 21 P | 2166 Structures - C/... Netati Shop Pro - Image 1 2266

🕅 Analysis & Design models							
Analysis model name	Creation method	Results	Number of	parts			
Modell 1	By selected p	Status unkno	91				
	1 1		.		[- 1
Properties	New	Delete	Hetresh	Details	Load combinations	Select objects	Draw
Run		Create model	View results	Hide results	Get results	Get results for selected	<u>C</u> lose

🕅 Analysis model attributes	
Design - Steel	Design - Concrete Design - Timber
Analysis model Analysi	s Job Output Seismic Seismic masses Modal analysis
Analysis engine:	
Analysis model name:	Format: CIS/2
Model merging	Format: IFC2x2 Format: IFC2x3
Creation method:	AxisVM AD Engine v1.0
Member axis location	Reference axis
Member end release method:	E By connection
Node definition:	Force to centric connection 💌 🔽 Extended clash check
🔲 Modal analysis model	

If AxisVM AD Engine does not appear in the dropdown list the registration was not successful and has to be repeated.

Getting back to the *Analysis & Design models* dialog click *Run* to start the transfer of the model. The process status is displayed in dialog. If the transfer is completed successfully click the OK button to see the model in AxisVM.

	Materials:	4	Lines	115	Loads:	348
Cross-	Sections:	11	Supports:	70	Load cases:	5
	Nodes:	70			Load Combinations:	14
atus						
007.10.03	12:40:35	.687]	ок			
007.10.03	12:40:35	.687]	Creating load combination (Te	kla load c	ombination ID: 9)	1
007.10.03	12:40:35	.703]	OK			
007.10.03	12:40:35	.703]	Creating load combination (Te	kla load c	ombination ID: 10)	
007.10.03	12:40:35	.703]	OK			
007.10.03	12:40:35	.703]	Creating load combination (Te	kla load c	ombination ID: 11)	
007.10.03	12:40:35	.703]	OK			
007.10.03	12:40:35	.703]	Creating load combination (Te	kla load c	ombination ID: 12)	
007.10.03	12:40:35	.703]	OK			
007.10.03	12:40:35	.718]	Creating load combination (Te	kla load c	ombination ID: 13)	
007.10.03	12:40:35	.718]	OK			
007.10.03	12:40:35	.718]	Creating load combination (Te	kla load c	ombination ID: 14)	
007.10.03	12:40:35	.7181	OK	na ioaa o	ombination ib. Try	

The model transferred to AxisVM:



Loads and load cases specified in Tekla Structures are also converted.



3.1.8. Page Header

Lets you specify a header text (two lines), which contains the name of the project and designer. It will appear on the top of every printed page. An additional comment line can be added.

Page Header	×
Steel frame structure A2b	Α
Designed by Inter-CAD Kft.	Α
Comment Page header example	
Steel frame structure A20 Designed by Inter-CAD Kft. Page header example Model: SteelFrame.axs	
A Default Settings OK Cancel	

3.1.9. Print Setup

ű,

Allows setting the parameters of the default printer.

This is a standard Windows dialog therefore its language corresponds with the language of the installed operating system.

3.1.10. Print

Printing drawing



[Ctrl]+[P] Lets you print the model according to the current display settings. Allows the setup of the printer, and of the page.



Send To

Lets you send the output directly to the printer/plotter or to a graphics file (DXF, BMP or Windows Metafile [WMF/EMF]).

Printer

Lets you select and setup the printer.

If a file is selected as output, the printing will be stored in the *Name.prn* file, where Name is a file name to be entered.

You can set the number of copies required.

The Setup button invokes the standard Windows Printer Setup dialog where you can change printer and printer settings in detail.

Scale

Lets you set the scale of the drawing to print. In case of perspective or rendered view or if the output is sent to a Windows Metafile the scale cannot be set.

Margins (Printer/DXF)

Lets you set the size and the units of the page margins. You can also drag margin lines within the preview area by their corner and midside handles.

Bitmap Size (BMP, JPG)

Lets you set the bitmap size in pixels, inch, mm or cm and bitmap resolution in dpi (dots per inch).

Preview

Lets you view the printed image prior printing. If you select Printer as a target the graphics cursor turns to a hand whenever it enters the preview area. By pressing the left mouse button and moving the mouse you can specify an additional panning which will affect the printed output only.





Page Header

Lets you set the date and remark that will appear on each page, and the starting number for the page numbering. If the Page numbers checkbox is turned off a blank space will appear after *Page* allowing handwritten page numbers.

Orientation

Lets you set the orientation of the page.

Color Options

Lets you select printing in grayscale, color, or black and white.

If your printer cannot print in color you may get different results in the first two cases. If you select Grayscale the output will be converted to grayscale using an internal grayscale palette of AxisVM. If you select Colors the conversion to grayscale will be performed by the Windows printer driver. Try both to find which works better for you. When black and white printing is selected, all entities are printed in black.

Paper size

Lets you set the size of the paper.

Change Fonts

Lets you select fonts to be used in printing and set the font size.

Pen widths

Sets the size of the pens for printing.

Thick lines are used for drawing supports and rigid elements. Medium lines are used for isolines and section line. Thin lines are used for elements and geometry and other entities.

Pen Widths	×
Pen Widths	
📀 mm 🔿 in	
	Thin = 0,15
	Medium = 0,30
	Thick = 0,50
ок	Cancel

Windows to Print

Lets you print either the active window or all windows displayed.

Printing to file When Print to File is selected the printing is redirected to a file, name.prn that you can print anytime later.

If the file *name.prn* already exists, you can add your printing to it, or overwrite it.

If you want to print only into files, you can set the operating system to do so in the *Start/Settings/Printers* choosing Properties and setting the Print to the Port as File. In this case you can not append print files.

Printing table When printing from the table browser, you can set the pages (all / even / odd) of all / current / selected pages you want to print.

Example: Entering 1, 3, 7-10, 20-18 in the Selected field the 1st, 3rd, 7th, 8th, 9th, 10th, 20th, 19th, and 18th page will be printed in this order.

able Printer				x
Send To Printer DXF File BMP File GyG file C Windows Metafile	Printer \\SERVER\HP Laser Jet 4000 S (LPT1:) Network; Ready Setup	eries PCL 6 Copies: 1	Page 11 of 11	
✓ Page Header Date: 12/14/2009 Comment: ✓ ✓ Page numbers ✓ Table Comment ✓ Color Options ✓	Setup First Page Number 1	Margins Unit: mm ▼ Left: 20.0 ♀ Top: 10.0 ♀ Right: 10.0 ♀ Bottom: 10.0 ♀ Orientation ● ♀ © Portrait ⊂ Lgndscape		
Table of contents	<u>P</u> aper Size	A4 💌		
Pages To Print All Current Page C Selected	Selected: 1-1 All p	ages		OK Cancel

3.1.11. Printing from File

You can print the prn file you created from the following window.

rinting Options			2
Printer			
Name: HP LaserJet 4000 Series PCL 5e		9	Setun
Status: Default Printer; Ready			
Location \\SERVER\HP 4000 HU			
		Copies:	: 1 🌒
C:VAxisVM60VProjectVPage 10.PRN	i i i i i i i i i i i i i i i i i i i		
C:VAxisVM60/Project/Page 9.PRN			
C:VAxisVM60\Project\Page 8.PRN			
C:\AxisVM60\Project\Page 7.PRN			
C:VAxisVM60/Project/Page 6.PRN			
C:\AxisVM60\Project\Page 4.PRN			
C:\AxisVM60\Project\Page 3.PRN			
C:\AxisVM60)Project/Page 2 PRN			
C1AxisVM60)Project/Page 1 PRN			
	- 1	ОК	Cancel
C:VAxisVM60/Project/Page 2.PRN C:VAxisVM60/Project/Page 1.PRN	* *	ок	Cancel

You can print more than one prn file at a time. You can set the printing order with the up/down arrows in the right of the file list box, or dragging the file names to a new position with the mouse.

3.1.12. Model Library

Ø

The *File/Model Library* command lets you preview, get information and manage your model files.

As in Open and Save As dialog windows the standard file access dialog box items are displayed, but in the list box you can select multiple files.

Gev The AxisVM model files are marked with the \mathbb{A} symbol. If a model has a result file the symbol has a blue right-bottom corner, \mathbb{A} .

	Curren	nt drive			C	Curren	t folder			
C+\ AvicYM60 \ Deo	iect\Droject	006								Y
		_000		-					_	
	C:VAxisVM60VP	roject 💌	14,7G free	•	≣ ::[]	<u> </u>	C:\\Project		14,5	G free
Name	Size	Saved M	lodified 🔺	-	Name		Size	Saved	Modified	
1					1					
Project_001	242k	01.01.23 01	1.01.23	P -	\Lambda AG1-ST-	-1	57k	99.12.08	99.12.04	
Project_002	242k	01.01.23 01	1.01.23	40	\Lambda Ah-ki		143k	00.05.31	99.12.05	_
Project_003	242k	01.01.23 01	1.01.23	B	AK-KI		114k	01.01.23	99.12.04	
Project_004	242k	01.01.23 01	1.01.23	abc	Ak-P-NL		29k	99.12.08	99.11.28	
Disproject_005	242K	01.01.23 0	1.01.23				OUK	00.05.31	99.12.04	
A Shell 1	1334	01.01.23 9	9 1 2 04	\sim	A AK-ST-I		2194	00.03.31	00 11 24	
A Shell 2	133k	01.01.23 90	912.04	-24	A AK-ST-II		210k	00.12.00	00.11.24	-
•			▶		 					
1 of 12 files	Model / R		474k		23 files					
C:\AxisVM60\Project					🔽 Pre	view				
Project_006		Nodes:		448						
Vereien: R.O.		Lines:		987		1				
No result file found		Trusses:		324		1000	TA (1000		
Model file:	474k	Supports:		662	6		P A E	7 A E	an the	
Created:	14.09.99-20.09	Materials:				ΛÌ	KV E	7 K	XD,	1
Modified:	23.01.01-15.29	STEEL			K	187	Y K	176	V A	2
		Cross-Sect	tion:	16		¥ AS	8 A	2 R	AR	Se l
		Parts:		6	1	SAR	7 A	4 K	5A /	P
		Load Cases	s: inations:	4	6	VBK .	L A	4 1	XX/	1
		Eodd Comb	in following.	4	Z v	20	342	57	ØS -	
					K.,	([–]		4		
Project 1				_						
Analysis by M mérnökir	oda Kft.				Dimens	ions: 81,0	×81,0×24,0 m		Clo	se
. ,										
					urrontr	model	provio	47		
	Current model preview									

New

Creates a new sub-folder in the current folder with the name you enter.

Copy

Copies the selected files to a different folder. You can specify whether to copy the result files or not.

Rename/Move

Renames the selected files in the current folders or moves them into a different folder.

Delete

Deletes the selected files from the current folders. You can specify to delete only the result files or all.

子 Open

Opens the selected file for editing.

 \checkmark AxisVM files are marked with \blacksquare . If a result file is available, the bottom right corner of the icon \blacksquare is blue.

Preview

Shows the model wireframe in front, side, top view or in perspective depending on the model dimensions. Model information is also displayed in a list.

Close

Quits the Model Library.

3.1.13. Material Library

٢

AxisVM provides a preloaded material library (that contains the most frequently used structural materials) and allows you to create material property sets that you can use over and over again in many different models. You must assign different names to each material property set.

🔀 Table Browser											١×
<u>File E</u> dit F <u>o</u> rmat <u>R</u> eport <u>H</u> elp	5										
Loads → × № № 11 🖨 🖸 1											
- Load Combinations Structural Materials - Eurocode											
Weight Report LIBRARIES Material Library		Name	Туре	E _x [N/mm ²]	E _y [N/mm ²]	ν	α _T [1/°C]	ρ [kg/m ³]	Material color	Contour color	-
Structural Mater	7	C40/50	Concrete	35000	35000	0,20	1E-05	2500			
Eurocode	8	C45/55	Concrete	36000	36000	0,20	1E-05	2500			
Eurocode [A]	9	C50/60	Concrete	37000	37000	0,20	1E-05	2500			
Italian code	10	S 235	Steel	2,1E+05	2,1E+05	0,30	1,2E-05	7850			
MSZ	11	S 235 H	Steel	2,1E+05	2,1E+05	0,30	1,2E-05	7850			
	12	S 235 W	Steel	2,1E+05	2,1E+05	0,30	1,2E-05	7850		•	-
Editing C30/37, Material Name											
								ок		Cancel	

The material library window can also be opened using the Table Browser icon and by selecting Libraries/Material Library. **See...** 4.9.7 Line Elements, 4.9.20 Creating model framework from an architectural model

See the detailed description of the Table Browser in section 2.9.

Properties of materials

This table contains the properties of materials often used in civil engineering to the MSz, Eurocode, DIN-1045, DIN-1045-1, NEN, SIA-162, a STAS and Italian codes. You can add, modify, or delete existing material data. In case of entering a new material with an existing name it will be added as **materialname_number**. These materials can be used in any model.

Changes in the material library does not reflect in models using the modified material.

Define new material

Change material properties When entering a new material, the following dialog is displayed:

Definig new material or clicking to a non-editable column (eg. national design code, type) a dialog appears, in which all material properties, calculation and design parameters can be defined or changed. The fields containing the basic properties independent of the design code can be edited in the table.

When a material with a name identical to one existing is entered an index is attached to the name (*name_index*) to differentiate from the existing one.

If no texture was assigned to the material click the sample rectangle to select one from the library. **See...** 2.15.6 Display Mode

	Material Properties				×
)	N <u>a</u> me <u>T</u> ype urrent Design Code	STEEL FE 510 Steel Steel	Color Outline Color	Texture -
	Na	tional Design Code	Eurocode	Steel design	a parameters
		<u>M</u> aterial Code	EN 10025	f _y [h	l/mm ²] = 355
	-Material parameters-			f _u [h	Wmm ²] = 510
	Material model:		E _x [N/mm ²] = 210000	- ք _y (Ի ք _u (Ի	Wmm ²] = 335 Wmm ²] = 490
	Isotropic		v = 0.30		
	Orthotropic		α _T [1/°C] = 1.2E-5		
			ρ [kg/m³] = 7850		
				_	OK Cancel
Properties	For each materia - Material ty - Design cod - Material na - Fill color or - Contour liu	al the follow pe: [Steel, le, material me n the screen ne color on	ving properties ar concrete, timber, code the screen.	e stored: aluminum, o	other]

- Texture

Analysis Parameters

Mater

You can specify the material as isotropic or orthotropic **General parameters:**

Ex	Young's modulus of elasticity in the local x direction
Ey	Young's modulus of elasticity in the local y direction
ν	Poisson's ratio
α_{T}	Coefficient of thermal expansion
ρ	Mass density

In case of timber materials:

 ρ is the air dry mass density (12% humidity) and, the modulus of elasticity *E* is based on bending test results. The effect of time (relaxation) is not taken into account.

User's Manual

Design Parameters

Design parameters depend on the material type and the design code.

EC,		f_y	Yield stress						
DIN 1045-1,	staal	f_u	Ultimate stress						
SIA 26x,	steer	f_y^*	Yield stress 40mm <t< 100mm)<="" td=""></t<>						
Italian		f_u^*	Ultimate stress (40mm <t< 100mm)<="" td=""></t<>						
		f_{y_d}	Yield stress						
	steel	f_{yt}	Ultimate stress						
NEN		f_{yd}^*	Yield stress (40mm <t< 100mm)<="" td=""></t<>						
		f_{yt}^*	Yield stress (40mm <t< 100mm)<="" td=""></t<>						
		f_{ck}	Characteristic compressive cylinder strength at 28 days						
EC,		γ_c	Partial factor						
Italian	concrete	$lpha_{cc}$	Concrete strength reduction factor for sustained						
			loading						
		Φ_t	Creeping factor						
		f_{ck}	Characteristic compressive cylinder strength at 28 days						
		f_{ck} , $_{cube}$	Characteristic compressive cylinder strength of cube						
DIN 1045-1	concrete	γ_c	Partial factor						
	concrete	α	Concrete strength reduction factor for sustained loading						
		Φ_t	Creeping factor						
		f _{ck}	Characteristic compressive cylinder strength at 28 days						
SIA 26x	concrete	V	Partial factor						
		Φ.	Creeping factor						
		f'_{ck}	Characteristic compressive cylinder strength at 28 days						
NEN	concrete	Φ	Creeping factor						
		f _{m.k}	Characteristic bending strength						
		$f_{t,0,k}$	Characteristic tensile strength parallel to grain						
		$f_{t,90,k}$	Characteristic tensile strength perpendicular to grain						
		$f_{c,0,k}$	Characteristic compression strength parallel to grain						
		$f_{c90,k,y}$	Characteristic compression strength perpendicular to						
		2,,9	grain (y)						
			(for solid and Glulam timber $f_{c90,k,y} = f_{c90,k,z} = f_{c90,k}$)						
		$f_{c90,k,z}$	Characteristic compression strength perpendicular to						
			grain (z)						
			(for solid and Giulam timber $J_{c90,k,y} = J_{c90,k,z} = J_{c90,k}$)						
EC	timber	f _{v,k,y}	Characteristic shear strength (y)						
	uniter		(for solid and Glulam timber $f_{v,k,y} = f_{v,k,z} = f_{v,k}$)						
		$f_{v,k,z}$	Characteristic shear strength (z)						
			(for solid and Glulam timber $f_{v,k,y} = f_{v,k,z} = f_{v,k}$)						
		E _{0,mean}	Mean Young's modulus of elasticity parallel to grain (x)						
		E _{90,mean}	Mean Young's modulus of elasticity perpendicular to $\frac{1}{2}$						
		Ecos	5% modulus of elasticity parallel to grain (y)						
		G _{mean}	Mean shear modulus						
		ρ_k	Characteristic density						
		0 moon	Mean density						
l		V NA	Partial factor of the material						
		<i>i IV</i> 1	Size effect exponent (for LVL materials)						
		5	Size effect exponent (for LVL materials)						

3.1.14. Cross-Section Library

AxisVM provides preloaded cross-section libraries, that contain the most frequently used steel shapes and concrete cross-sections, and allow you to create standard cross-section property sets that you can use over and over again in many different models. The libraries includes products of manufacturers worldwide.

For the description of the Table Browser see 2.9 Table Browser.



The Undo function does not work when libraries are modified.

Create a new library

You can create a custom cross-section library by the *File / New Cross-Section Table* command in the Table Browser. You have to specify library name, library file name and a cross-section type. Standard and custom cross-section library files (*.sec) are stored in the folder where the application is stored.

Assign a name to each cross-section, and specify the following properties:

Name	
Fabrication process	Rolled, welded, cold-formed, other.
Shape	I (H, W), U, L, Pipe, Round, Rectangle, C, Z, S, J, T, Box, Custom

Cross-section properties

When creating a new cross-section in the table all property values have to be entered.

Axial (cross-sectional) area
Shear area associated with shear forces in local 1 st direction
Shear area associated with shear forces in local 2 nd direction
Rounding (corner and fillet) radii
Torsional inertia
Flexural inertia about local y axis
Flexural inertia about local <i>z</i> axis
Centrifugal inertia
Principal inertia about local 1 st axis
Principal inertia about local 2 nd axis
Warping modulus (used for the design of steel shapes)
Elastic cross-section modulus, top = $I_1/e2_{max}$ (see diagram below)
Elastic cross-section modulus, bottom = $I_1/e2_{min}$
Elastic cross-section modulus, top = $I_2/e1_max$
Elastic cross-section modulus, bottom = $I_2/e1_{min}$
Plastic cross-section modulus
Plastic cross-section modulus
Radius of inertia about local 1 st axis
Radius of inertia about local 2 nd axis
Dimension in the local <i>y</i> direction (width)
Dimension in the local <i>z</i> direction (height)

Ø

yс	Position of the center of gravity of the cross-section in local <i>y</i> direction relative to
	the lower-left corner of the circumscribed rectangle
ZG	Position of the center of gravity of the cross-section in local <i>z</i> direction relative to
	the lower-left corner of the circumscribed rectangle
$\mathbf{y}_{s\prime}\mathbf{z}_{s}$	Position of the shear center in local \mathbf{y} and \mathbf{z} directions relative to the center of
	gravity
S.p.	Stress calculation points

- (*) If first and second principal axes are the local y and z axes values with (*) appears with indices y and z.
- *Table properties* Custom library properties can be modified by the File / Cross-Section Table Properties command in the Table Browser. Custom library properties can be deleted by the File / Delete Cross-Section Table command in the Table Browser.
- *Import/Export* You can import and export numerical values in libraries as dBaseIII files by File / Import values dBase file.
- *Copy/Paste a* You can copy and paste cross-sections with their full graphical description within the Table cross-section Browser. Numerical data exchange with other applications is supported via clipboard.
- You can add a new cross-section to any custom or standard library by *Edit / New Row* (or by Add/Modify / *Delete a cross*pressing [CTRL+INS] or the toolbar button) in the Table Browser and entering field values.
- section

You can also call the Cross Section Editor to specify cross-section data. Use Edit / Design New Cross-Section (or [CTRL+G]) to add a new cross-section and Edit / Modify Cross-Section (or [CTRL+M]) to modify an existing one.

Changing any dimension of a standard shape AxisVM automatically recalculates all crosssection parameters and updates the graphics.

You can delete a cross-section with the aid of deletion icon or by pressing [CTRL+Del]. See description of the cross-section editor in section 3.1.14.1.

P Cross-section libraries contain the values of the warping inertia Io used in the Steel Design module.

The property values in standard libraries are taken from manufacturers' databases. You must verify them before use.

Table Browser										_ [×
<u>File E</u> dit F <u>o</u> rmat <u>R</u> eport <u>H</u> elp											
Cross-Section Libr		四 + ×	Ē		8	3	2				
- AISC M Shapes		European I-beai	ms								_
- AISC S Shapes - AISC W Shape:		Name	Drawing	Process	Shape	h [mm]	b [mm]	tvv [mm]	tf [mm]	A× [mm ²]	1
HD wide flange HE European v HL_EU.sec	31	IPE A 270	<u>İ</u>	Rolled	I	267,0	135,0	5,5	8,7	3915,00	
- HP wide flange - I Hungarian I-k - I Romanian I-b - IPE European I - IPN European S - UB British univ	32	IPE A 300	- <u>†</u>	Rolled	I	297,0	150,0	6,1	9,2	4653,00	
	33	IPE A 330	Ì	Rolled	I	327,0	160,0	6,5	10,0	5474,00	
UC British Univ U Channels Angles Double Angles	34	IPE A 360	- <u>Ť</u>	Rolled	I	357,6	170,0	6,6	11,5	6396,00	-
]								Þ	
Editing IPE 80, Cross-Section Nam	e										
								ок		Cancel	

The table below shows the shape and reference coordinate system of the cross-sections. The properties that were not published by the manufacturers were calculated.

Cross-sections

Steel cross-section



In the calculation of cross-section properties and displaying the cross-section the rounding (corner and fillet) radii (r_1, r_2, r_3) are also taken into account.

The explanation of the these radii, height, width, wall-thicknesses and diameters can be seen in the schematic diagrams below.













94		
3.1.14.1.	Cross-Section Editor	

I	The Cross-section Editor allows you to edit thin and thick walled cross-sections. You can use parametric circular, rectangular, ring and polygonal shapes, or any shape listed in the cross- section libraries to edit composite cross-sections. The shapes used to build a new cross- section are referred to as components, and have to be of the same material. You can translate, rotate, mirror, copy or move the selected components at any time during the editing. When a component is placed to its location graphically, the principal axes and the cross-sectional properties of the composite cross-section are computed. You can use keyboard commands the same way as in main editing windows.
	The OK button exits and closes the cross-section editor window, and saves your current cross-section into the cross-section table of your model with a name you specify.
	Cross-section editor is on the toolbar of the Cross-section Library and can also be launched from the line element dialog. See 4.9.7 Line Elements The editor can be used when creating a native model from an architectural model through the IFC interface. See 4.9.20 Creating model framework from an architectural model
Editor Keys	See 2.5 Using the Cursor, the Keyboard, the Mouse
Toolbar	Most important functions are available from the toolbar.
G.	Prints the cross-section. See 3.1.10 Print
	Adds the image of the cross-section to the Gallery. See 3.2.10 Saving drawings and design result tables
ŝ	Undoes the last operation.
2	Redoes the operation which was undone.
	Copies the image of the cross-section to the Clipboard.
From Cross- section Library	Loads a cross-section from the Cross-section Library. Only thick or thin-walled cross- sections are available depending on the cross-section editor tab position.
From DXF file	Contour of thick walled cross-sections can also be imported from a DXF file.
Stress-points	You can specify the points you want to calculate stresses for. The default stress-point is the center of gravity. You can specify up to 8 stress-points for each cross-section. When applying a move command the stress-points can also be moved.
Ē	Stress calculations are performed at the specified stress-points only. If you don't specify any stress-points, stress will be calculated in the center of gravity only. It means that no bending stress will appear.

lcon bar

Editor functions and settings can be found on the Icon bar on the left. The behaviour of the Icon bar is the same as that of the main Icon bar. **See...** 2.15 Icon bar.

The only difference is that this Icon bar can be moved above the menus at the top or at the bottom but it is not dockable.



Thin-walled crosssections



A component belonging to the thin-walled category can be added to your cross-section.

Base-point You can select a base-point to each cross-section component, that allows you to position the component during editing, depending on its shape and final location within the composite cross-section.

Standard shapes can also be defined parametrically. In this case the following parameters has to be defined in the dialog:

- Manufacturing There are three options (rolled, welded, cold formed.)
 - *Dimensions* Values depending on the type of the cross-section (height, width, thickness, corner/fillet radius, diameter etc.).
 - *Rotation* Lets you define a rotation by angle α . The default value is 0.



process

Definition of an I or wedged I shape by its height, width, web and flange thicknesses and a fillet radius.

Wedged I shape





AXIJVM1()

Asymmetric I Definition of an asymmetric I shape shape by its height, width, web and upper / lower flange dimensions.



b [cm] = 70

v [cm] = 8 **Rotation** α. [°] = 37

Place

Cancel

RectangularDefinition of a rectangle by its
parameters b (width), v (thickness),
and α , with b>v.

Definition of a pipe by its parameters d (outside diameter), and v (thickness). The centerline is considered as the contour of a closed domain, which is displayed with a dashed line.

Element



Pipe

Definition of cross-sections by height, width, thickness and in the case of rolled or bended cross-sections by the corner/fillet radius.



The base cross-section can be defined parametrically (width, height, web and flange thickness) or taken from the Cross-section Library. Special parameters for double shapes:

distance: a

orientation : facing or back-to-back (in case of 2U)

Polygonal Definition of a polygonal shape.

Before the definition the position of the control line of the segment can be selected:

- 1. left side
- 2. center line
- 3. right side
- R parameter : Rounding (corner and fillet) radii



Arc shape Definition of an arc shape by its diameter, central angle and thickness.



Changing wall thickness

Change wall thickness	×
Thickness [cm] =	1,0 💌
ОК	Cancel

Deletes the selected stress-points.

For thin-walled cross-sections thickness of selected segments can be changed individually. For parametric shapes wall thickness can be changed through the parameters.

Delete

Using the **[Del]** key you can invoke the Selection Icon Bar, and select the components you want to delete.

When deleting a component the stress-points will also be delete.

Stress-point

You cannot delete the default stress-point (the center of gravity).

Options Lets you set the grid size, cursor step, and the zoom factors.

Thick-walled crosssections Cross - 🗆 🛛 Section Editor - New Cross-Secti <u>File Edit Display Window</u> 🖨 📳 🗠 👻 🖓 Thin Thick 🥝 📽 I Ē × \mathbb{R} Ax[mm²]= 966746910, 966746910, Iy[mm^a]= Ð Iz[mm⁴]= Iyz[mm⁴]= y_G*[mm]= z_G*[mm]= 500045083, Q 23 ²G [.....] I1[mm^a]= 1.46679196E9 466701807,0 I2[mm⁴]= ÷¢ a[°]= i1[mm]= 45.00 135, is[mm]= 76. More parameters ĂΔ Â 2 €€\$#G@@• F OK Cance

Rectangular

Definition of a rectangle by its parameters b (width), h (height), and α .

Circular, Semicircular Definition of a circular or semicircular shape by its diameter and $\boldsymbol{\alpha}.$



Definition of an I shape by its parameters a1, a2, a3, b1, b2, b3, and α . (a1, a3), (b1, b3). Parameters can be set to 0, allowing the creation of T, U, L shapes.

98	10
Polygonal	Definition of a polygonal shape by drawing a polygon.
Insert a vertex	Insertion of a new vertex on the contour of the cross-section. Shape of the cross-section can be changed by dragging a vertex by the mouse.
Contour	If the Contour button is down the cross-section can be defined. If the Hole button is down a hole can be specified.
Hole	You can specify a hole in rectangular, circular, and closed polygonal shape components. The hole can be rectangular, circular, and closed polygonal.
Delete	Using the [Del] key you can invoke the selection window, and select the components you want to delete. When deleting a component, the stress-points will also be deleted.
Polygon	Deletes the selected components.
Stress-point	Deletes the selected stress-points.
Ē	You can not delete the default stress-point (from the center of gravity).
Options	Lets you set the grid size, cursor step, and the zoom factors.

Compute properties

Following cross-section properties are calculated:

AxisVM calculates Ax, Iy, Iz, Iyz by integration, Ay, Az, Ix, I ω , ρ_y , ρ_z , ρ_{yz} , ρ_1 , ρ_2 , A1, A2 by performing a finite element analysis of the cross-section.

^{**} In case of a cross-section consisting of two or more independent parts, $A_{y'}$, $A_{z'}$, $\rho_{y'}$, $\rho_{z'}$, $\rho_{yz'}$, $\rho_{y'}$, $\rho_{z'}$

A _x	Axial (cross-sectional) area
A _v	Shear area in local <i>y</i> direction
Az	Shear area in local <i>z</i> direction
I x	Torsional inertia
$\mathbf{I}_{\mathcal{Y}}$	Flexural inertia about local y axis
\mathbf{I}_z	Flexural inertia about local <i>z</i> axis
\mathbf{I}_{yz}	Centrifugal inertia
$I_1^{(*)}$	Principal inertia about local 1 st axis
$I_2^{(*)}$	Principal inertia about local 2 nd axis
α	Angle between local 1 st axis and the local y axis.
\mathbf{I}_{ω}	Warping modulus (used for the design of steel shapes)
ρ _y	shear factor in local y direction
ρ_z	shear factor in local y direction
ρ_{yz}	shear factor for local <i>yz</i> cross
$\mathbf{\rho}_1$	shear factor for local 1st direction
$\mathbf{\rho}_2$	shear factor for local 2nd direction
$A_1^{(*)}$	Shear area associated with shear forces in local 1 st direction
$A_2^{(*)}$	Shear area associated with shear forces in local 2^{nd} direction
${\bf W}_{1, el, t}^{(*)}$	Elastic cross-section modulus, top = $I_1/e2_max$ (see diagram below)
$\mathbf{W}_{1, el, b}^{(*)}$	Elastic cross-section modulus, bottom = $I_1/e2_{min}$
$W_{2,el,t}^{(*)}$	Elastic cross-section modulus, top = $I_2/e1_max$
$W_{2,el,b}^{(*)}$	Elastic cross-section modulus, bottom = $I_2/e1_{min}$
$W_{1/pl}^{(*)}$	Plastic cross-section modulus
$W_{2,pl}^{(*)}$	Plastic cross-section modulus
\mathbf{i}_1	Radius of inertia about local 1 st axis
i ₂	Radius of inertia about local 2 nd axis

УG	Position of the center of gravity of the cross-section in local y direction relative to
	the lower-left corner of the circumscribed rectangle
z _G	Position of the center of gravity of the cross-section in local z direction relative to
	the lower-left corner of the circumscribed rectangle
y _s , z _s	Position of the shear center in local y and z directions relative to the center of
	gravity
Po	Outer circumference (cross-section contour)
P _i	Inner circumference (holes)

(*) If first and second principal axes are the local *y* and *z* axes values with (*) appears with indices *y* and *z*.



Principal inertia

/ ₁	$I_{1} = \frac{I_{x} + I_{y}}{2} + \sqrt{\left(\frac{I_{x} - I_{y}}{2}\right)^{2} + I_{xy}^{2}}$
I ₂	$I_{2} = \frac{I_{y} + I_{z}}{2} + \sqrt{\left(\frac{I_{y} - I_{z}}{2}\right)^{2} + I_{yz}^{2}}$
α	$tg(2\alpha_n) = \frac{2n_{xy}}{n_x - n_y}$

 $-90^{\circ} < \alpha \le +90^{\circ}$, relative to the cross-section's local *y* axis.

Calculation of elastic cross-section modulus

Shear deformations

For beam elements the shear deformations are not taken into account even if the crosssection was entered with nonzero for the shear area.

The shear areas are used by the rib element and must be positive nonzero values ($A_y \neq 0$ and $A_z \neq 0$).

In the steel design module, the shear areas are calculated according to the corresponding design code, instead of using the values entered here.

 ρ = shear factor

Where: $A_y = \frac{A_x}{\rho_y}$ $A_z = \frac{A_x}{\rho_z}$

3.1.15. Exit

[Ctrl]+ [Q] Exits the program.

3.2. Edit

<u>F</u> ile	<u>E</u> dit	t <u>S</u> ettings <u>V</u> iew <u>W</u> indow <u>H</u> elp	
	K)	Undo Modify Supports (RYY=1E+07 kNm	/rad) Ctrl+Z
	$\bigcirc I$	<u>R</u> edo	Shift+Ctrl+Z
	*	<u>S</u> elect All	Ctrl+A
	Þ	<u>С</u> ору	Ctrl+C
	Ē.	<u>P</u> aste	Ctrl+V
		C <u>o</u> py/paste options	
	${}^{\times}$	<u>D</u> elete	Del
			F12
	1	R <u>e</u> port Maker	F10
	鑦	Add drawing to Gallery	F9
	ů	<u>W</u> eight Report	F8
	4	Asse <u>m</u> ble structural members	Ctrl+F7
	4	Break apart structural members	Shift+Ctrl+F7
		Co <u>n</u> vert surface loads distributed over b	eams
		Con <u>v</u> ert automatic references	

3.2.1. Undo



6	5:22 PM Define Domains (A	LUMINIUM; Plates; 4.0 cm)
	5:22 PM Define Line Elemer	it beam; HP 10X42; ALUMINIUM
	5:22 PM Define Line Elemer	it beam; HP 14X117; ALUMINIUM
	5:22 PM Generating Rectar	gles (3.300 m x 1.500 m)
	5:19 PM Create Line (2x)	

Undoes the effect of the previous commands. To undo a sequence of actions (more levels), click the down arrow next to the Undo icon, and then select the actions you want to undo based on the time or type of the commands.

You can set the number of undo/redo levels (maximum 99) in the *Main menu/Settings* dialog box.

3.2.2. Redo

[Shift]+[Ctrl]+[Z]

C^a

C4 🔻	5:19 PM Create Line (2x) 5:22 PM Generating Rectangles (3:300 m x 1:500 m) 5:22 PM Define Line Element beam; HP 14X117; ALUMINIUM 5:22 PM Define Line Element beam; HP 10X42; ALUMINIUM
	5:22 PM Define Domains (ALUMINIUM; Plates; 4.0 cm)

Undoes the undo command or goes forward to reverse one or more undo commands. You can select the actions you want to redo based on the time or type of the commands.

3.2.3. Select All

See... 2.15.1 Selection [Ctrl]+ [A]

3.2.4. Copy

Ð

[Ctrl]+ [C] Copies the selected elements of the model to the Clipboard. If nothing is selected but there are active parts, active parts are copied. If neither selection nor active parts are present the entire model is copied.

This function copies the drawing of the current graphics window to the clipboard like in earlier versions but this operation can be deactivated.

3.2.5. Paste

Ê.

Pastes AxisVM elements from the Clipboard. For paste options see Copy / paste options. [Ctrl]+ [V]

3.2.6. Copy / paste options

Copy options Selected elements are always copied to the Clipboard. User-defined parts containing the selected elements are also copied.

If domains, beams, ribs, trusses are copied certain associated objects (supports, loads, dimension lines, reinforcement domains) are also copied.

If you want to control which associated objects should be copied select them and choose one or more of the following options: *Selected supports / Selected loads / Selected dimensions / Selected reinforcement domains*.

Load cases are copied with loads. If you want to copy all load cases choose *Copy all load cases* instead of *Copy load cases of the loads copied*.

Load combinations and load groups can also be copied.

Turn on *Copy active window as a drawing* to copy the active window as graphics as well (it was the only option in earlier versions).

Paste options Load cases

Pasting of load cases can be controlled with the following options: *Paste as new load case:* load cases found on the Clipboard are copied as new load cases. If *Merge load cases with the same name* is turned on and the model has load cases with the same name as the clipboard load case these load cases will be merged (loads of the clipboard load case will be added to model load case). This option must be turned on when copying within the model to avoid creating unnecessary load cases.

Copy/paste options
Copy Paste
Copy associated objects
Supports Selected supports
Loads Selected loads
Dimensions Selected dimensions
Reinforcement domains Selected reinforcement domains
Copy load cases of the loads copied
C Copy <u>a</u> ll load cases
Copy all load combinations
🔽 Copy all load groups
Copy active window as a drawing
Cancel

Copy/paste options 🛛 🗙
Copy Paste
Load cases
Paste as new load case
Merge load cases with the same name
C Merge loads from all load cases into the current one
Parts
C Paste into all active parts
Paste into the original parts
Paste position
C Paste into original position
C Drag by the relative origin
• Drag by a corner node of the structure
Cancel

Merge loads from all load cases into the current one. This option copies all loads from all clipboard load cases into the current load case of the model.

Parts

User-defined parts containing the selected elements are also copied to the clipboard. The first option is to paste elements of parts into all active parts of the model. The second option is to paste the parts themselves.

Paste position

There are three options.

Paste into original position: pasted elements will get into their original coordinate position. *Drag by the relative origin / Drag by a corner node of the structure:* If one of these options are selected paste position can be defined by clicking the left mouse button. In the first case the clicked position will become the position of the relative origin in the source model when the elements were copied. In the other case the clicked position will become the position of an automatically identified corner of the copied structure.

3.2.7. Delete

х

[Del]

Deletes the selected entities. If no elements are selected it brings up the Selection icon bar and then the Delete dialog window.

Lets you delete the selected geometric entities. To delete:

- 1. Select the geometric entities to be deleted. You can select them by holding the [Shift] key pressed while you click on the entities with the left mouse button or use the Selection Icon Bar.
- 2. Press the **[Del]** key. If there is no selection, the selection toolbar appears and objects can be selected for deletion. **See...** 2.15.1 Selection.
- 3. Enable the check-boxes of the entities you want to delete.
- 4. Press the **OK** button, to finish and close the dialog window.

In the dialog window the check-boxes are active or inactive according to the contents of the current selection set (intended for deletion).

Seometry	Loads	R.C. Design	
Node (314) Line (796) Surface (482)	X Nodal Force (1) X Concentrated Force (2) X Load on Line (12)	Reinforcement param. (486) Actual Reinforcement (4)	
lements	X Load on Surface (2)	Design parameters	
X Line Elements (31) X Surface Elements (462) X Supports (24) X Rigid Element (1) X Diaphragm (1) X Spring (1) X Spring (1) X Edge (1) X Link (1)	Formation Length (1) Fault In Length (1) Tension (7) Seft Weight (498) Support Displacement Nodel acceleration Influence Line Tensioning Moying load	Steel design parameters (11) Timber design parameters (11) Botted joint Oimensions X Orthogonal dimension line (1) X Aligned dimension line (1) X Angle Dimension (1) Aligned Dimension (1)	
X Domain (4) X Hole (1) Reference	X Surface mesh (4) X Line mesh (2)	Level Elevation (1) T text box (1) Object info fresult text box. Object info text box (2) I isoline label	

Geometry	Lets you select geometric entities for deletion. Deleting geometric entities that have assigned finite elements, will result in the deletion of its finite elements and of the associated loads.			
Elements	Lets you select finite elements for deletion. Deleting finite elements will not delete the respective geometric entity, but will delete the loads.			
References	Lets you select references for deletion. All finite elements that use the deleted references, and the associated loads will be deleted too.			
Mesh	Lets you remove mesh from domains.			
R.C. Design	Lets you select the reinforcement parameters attached to the selected elements for deletion. Footing parameters are also deleted.			
Steel / Timber design	Lets you select the steel / timber design parameters attached to the selected elements for deletion.			
Dimensions	Lets you select the dimension lines, text boxes etc. for deletion.			

3.2.8. Table Browser

[F12] See... 2.9 Table Browser

3.2.9. Report Maker

[F10]

[F9]

锢

88888

See... 2.10 Report Maker

3.2.10. Saving drawings and design result tables

Add drawing to

Gallery

You can save drawings from AxisVM in many different contexts: you can save AxisVM main windows, beam displacement and internal forces diagrams, steel design results, nonlinear calculation results, reinforced concrete column and beam design diagrams, bolted joint diagrams. In case of a divided view you can select to save all windows or the active one only.

Trawings Library is another way to store diagrams. While Gallery contains static image files, the Drawings Library uses associative drawings following changes in the model.

See... 2.13 Drawings Library

Which file formatBitmap formats (.BMP, .JPG) store the pixels of the diagram, so Windows metafiles provide
higher resolution when printed. JPG is a compressed format with a slight loss of quality but
these files are much smaller than BMPs.

Windows metafiles (*.WMF*, *.EMF*) store a series of drawing commands so they can be scaled and printed in any size in the same quality. However if you choose hidden line removal or a rendered view drawn by OpenGL technology metafiles will contain only bitmaps. To get a high resolution rendered view print the picture directly.

Drawings will be saved to a subfolder *Images_modelname* automatically created under the folder of the model file. These pictures can be inserted into a report. Do not modify the name of the subfolder *Images_modelname*.

3.2.11. Weight Report

(F8]

The weight of the entire model, selected elements or details can be listed in tabular form per material, per cross-section or surface type.



3.2.12. Assemble structural members

4

AxisVM handles line elements as structural members. It means that *Meshing of line elements* on the *Mesh* tab creates finite elements but the line elements themselves are not divided. The *Find structural members* menu command joins adjacent line elements into a single element until a breaking point is found. A breaking point is defined by different local x or z directions, different material, cross-section or eccentricity, end release or a domain boundary. Line elements must be on the same line or on the same arc.

3.2.13. Break apart structural members

4

The *Break apart structural members* menu command breaks apart line elements created with the *Assemble structural members* command.

3.2.14. Convert surface loads distributed over beams

This menu item converts selected surface loads distributed over beams into individual distributed beam loads.

3.2.15. Convert automatic references

This menu item converts automatic references assigned to line or surface elements into reference vectors.

3.3. Settings



3.3.1. Display



3.3.2. Options

Eile

Ľ

;	<u>E</u> dit	Set	tings	⊻iew	<u>W</u> indow	<u>H</u> elp		
		69	<u>D</u> ispl	ay Opti	ons	٠.	1	
		Ľ	<u>O</u> ptio	ns			雔	Grid & Cursor
		∅	Laye	r Mana	ger	F11	\mathbb{Q}	<u>E</u> diting
		^{20,00}	<u>S</u> tori	es		F7		<u>⊻</u> iews…
		4	<u>G</u> uide	elines S	etup	Ctrl+G	Γ	
		\bigcirc	D <u>e</u> siç	an Code	as			
			<u>U</u> nits	and Fo	rmats			
			G <u>r</u> av	itation.				
		R	<u>P</u> refe	rences		×		
		27 55 55 27	Lang	uage		•		
		2 <u>2 55</u> 55 22	Repo	rt Lang	uage	•		
			Tool	ars to	default po	sition		

See... 2.15.14. Options





	Layer Manager		
:11]	AxisVM layers	Properties Colour	New Axis∀M layer
	Structural layers		Delete
	·····································	Line style	Delete empty AxisVM layers
	w rk contract Elements 	Line weight	Delete empty DXF layers
	in the supports in the support of t	0,00 mm _	
	in ♥ ▲ 8. Floor in ♥ ▲ 7. Floor in ♥ ▲ 6. Floor	Apply to all	
	Show full path	Visible	ОК
	I Refresh All	Z Layer detection	Cancel

The Layer Manager allows you to manage AxisVM layers, imported DXF or ArchiCAD layers. While only one ArchiCAD layer can be imported, multiple DXF layers are allowed. If no AxisVM layers are defined AxisVM automatically creates a new layer for dimension lines with the name *Dimensions*.

On the left side of the Layer Manager dialog a tree view of the available layers is displayed. If you select (highlight) a DXF layer in the tree, you can modify its properties in the right side (Name, Color, Style, Size). If you select the main DXF file entry of the tree, you can modify all the DXF layers at a time. Properties of AxisVM structural layers cannot be modified.

Apply to All: When using this button, a dialogue window will allow you to select the items in the DXF layers that will have their properties set based on the layer's settings.

The visibility of the layers or DXF files can also be set by clicking on the bulb or cursor symbol next to the layer or file name.

New AxisVM Creates a new AxisVM layer. You can set the layer's name, color, line style, and size.

Delete More than one layer or group can be selected and deleted by the [Del] key.

Delete Empty Deletes all AxisVM layers that are empty (contain no entities).

AxisVM Layer

Layer

Delete Empty Deletes all imported DXF layer that are empty (contain no entities).

DXF Layer

3.3.4. Stories

🔽 [Ctrl] + [R]



Stories are to make it easier to overview and edit the model. They can be defined before building the model or assigned to an existing structure.

A story is a workplane parallel to the global X-Y plane, with a given Z position. If a story is selected mouse movements will be projected to the plane of the story even if you find an element at a different Z position. Coordinates will always be projected to the story plane to help tracing objects at different levels.

Stories are always listed by decreasing Z position, having automatic names. Changing the report language makes story names change.

Elements are considered to be part of a story if their lowest Z coordinate is greater than or equal to the story level but less than the next story level. Therefore if a multi-story column or wall was defined as a single element it will appear only at the lowest level. To change this behaviour the element has to be cut with story planes. See ...

New elements will be linked to their story automatically.

Stories are logical parts of the model created for editing purposes and they do not affect the analysis results.

If torsion effects has to be taken into account in seismic analysis seismic stories have to be defined separately in the seismic parameters dialog.

Stories can be managed in the following dialog.

👕 Stories		×			
	\$x 99	1			
Z [m] = 0	+ +				
Active story Grour	nd floor				
Stories	Z[m]				
Story 3	+8.850				
Story 2	+5.850				
Story 1	+2.850				
 Ground floor 	-0.150				
Story -1	-3.150				
🔽 Auto Refresh					
🦳 <u>R</u> efresh All					
ОК	Cancel				



Display current

If this button is down no stories are displayed. Windows will show the entore strucutre or the its active parts. Stories can be added or deleted in this state as well.



If this button is down and an active story is chosen the active story will be displayed. The active story can be chosen by clicking the radio button before its name.

Selection status of the list items is independent of this choice. More than one story can be selected. Ctrl+click adds individual list items to the selection, Shift+click adds ranges to the selection. Delete operation works on the selected stories and not on the active story.

- There can be only one active story. However display of neighbouring stories is also possible. Editing will be constrained to the active story.
- *Pick up*Click this icon to get back to the model and click one or more nodes to pick up Z coordinates. Close the process by clicking on an empty area. Z coordinates will be added to the list of stories.

Enter a new story

y Enter the Z coordinate into the edit field and click the + button. A new story will be added to the list.



If you have an existing multi-story structure with slabs you can find and add Z coordinates of horizontal domains to the list with one click. If not all horizontal domains refer to a real story you can delete unnecessary stories later.

Story position cannot be changed. Delete the story and define a new one.

turned on. Choosing a new active story overrides the parts settings.

Deleting a story does not delete any element.

tracing other objects.

tracing other objects.

Delete

te Deletes selected stories. Remaining stories will be renamed and story assignments of the elements will be updated automatically.

If this button is down elements of the story below the active story is also displayed to help

Ŧ

Display the story below the current



Display the story

above the current story



æ

To display further stories open the Parts dialog instead where logical parts of any story can be

If this button is down elements of the story above the active story is also displayed to help



stories

Numbering of stories can be controlled with these buttons. If the left one is down (*Numbering of stories from the bottom*) the lowest floor will be considered as ground floor and other stories will have a positive number. If *Signed numbering of stories* is selected the story closest to the zero level will be the ground floor. Underground stories will get a negative number, others will get positive numbers.

3.3.5. Guidelines



Guidelines [CTRL]+[G] See... 2.15.7 Guidelines

3.3.6. Design Codes



Sets the Design Code to be used in case of code specific tasks. Changing Design Code changes the method of calculating critical load combinations therefore all load group parameters but partial factors will be deleted. Seismic analysis parameters and seismic load cases will also be deleted. As material properties and certain reinforcement parameters are not the same in different codes it is recommended to revise the values you have specified.

If *Set current settings as default* is checked, new models will be created with the current design code.

3.3.7. Units and Formats



Lets you configure the units (SI and/or Imperial) and formats of variables used throughout the program (number of decimals used for displaying or exponential format). You can use predefined sets as the SI set, or create and save your own custom sets.

3.3.8. Gravitation

Gravitation	×
Direction	
C +X	○ -X
C +Y	O -Y
C +Z	• -Z
Gravitational # g (m/	Acceleration
ок	Cancel

Lets you set the gravitational acceleration constant and the direction of gravitation as one of the global coordinate directions
3.3.9. Preferences



Recent file list Lets you set the number of recently opened AxisVM model files listed in the bottom of the File menu, and set if you want the last edited file to be opened at startup. The welcome screen (**See...** 2.2 Installation) will be shown on startup if the *show welcome screen on startup* checkbox is checked.

Save Auto Save option

To make sure that you do not lose your work, select the *Auto Save* option by the check box. In the Minutes box, enter the interval at which you want to automatically save the opened model (1-99 minutes). You must still save the model when you exit. A model that is saved automatically is stored in the default temporary folder of the operating system (by default it is *c*: \ *Documents and Settings* \ *username* \ *Local Settings* \ *Temp*) as ~*modelname.avm* until you perform a save command. When you have to restart AxisVM after a power failure or due to any other problem that occurred before you saved your work, AxisVM can recover it from the temporary file stored in the above folder under the name \$modelname.avm.

Create Backup Copy

If this checkbox is checked and a model is saved after making changes a backup copy is automatically created from the previous state of the axs file. Name of the backup file is *modelname*. $\sim AX$.

Save derivative results

If this checkbox is checked stresses, envelopes, critical combinations and design results will be saved as well.

- *Undo* You can undo your last actions. You have to specify the maximum number of actions you want to undo. This number must be between 1 and 99.
- *GroupUndo* The Group Undo option allows you to undo the effects of complex commands in a single step. Undo data can be stored in memory or on hard disk. The first option is faster, the second option leaves more memory for the program (it may be important if a huge model is calculated).

Work on localIf models are opened through a network, the speed of data transfer may reduce the per-
formance of AxisVM. This effect can be eliminated by allowing making local copies of net-
work files. Local copies will be placed into the folder where the temporary files are stored
during the analysis – except when this folder is set to the model folder. In this case the files
are saved to the default folder for temporary files. The original files will be updated at each
save operation.

Network time-out In case off network hardware protection keys, if in a time period set here there is no activity (checks) with the key, the current AxisVM session is closed.

Disconnecting may also happen in a situation when you get a phone call and you do not use the program for a time longer than the network time-out. If another user asks for access to the key the server gives a license to him/her and when you try to continue your work the program displays an error message and halts at the next key check.



Lets you select graphics area background color (black, dark gray, light gray or white). Labels, numbers, symbols and elements will automatically change their colors to remain visible

	Fonte	
Data Integrity	Drawing Labels	
Colors	Brial 40 mt	
Fonts	Anaritopi	
e Edit		
Meshing	Drawing Labels	Default Settings
Toolbar		
Display	Information Windows	
Analysis	Verdana 7 nt	
Report	verdana r pr	
i Update	To formation total datas	
	Information windows	Default Settings
	Dialog boxes	
	Arial 8 pt	
	Dialog boxes	Default Settings
		Dejadit Settings
		Preferences
		· · · · · · · · · · · · · · · · · · ·
		OK Cancel

Lets you change the typeface and size of the fonts that are used when displaying your model and the Floating Palettes. Click white sample area to get to the font selection dialog.

Default settings can be restored by pressing the button on the right.

Colors

Fonts

Edit



Circle Closing Parameter for drawing arcs. If the center angle of the arc is smaller than this angle or it is *Angle* closer to 360° than this angle then a whole circle will be drawn.

Projection line toDisplay of projection lines can be turned on/off. Its shows the distance of the cursor from
workplaneworkplanethe current workplane.

Turn on logical parts when loading models from previous version If turned off no logical parts will be created for older models.

- *Move mouse pointer automatically to dialog windows.* If turned on mouse pointer will jump to the OK button of dialog windows.
- Include internal lines of domains into parts dy default

If turned on internal lines of domains will be included into parts containing the domain.

Elements of a hidden mesh can be selected

If display of mesh is turned off this field controls if the hidden nodes / lines / surface elements can be selected or not. This switch also controls if these nodes and elements appear in tables or not.

Delete unnecessary contour lines after automatic domain intersection

Controls if contour lines are automatically deleted after domain intersection. If this function is turned off contour lines became internal lines of the union.

Meshing

Preferences	X
Data Integrity Colors Fonts Edit Meshing Toolbar Display Analysis Report Update	shing Vesh management

One of the following mesh management methods can be chosen.

Mesh management

Remove and create mesh automatically

Any editing performed on a domain deletes its mesh. When launching the analysis missing meshes will be recreated based on the meshing parameters of the domain.

Keep mesh editable Meshes can be edited manually.

Contour division	Uniform mesh size
method	Meshes will be generated according to the user defined element size regardless of the shape of the domain (least number of finite elements).
	Adaptive mesh size
	Takes the shape of the domain into consideration and creates a better mesh by increasing mesh density wherever it is necessary.
Default mesh size	When defining meshing parameters for a domain for the first time this value will appear by default.

Toolbar

 Data Integrity 	Toolbar
Colors Fonts Edit Meshing Toolbar	e Horizontal toolbars expanded ✓ ✓ □ □ □ □
 Display Analysis Report Update 	C Elyout toolbars ∕, ⊙, +/ ≯ 囲, / > / □ ♪
	Pet palette position \bigcirc Relative \bigcirc Appear in the latest position $dx = \begin{bmatrix} 60 \\ by \\ c \\ dy \\ c \\ by \\ c \\ c \\ dy \\ c \\ $

Displaying toolbar If *Horizontal toolbars expanded* is chosen, all icon appears in a row. Separator lines indicate different groups of functions.

If *Flyout toolbars* is chosen, different functional groups will be represented by a single icon. Clicking the arrow in the right bottom another toolbar flies out showing different tools.

Pet palette Pet palette position can be:

position Relative

Specify the horizontal (dx) and vertical (dy) distance from the operation in pixels. *Appear in the latest position*

Pet palette appears in its latest position.

Display

 Data Integrity Colors 	Displav Moment diagram	
Fonts		3 - 1
Edit		
Meshing		3 1
Toolbar	On tension side	C On compression side
Display		
Analysis Report	Arc resolution	
Neport		
opuate	coarse	fine
	Turn on 3D wireframe when	a drawing objects
	J♥ Tarron 3D wirename wher	r drawing objects
	Display of line loads on all c	onnecting elements
	, <u></u> _, <u></u> , <u></u> _, <u></u> , <u></u> _, <u></u> , <u></u> _, <u></u> , <u>_</u> , <u></u>	
	Model graphics style	
	C Classic (for lower scree	n resolution)
	AxisVM10 (for higher sc	reen resolution)

Moment diagram

Placement rule for moment diagrams can be set.

Arc resolution Arcs are displayed as polygons. Set the display resolution here. The finer the resolution the closer the polygon will get to the arc. This parameter affects drawing only and is not related to the precision of the analysis.

Model graphics style

classic style is recommended on systems with a lower resolution. *AxisVM10* style draws line elements with thicker pen, automatically fills domains and display surface loads as a hatched pattern.

Switches Turn on 3D wireframe when drawing models

Displays 3D wireframe of objects while drawing (see... 4.9.3 Direct drawing of objects) even if the active view is not in rendered mode.

Display of line loads on all connecting elements

If an edge load is applied where a wall and two plates meet and parts are turned on (**see..** 2.15.11 Parts) the load will be displayed according to this setting. If this option is turned on the load will be displayed if an active part contains any of the three elements. If this option is turned off the load will be displayed only if an active part contains the elements the loads were originally assigned to. This is useful to check the local system of the load components.

Analysis

Data Integrity Colors Colors	al memory used in a ory: ry: ble memory block: ended memory acce	Available 1152 M 1710 M 807 M	1500 M , , , , , , , , , , , , , , , , , , ,
Toolbar Display Analysis Update Update Toolbar Display Physical memo Largest avail. Folder for t C Mode C Local	ory: ny: ble memory block: ended memory acce	Available 1152 M 1710 M 807 M	7otal 2047 M 2047 M
Toolbar Display Analysis Report Update Folder for t C Mode C Local	ony: ny: ble memory block: ended memory acce	1152 M 1710 M 807 M	2047 M 2047 M
Display Analysis Report Update Update Virtual memo Largest avail. Folder for t C Mode C Local	ny: ble memory block: ended memory acce	1710 M 807 M	2047 M
Annupans Report Update Folder for t C Mode C Local	ble memory block: ended memory acce	807 M	
Update Folder for t C Mode C Local	ended memory acce	(0)05	
C Qusto C:Voc	system temporary f m Settings cuments and Setting e analysis log file	ooder 3sWoe11Local Settings	s\Temp\ 🗗 🏕
 Using 	a single thread		
C U <u>s</u> ing	multiple threads/co	res	
Message :		sis	

At the beginning of the analysis AxisVM divides the system of equations into blocks according to the available physical and virtual memory. It makes analysis more efficient but can considerably slow down other applications. Set the amount of virtual memory you let AxisVM use during the analysis here.

Enable extended memory access (AWE)	If more than 4 GB of memory is installed, this option makes it possible to get more memory for analysis. If this option is disabled it means that memory pages are not locked. See 2.1 Hardware Requirements for details
Using a single thread / Using multiple threads	<i>Using multiple threads</i> makes AxisVM run analysis on multiple threads. To make the most of this option it is recommended to use a processor with HT-Hyperthread or DualCore technology. Multi-threading improves speed of calculation. Improvement depends on the available memory and the model size. Linear analysis will be 1.5 times faster, while vibration analysis can be 4 times faster.
Folder for tempo- rary files during	You can specify the location of temporary files during analysis.
analysis	Select any of these options :
	 Model file folder Local system temporary folder Custom
	<i>Create analysis log file</i> If this option is turned on technical details of the analysis will be logged and saved to a text file <i>modelname_msg.txt</i> .
Message sounds	If this option is activated system sounds will be played after completing an analysis or get-

Message sounds If this option is activated system sounds will be played after completing an analysis o *during analysis* ting an error message. Sound card and speakers must be present.



- Report language Depending on your configuration you can select from the following languages: English, German, French, Italian, Spanish, Dutch, Hungarian, Russian, Portugese, Romanian, Serbian.
 - Table layoutIf Allow multiple columns is checked, narrow report tables will be printed in a multi-column
layout to reduce the space required. Minimum number of rows per column can be specified to
avoid column breaks for short tables.
 - *Printer buffer* If a report includes many pictures building the entire report in memory may consume too much system resources an cause printing problems. In this case set printer buffer to hard disk.

Print page numbers even if page header is turned off

If this option is turned on page numbers appear on printed pages even if headers are disabled in the printing dialog.

Translate automatic item names if report language is changed

If this option is turned on AxisVM-generated names of Drawings Library or report items will be translated automatically.



Searching for program update

Update

The for AxisVM checks regularly if there is an update available on the web. The frequency of update checks can be controlled. If Never is chosen an update process can be launched by clicking AxisVM Web Update. The date of the latest search is displayed. If internet connection goes through a proxy server, proxy settings has to be defined after clicking Proxy settings.



Click the button to get to the AxisVM Web Update Wizard which is a guide to the download process. If download is complete and the *Update the program* option is checked on the last page, the program quits and start the installation of the new release.



3.3.10. Language



If program configuration includes the DM module this menu item allows the user to change the program language (used in menus and dialogs).

3.3.11. Report Language



If program configuration includes the DM module this menu item allows the user to change the report language (used when displaying printable drawings, tables and reports).

3.3.12. Toolbars to default position

The moveable Icon bar will get back to the left side. All flyout toolbars undocked and dragged to a new position will get back to the Icon bar.

3.4. View

<u>F</u> ile	<u>E</u> dit	<u>S</u> ettings	⊻ie	w <u>W</u> indow <u>H</u> elp	
			tĽ,∗	<u>F</u> ront View	Ctrl+1
			Ľ×	<u>T</u> op View	Ctrl+2
			ťγ	<u>S</u> ide View	Ctrl+3
			7	<u>P</u> erspective	Ctrl+4
				P <u>e</u> rspective Setting	s
			-2	<u>W</u> orkplanes	۲
			€,	Zoom in	Ctrl+/
			Q,	Z <u>o</u> om out Sh	ift+Ctrl+/
			\mathbb{R}	Fit in Window	Ctrl+₩
			⇔	P <u>a</u> n	
			Ψ	<u>R</u> otate	
			۹۹,	<u>V</u> iew undo	Ctrl+[
			∎¢	View re <u>d</u> o	Ctrl+]
			•	Wirefra <u>m</u> e	
				<u>H</u> idden line remova	I
				Re <u>n</u> dered	
				Te <u>x</u> ture	
				Rendering options	
				Wireframe <u>c</u> ross-se	ctions
			٠	Actual cross-section	ıs
				Wireframe while dr	agging
			~	No la <u>b</u> els while drag	gging

Front view



See... 2.15.3 Views

Top view

1¥ ¥

See... 2.15.3 Views

Side view

[Ctrl]+ [3] See... 2.15.3 Views

Perspective view

[Ctrl]+ [2]



See... 2.15.3 Views

Setting Perspective View

Work planes



See... 2.15.4 Workplanes

See... 2.15.3 Views

See... 2.15.2 Zoom

See... 2.15.2 Zoom

Zoom in

 \oplus

[Ctrl]+ [/], See... 2.15.2 Zoom



[Ctrl]+ [Shift]+[/],[-] See... 2.15.2 Zoom



[Ctrl]+ [W]



See... 2.15.2 Zoom



View undo

÷Q,

[Ctrl]+[See... 2.15.2 Zoom

View redo		
G I	Ctrl]+]	See 2.15.2 Zoom
Wireframe		See 2.15.6 Display Mode
Hidden line rer	noval	
		See 2.15.6 Display Mode
Rendered		See 2.15.6 Display Mode
Texture		See 2.15.6 Display Mode
Rendering opti	ions	See 2.15.6 Display Mode
Wireframe cros sections	SS-	In rendered mode thin walled cross-sections will be displayed only with mid-planes.
Actual cross-sections		In rendered mode thin walled cross-sections will be displayed as solid objects with their actual shape.
Wireframe while dragging	e	If it is switch on, the program display the wireframe of the model during the rotation or pan.
No labels while dragging	9	If this option is turned on, labels are not drawn during rotation or panning.

3.5. Window



3.5.1. Property Editor

Property Editor provides the fastest way to change properties of the selected nodes, elements or loads. All changes are made immediately. If the selection contains different elements it is possible to change their common properties (e.g. after selecting trusses, beams and ribs their material and cross-section will be editable). If result or design tabs are active the values are read only.

In certain fields regular mathematical expressions are also accepted.

Available operators and functions are:

(,), SIN, COS, TAN, EXP, LN, LOG10, LOG2, SINH, COSH, TANH, ARCSIN, ARCCOS, ARCTAN, ARCSINH, ARCCOSH, ARCTANH, INT, ROUND, FRAC, SQR, SQRT, ABS, SGN. Few fast operators:

++8 adds 8 to the actual value

--8 substracts 8 from the actual value

Negative numbers within operation have to be in brackets.

In these expressions # substitutes the actual value (For instance #/3 divide it by 3). When entering a value of nodal coordinates, load values, surface thicknesses you can refer to global coordinates as X, Y, Z or x, y, z. In case of certain load types variables refer to other load components as well.

For nodal loads or point loads on beams variables Fx, Fy, Fz, Mx, My, Mz refer to force and moment components. For distributed beam loads px1, py1, pz1, m1, px2, py2, pz2, m2 refer to load components. Variable names are not case sensitive.

Example 1: If you want to turn selected distributed wind loads with different X components to Y direction enter 'px1' into field pY1 and 'px2' into field pY2 then enter zero into fields pX1 and pX2.

Example **2**: to scale the structure in direction X by 200%, first select all nodes then click the line first line and enter X*2 as X.

The question mark button turns on/off the help information.

Properties are displayed in a tree-like structure. Clicking a [+] or [–] symbol before the property name expands or collapses a list of sub-properties. If the (...) button appears in a line the property can be changed using a separate dialog.

If the (>>) button appears in a line the property can be picked up from another element by clicking it.

Property Editor can be used to modify data but also to select and filter elements with the same property.





Selecting a property and clicking the filter button you can select all the elements having the same property value.

Example: changing an existing cross-section in the whole structure.

Selecting the cross-section property of a rib element you can select all rib elements with this cross-section then change their cross-section property of them.

3.5.2. Information Windows

Lets you set the display of the Info, Coordinate, and Color Legend Windows to on or off. **See...** 2.17 Information Windows

3.5.3. Background picture

The submenu makes several options available. An automatically fitted background picture can be loaded to the main window of AxisVM to show the model in its future environment. *Load Background Picture...* submenu item or **[Ctrl+B]** opens a file browser dialog, *Reload Background Picture* shows the most recently used picture files. In multi-window mode each window can have its own background picture.

Picture in the active window can be turned on and off by clicking *Display* or by **[Ctrl+Alt+B]**. *Save Background Picture* saves the picture in the active window into a file. If the aspect of the picture differs from the window aspect *Shift Background Picture* makes it possible to drag the background to a new position. *Remove Background Picture* removes the picture in the active window.

Background pictures are saved into the AXS file.

After loading a background picture the model can be set to an appropriate view by zooming out, zooming in, panning, rotating and setting the perspective.

. <u>5</u> × þ..... ▼1 ● #100 |=| E • My Diegran Linea Code Case E (W) E (P) E (Eq) Comp. Inactive graphics window 282 42.90 Active graphics window 6.04 d f[m]: 11.036 d 4[°]: 23.89 dh[m]: 0

3.5.4. Split Horizontally

Splits the graphics window horizontally into two parts. The display settings of each window can be set independently.

You can maximize or minimize or restore the graphics windows by using the buttons at the top-right of the windows.



3.5.5. Split Vertically

Splits the graphics window vertically into two parts. The display settings of each window can be set independently.

You can maximize or minimize or restore the graphics windows by using the buttons at the top-right of the windows. Different load cases can be set in each window but only when displaying results.

3.5.6. Close Window

Closes the current graphics window.

3.5.7. Drawings Library

The Drawings Library contains drawings saved in the program. Drawings are not saved pictures but instructions how to draw a view of the model or parts of it including multiwindow settings. Drawings can be reloaded to restore saved view and display settings. Including drawings into a report makes it easier to update the report when the model has changed and recalculated as drawings will be updated automatically like tables.

Drawings Library can store displacement, force, stress diagrams of line elements, diagrams of steel and bolted joint design, punching analysis, reinforced concrete column check and beam design in an associative way.



Clicking the arrow beside the tool button an existing drawing can be selected from a pop-up list, restoring its view and display settings.

After clicking the Drawings Library tool button a dialog appears.

Drawings Library	×
× 🗈 🖿	wind1, Side View
Stored drawings	Display
Wind1 wind1 wind1, Front View wind1, Side View wind1, Top View	
	OK Cancel

X

暍

This dialog is to overview, maintain and reload saved drawings.

Deletes a drawing from the Drawings Library



 \times

Loads a chosen drawing to the active window. (available in multi-window mode only)

¥....

Loads a chosen drawing to the window.

Restore result components

If this option is checked loading a drawing displaying results restores the result component as well and sets the appropriate tab (Static, Vibration, etc.).

If this option is unchecked loading a drawing does not restore the result component and the tab.

- **OK** Saves the changes and loads the selected drawing.
- **Cancel** Does not save changes.

3.5.8. Save to Drawings Library

By clicking this tool button one or more drawings can be saved into the Drawings Library.

If the current drawing already exists, a *Found in the Drawings Library* label is displayed in the dialog. It can be overwritten or the drawing can be renamed. *Multiple drawings* button opens additional options. Load cases, load combinations (and result components if results are displayed) can be chosen. AxisVM creates all combinations (i.e. all selected result components in all selected load cases) and saves them into the library with the current view and display settings.

Clicking the Drawings Library button displays the Drawings Library dialog.

3.6. Help

File	Edit	<u>S</u> ettings	⊻iew	<u>W</u> indow	Help	
					۲	Contents
					0	<u>A</u> xisVM Home Page
					G	A⊻isVM Web Update
						About
						Release information

Lets you use the online help of AxisVM. To get context-sensitive help information about the operations related to a dialog box press **[F1]**.

3.6.1. Contents Some and allows access to the topics you are interested in. (F1) Opens the table of contents of the help, and allows access to the topics you are interested in. 3.6.2. AxisVM Home Page Visits AxisVM Home Page using the default Internet browser. 3.6.3. AxisVM Update

Launches the AxisVM Web Update Wizard. See... 3.3.9 Preferences

3.6.4. About

12



Displays more information about your AxisVM program. You can determine the version and release number, configuration, serial number and time limit of your AxisVM version.

Available modules are black, others are gray.

3.6.5. Release information...

Latest release information and history of fixes and new developments.

3.7. Main toolbar

3.7.1. New	
	See 3.1.1 New
3.7.2. Open	
[Ctrl]+[0]	See 3.1.2 Open
3.7.3. Save	
[Ctrl]+[S]	See 3.1.3 Save
3.7.4. Print	
Gtrl]+[P]	See 3.1.10 Print
3.7.5. Undo	
∽ ▼ [Ctrl]+[Z]	See 3.2.1 Undo
3.7.6. Redo	
Car [Shift]+[Ctrl]+[Z]	See 3.2.2 Redo
3.7.7. Layer N	lanager

See... 3.3.3 Layer Manager

3.7.8. Stories

±0.00 ▼	See 3.3.4 Stories
[F7]	

3.7.9. Table Browser

[F12]	

See... 2.9 Table Browser

3.7.10. Report Maker



See... 2.10 Report Maker

3.7.11. Drawings Library



See in detail... 3.5.7 Drawings Library

3.7.12. Save to Drawings Library

6

See in detail... 3.5.8 Save to Drawings Library

4. The Preprocessor

The preprocessor lets you create or modify the geometry of the model, in a completely visual way. The advanced Visual Modeling feature allows quick and reliable modeling and design.

This chapter introduces the AxisVM modeling commands (geometry generation, element / mesh generation, and load case/combination definition).

4.1. Geometry

Geometry commands let you interactively and graphically create the model geometry in 3D.

The model geometry is defined by nodes (points), mesh lines (lines) between nodes, and surfaces (triangular or quadrilateral) created from three or four appropriate lines. Later you can define finite elements based on the geometry constructed here.

In the case of surface structures (plates, membranes, or shells) the mesh consists of quadrilaterals that represent the median plane of the elements.

Automatic meshing on domains

Automatic meshing on *macro* quads and triangles



In the case of frame structures (beams or trusses) the mesh consists of the axes of the elements.



4.2. The Geometry Editor



When AxisVM starts, the graphical user interface is ready for geometry editing. In case of a new model X-Y, X-Z or perspective view can be set as the default view. In case of an existing model the latest view settings will be loaded.

Using the horizontal icon toolbar at the top of the graphics area you can apply various commands to construct geometry meshes describing the geometry of your finite element model. **See**... 4.8 Geometry Toolbar

Using the vertical icon bar on the left you can apply commands that change the display of the model, and can configure the working environment of the editor. **See**... 2.15 The Icon bar

4.2.1. Multi-Window Mode

		When the model is complex, it is useful to display different views of the model simultaneously on the screen. AxisVM allows you to split the graphics area horizontally or vertically. Each newly created graphics window has its own settings, and allows the independent display of the model views. This feature is also useful when interpreting results. You can access split commands from the Window menu.			
Split horizontally		Splits the active graphics window horizontally into two equal parts. The top window wi become the active window.			
		See 3.5.4 Split Horizontally			
Split vertically		Splits the active graphics window vertically into two equal parts. The left window will become the active window.			
		See 3.5.5 Split Vertically			
Close Window		Closes the active window if there are more than one graphics windows in use. The new default window will be that in which you previously worked.			
		You can change views during any editing command.			
	P	In the perspective view some editing commands cannot be used, or are limited in use.			

4.3. Coordinate Systems

AxisVM uses different coordinate systems, to describe the model. The global coordinate system is used to describe the model geometry. Local coordinate systems are mainly used in the element definitions. The local systems are usually defined by the element geometry and additional references. AxisVM denotes the axes of the global system with capital letters, and the local axes with small letters.

The geometry can be created using Cartesian, Cylindrical or Spherical coordinate systems.

See... 4.3.2. Polar Coordinates

4.3.1. Cartesian Coordinate System

Base coordinate AxisVM uses Cartesian coordinates to store system geometry data. AxisVM uses the right-hand rule exclusively to define the positive directions of axes and rotation. The illustration shows the positive directions of the axes and of rotation according to the right-hand rule.



Global and relative A new model uses the view selected in the New Model dialog (see... 3.1.1 New Model). origo The origin of the coordinate system is shown by a blue X initially located at the left bottom corner of the editor window.

A fixed (X, Y, Z) and a relative (dX, dY, dZ) global system are used to locate points (nodes) in your model. The origin of the relative system can be moved anywhere (using [Alt]+[Shift] or [Insert]), at any time during modeling.

The Coordinate Window displays either the fixed or the relative global coordinates according to its current settings. If the relative mode is selected, the denotation of axes becomes dX, dY, dZ.

With the help of the Coordinate Window, and according to the movement of the relative origin you can make measurements on the model (distances, angles).

The nodal displacements and mode shapes refer to the fixed global system.

^(*) In the X-Y and Y-Z views the third axis (normal to the view's plane) is oriented toward you. As a result, when a copy is made by translation with a positive increment about the respective third axis, the copies will be placed in front (toward you). The opposite occurs with the third axis in the case of an X-Z view is oriented in the opposite direction.

See... 4.9.19 References

4.3.2. Polar Coordinates

In addition to the Cartesian global coordinate system, you can use either a cylindrical or a spherical coordinate system. One of the polar coordinate systems can be selected through its corresponding radio button in *Settings / Options / Editing / Polar coordinates*.

In the Coordinate Window three variables will be displayed depending on selection:

Cylindrical

- h: the value measured from the view plane to a point on the cylinder's main axis (that is perpendicular to the view plane) oriented outward from the screen
- r: radius that is the distance on the view plane from the projection of the point to the cylinder's main axis
- a: the angle between the line that joins the point with the origin and the horizontal

Spherical

- r: the radius, that is the distance from the point to the sphere's center (origin)
- a: the angle on the view plane between the line that joins the projection of the point with the origin and the horizontal
- b: the angle between the line that joins the point with the origin and the view plane, which is positive if the point is in front of the view plane (between the user and the view plane).



Cylindrical Coordinate System



Spherical Coordinate System

4.4. Coordinate Window

X	X[m]:	1.056		r[m] :	1.275
	a Y[m]:	25.000	a	a[°]:	34.06
e e	* Z[m]:	0.714	a	h[m]:	25.000
	L[m]:	25.032			

Displays the current absolute and relative values of the cursor position in the global coordinate system (Cartesian and cylindrical or spherical).

You can switch between absolute and relative coordinate displays, by clicking on the letters *d* in the Coordinate Window. The display of the **d** letters also show whether the relative coordinates are enabled or not.

X		dX[m] :	0.914	d r[m]: 1.308	l
	L.	dV[m]:	25.000	d a[°]: 45.65	l
	a	dZ[m] :	0.935	a dh[m]: 25.000	L
		dL[m] :	25.034		L

The positive angles, α:



- *The relative switch (delta) can be used together with the constrained cursor movements.* **See...** 4.7.4 Constrained Cursor Movements.
- *You can enter expressions in the edit fields (e.g.: 12.927+23.439, cos(45), sin(60))*

4.5. Grid

See in detail... 2.15.15.1 Grid and Cursor

4.6. Cursor Step

See in detail... 2.15.15.1 Grid and Cursor

4.7. Editing Tools

Editing tools help the work by several features. See... 2.15.15.2 Editing

4.7.1. Cursor Identification

Sets the size of the cursor identification area (in pixels).



When you position the cursor over the graphics area, AxisVM finds the entity of the model that is closest to the center of the cursor from among the entities that are located in or intersect the identification area. The size of the identification area can be set at *Settings / Options / Editing / Cursor identification*.

The current shape of the cursor shows what kind of entity was identified. Depending on entity type, the cursor will have the following shapes:

Node	h			
Mid-side node	$\mathbf{h}_{\frac{1}{2}}$			
Support	►	\searrow		
Edge hinge	\mathbf{F}			
Mesh independent load	►Ļ			
Load polygon vertex	►ţ.			
Center of an arc	\$,			
Arc	N			
Tangent	Ŋ			
References	•*	►,≊	► ˆ	
Line	•⁄			
Surface	Þ⊡	►∆		
Intersection	►×			
Perpendicular (normal)	⊾			
Guideline	•/			
Domain	•			
Rigid element	►			
Dimension line	×			
In case of <i>Pick up</i> function	l			
Text box, label	►			
Reinforcement domain, COBIAX solid area	▶			

If there are several entities at the same location, the program identifies the first entity according to the ordering of the list above. If there are multiple entities of the same type, the cursor will show a double symbol.

It was actually identified. Which one of the elements was actually identified.

Background The cursor can be set to detect the lines on architecture background layers. *detection*

4.7.2. Entering Coordinates Numerically

During the model editing, coordinates of the cursor can be specified directly entering the numerical values into the Coordinate Window. There are two ways to enter the numerical values:

- 1. by pressing the corresponding character button on the keyboard
- 2. by clicking with the left $^{\circ}$ button on the desired coordinate value display field, and then typing in the value.

If the relative mode is enabled (the letter d is depressed), the coordinates you enter will define a point from the relative origin.

If contradictory values are entered (in case of a constraint), the last entered value will update the others.

Tou can enter expressions in the edit fields (e.g.: 12.927+23.439, cos(45), sin(60)) Tous an enter expressions in the edit fields (e.g.: 12.927+23.439, cos(45), sin(60))

The relative origin can be moved at any time, anywhere. Therefore when drawing a line, you can specify its endpoint coordinates relative to different origins.

To draw a line with a given length and direction move to relative origin to the starting point (using [Alt]+[Shift] or [Insert]), enter the angle at d a[°] and enter the length at d r[m] then press the Enter button.

4.7.3. Measuring Distance

The distance between two points or the length of a line can be measured by moving the relative origin onto the first point and then identifying the second point by positioning the cursor over it. In this case the value of dL in the Coordinate Window is the distance between the points.

The cursor can be moved to a location relative to a reference point by moving the relative origin onto the reference point, then entering the angle in the input field da and the distance in the dr input field.

4.7.4. Constrained Cursor Movements

The cursor movement constraints can be customized in the *Settings / Options / Editing* dialog. The constrained cursor movements use the following values:

Constraint Angle
Δα. [°] = 15,00
Custom a: ["] = 0

 $\Delta \alpha$ Holding the **[Shift]** key pressed, the cursor is moving along a line that connects its current position with the origin, and that has an n* $\Delta \alpha$ angle, where the value of n depends on the current cursor position.

Custom α Holding the [Shift] key pressed, the cursor is moved a line that connects its current position with the origin, and that has an α or α +n*90 \oplus angle, where the value of n depends on the current cursor position.

 $\Delta \alpha$ and α can be set in *Settings/Options/Editing/Constraint Angle*.

The meaning of origin depends on the d switches of the coordinate palette. Turning off both the origin will be the global origin. Turning on any of the d switches the origin will be the local origin.

You cannot use $\Delta \alpha$ and Custom α constraints in perspective view.

If the cursor is over a line, holding the key **[Shift]** depressed, will constrain the cursor movement to the line and its extension .

If the cursor identifies a point, holding the key **[Shift]** depressed, makes the cursor move along the line defined by the point and the relative origin..



When the cursor identifies a **domain or surface element** pressing **[Shift]** makes the cursor move in the plane of the element.



Geometry Tools

The icons of **Geometry Tools** allow you to lock the direction of drawing a line. **See...** 2.15.8 Geometry Tools

4.7.5. Freezing Coordinates

You can freeze the value of a coordinate, allowing for better positioning. A frozen coordinate will not change on cursor motion. Freezing can be achieved by using **[Alt] + [X],[Y],[Z],[L],[R],[A],[B], [H]** respectively.

×		X[m]:	3.000		r[m] :	3.081
	4	Y[m]:	0.700	а	a[°]:	13.13
	a	Z[m]:	0.000	a	h[m]:	0.000
		L[m]:	3.081			

A black rectangle over the coordinate input field shows that the coordinate is frozen. To cancel coordinate freezing, press the same button combination, that was used to freeze it or press [Alt]+ [Space].



4.7.6. Auto Intersect

At the intersection point of the lines, a node will be generated and the lines will be bisected. If surfaces are intersected by lines, they will be split, and the resulting elements will have the same material and cross-sectional properties as the original. Set the line intersection options in *Settings / Options / Editing / Auto Intersect*. See... 2.15.15.2 Editing If Auto Intersection is on, surfaces will be divided into smaller surfaces if necessary. Surface

finite elements are also divided and the new elements inherit the properties and loads of the original element.

4.8. Geometry Toolbar

Geometry	Elements	Loads	Mesh	Static	Buckling	Vibration	Dynamic	R.C. Design	Steel design	Timber design
• / /		5	$\supset \bigcirc$	$ \odot$	$\bigcirc +$	4 津田	1 888 ₽		$\times \times \rtimes$	田単丸ゆる

These tool buttons create new geometry or change the existing one..

If you are working on parts and Settings / Options / Editing / Auto / Part Management option is checked then all the newly created geometric entities will be added to the active parts.
The geometric entities can be selected prior to applying the geometry construction commands, as well.

4.8.1. Node (Point)

- Lets you place new nodes or modify existing ones.
 - To place a node:
 - 1. Move the graphics cursor to the desired location and press the **[Space]** key or the left mouse button (in perspective view you can place nodes only to special locations).
 - 2. Enter the node coordinates numerically in the Coordinate Window, and then press **[Space]** or **[Enter]** (it works in all views).

You can place a node on a line or surface. If the *Settings / Options / Editing / Auto Intersect* check-box is enabled, the line or surface will be divided by the new node, otherwise it remains independent of the line.

If nodes are generated closer to each other than the tolerance specified in Settings / Options / Editing / Editing Tolerance value, nodes will be merged.

When working on parts with Settings / Options / Editing / Auto / Part Management turned on all geometric entities created will be automatically added to the active parts.

4.8.2. Line

The Line Tool is to construct lines or other simple shapes. The line type can be chosen by clicking on the arrow at the bottom-right corner of the currently used Line Tool Icon, and then clicking on the desired Line Icon.

The Line Tool offers the following options to draw simple shapes:



Line Constructs straight lines by defining their end points (nodes). You must graphically or numerically (by the Coordinate Window) specify the endpoints (nodes). The command lets you generate one or more independent lines. You can cancel the process by pressing the **[Esc]** key or the right mouse button.

In perspective view lines are drawn on the Z=0 plane by default. To draw lines in perspective in a different plane workplanes can be used.

See... 2.15.4 Workplanes.

Polyline Constructs a series of connected straight lines (a polyline). You must specify the vertices.



- fut callent polymic by pre
 - 1. **[Esc]** key
 - 2. **[Esc]** key a second time will exit polyline drawing mode.
 - 3. ^(†) right button & Quick Menu/Cancel
 - 4. Heft button while pointing to the last point (node) of the current polyline.

Rectangle Constructs a rectangle (its corner points (nodes) and edge lines). You must specify two opposite corner points.



After you specified the first corner you can cancel the command by pressing the **[Esc]** key. This command is not available in perspective view.

Skewed rectangle

 \sim

Constructs a skewed rectangle (its corner points (nodes) and edge lines). You must specify one of its sides (by its endpoints), and then the other side.



After you specify the first corner you can cancel the command pressing the **[Esc]** key. In perspective view, you can draw skewed rectangles using only the existing points.

Polygon

Number of sides has to be defined in a dialog. Polygon has to be defined by entering a centerpoint and 2 polygon points.



Number of sides has to be defined in a dialog. Polygon has to be defined by entering three points of the arc.

4.8.3. Arc





Draws an arc or a circle. Arcs and circles will be displayed as polygons according to the Arc resolution set in *Settings / Preferences / Display*. **[Esc]** cancels the command.



Defining an arc by its radius, and starting and ending points.





Defining an arc by three points. The command can be applied in perspective setting as well.



4.8.4. Horizontal Division

-

This function creates a horizontal divider line passing through the cursor position. This line is in a plane parallel with the X-Y, X-Z or Y-Z plane depending on the actual view (or parallel with the workplane if a workplane is used). Creates new nodes at the intersections. If finite elements are intersected new elements inherit properties and loads of the original element.



4.8.5. Vertical Division



This function creates a vertical divider line passing through the cursor position. This line is in a plane parallel with the X-Y, X-Z or Y-Z plane depending on the actual view (or parallel with the workplane if a workplane is used). Creates new nodes at the intersections. If finite elements are intersected new elements inherit properties and loads of the original element.



4.8.6. **Quad/Triangle Division**



Constructs a mesh of quads/triangles over a quad or triangle. Use this command to generate a macro mesh before applying a finite element mesh generation command. If the mesh is fine enough, it can be used directly as a finite element mesh.

Quad-to-quads Generates an $n \times m$ mesh between the corners of a 3D quad (not necessarily flat, or with any side lines). You must successively graphically select the corners (four points), and specify the number of segments $(N_1 \ge 1)$ between corners 1 and 2, and the number of segments $(N_2 \ge 1)$ between corners 2 and 3.



The quad and the mesh are displayed with solid grey lines. G.

If the mesh leads to quad subdivisions that are distorted (have an angle smaller than 30° or greater than 150°), the quad is displayed with grey dotted lines.

If a quad shape is entered that is not allowed (e.g. concave), the quad is displayed with red dotted lines.



Quad-to-triangles

The command is similar to the quad-to-quads command, but each generated quad is divided into two triangles by its shorter diagonal.

The quad and the mesh is displayed with solid grey lines. If the mesh leads to triangle subdivisions that are distorted



the quad is displayed with grey dotted lines.

\$ \$ Creating Surfaces ok Cancel

Quad Division

If a quad shape is entered that is not allowed (e.g. concave), the quad is displayed with red dotted lines.



Triangle-to-quads

Constructs a mesh between the corners of a triangle (not necessarily with any side lines). The mesh will also contain triangles along the side that corresponds to the first two corners entered. You must graphically select the corners successively (three

Triangle Divisio \$ 4 Creating Surfaces OK Cancel

corners.

points), and specify the number of segments N between

The triangle and the mesh are displayed with solid grey lines. G____

If the mesh leads to quad subdivisions that are distorted (have an angle smaller than 30° or greater than 150°), or to triangle subdivisions that are too distorted (has an angle smaller than 15^o or greater than 165^o), the triangle is displayed with grey dotted lines.

If a quad shape is entered that is not allowed (e.g. three collinear corners), the triangle is displayed with red dotted lines.



Triangle-totriangle

The command is similar to the triangle-to-quads command, except that each generated quad is divided into two triangles by its diagonals which are parallel to the side first entered.



 G_{C} Same as for triangle-to-quads.



4.8.7. Line Division



Lets you create new point (nodes) on the selected lines. The following input options are available:

- **By Ratio:** Lets you divide the selected lines into two segments. You must specify the parameter *a* of the location of the inserted node relative to the first node (*i*). The parameter *a* must be between 0 and 1. a=0.5 represents a division of the selected lines into two equal segments.
- **By Length:** Lets you divide the selected lines into two segments. You must specify the length (*d*) of the segment corresponding to the first node (*i* end). The parameter *d* must be between 0 and the total length.

Divide Line	ĸ
0 1	
ů <u> </u>	
a = 0,35 👤	
Method	
By <u>R</u> atio	
O By Length	
◯ Into <u>N</u> equal parts	
C Uniform by length	
	1
OK Cancel	

Evenly: Lets you divide the selected lines into several equal-length segments. You must specify the number of segments (N).

Uniform by length: Lets you divide the selected lines into several equal-length segments. You must specify the length of segments (d).

before division

after division

If finite elements are divided the new elements inherit properties and loads of the original elements.

^{The sector of the surface of the su}

4.8.8. Intersect



Divides the selected lines by creating nodes (points) at their intersections. If finite elements are assigned to the lines, finite elements are also divided and inherit the properties and loads of the original element.

If the Settings / Options / Editing / Auto / Intersect check-box was not enabled in the dialog window at the time of creating the geometric entity, using this command you can intersect the selected lines. You can select elements for intersection beforehand.

4.8.9. Remove node

 \geq

Removes the selected nodes at the intersections of lines. It makes it easier to construct trusses crossing but not intersecting each other or to remove unnecessary division points along a line.

Intersection nodes can be removed only if the number of connecting lines are even and lines can be joined.

4.8.10. Normal Transversal

Creates a connection between two lines along their normal tranversal.

4.8.11. Intersect plane with the model



After defining the intersecting plane intersection lines and nodes will be added to the model. Domains, beams and ribs will be divided.

4.8.12. Intersect plane with the model and remove half space



This operation is similar to Intersect plane with the model, but after defining the plane a half space can be selected. Elements within that half space will be deleted.

4.8.13. Domain Intersection



Creates intersection lines of domains and line elements. After clicking the tool button select domains to create their intersection or select a domain and a line to create the intersection.



4.8.14. Geometry Check

This function selects (if *Only select nodes* is checked) or eliminates extra nodes and lines within a given tolerance and fixes domain contours forcing contour segments into the same plane and adjusting arcs if radius is not the same at the startpoint and the endpoint. You can specify the maximum tolerance (distance) for merging points. The default value is ΔL =0.001 [m].

Geometry Check 💌
Geometry Check
Tolerance [m] = 0.001
Qnly select nodes
List deleted nodes
Select unattached nodes or lines
✓ Checking domain contours
Tolerance [m] = 0.001
OK Cancel

Points that are closer together than this distance are considered to be coinciding.

If *Only select nodes* is checked, nodes closer than *Tolerance* will be selected but the model remains unchanged. If it is not checked, nodes closer than *Tolerance* will be deleted and a new node will be created with averaged coordinates. Lines connected to the nodes will be replaced with a single line to the new node. The command reports the number of merged nodes/lines. If *List deleted nodes* is checked a list of deleted nodes is displayed using the node numbers *before* the deletion. If *Select unattached nodes or lines* is checked a warning will be displayed if there are independent lines or nodes not connected to the rest of the structure.



Select unattached nodes or lines:

If this check-box is enabled, AxisVM will send a warning message if unattached (independent) parts are encountered.

The following case is not identified by the Check command.

To avoid having hiding lines check Settings / Options / Editing / Auto / Intersect or click Intersect on the Geometry Toolbar.



4.8.15. Surface

In any cases when you wish to model surfaces (plates, membranes, or shells) you have to create a mesh that consists of triangles and convex flat quadrilaterals. The mesh then can be refined. The command searches all triangles and quads in the selected mesh of lines. You must select all surface edges when applying the command. The number of surfaces detected is displayed in an info dialog.

The reported surfaces are geometry surfaces but not surface elements. You can make them surface elements by assigning material and cross-section properties to them.



Quads have to be flat. AxisVM takes into account only those surfaces that have an out-ofplane measurement smaller that the tolerance entered in the Settings / Options / Editing / Editing Tolerance.

4.8.16. Modify, transform

Lets you modify existing geometric entities.

To modify nodes or lines:

- 1. Position the cursor over the node/line/centre of surface.
- 2. Holding the left mouse button pressed, drag the node/line/surface.
- 3. Drag the node/line/surface to its new position, or enter its new coordinates in the Coordinate Window, and then press enter or press the left mouse button again.

If multiple nodes and/or lines are selected, the position of all nodes and lines will be modified.

Fast modify: Clicking a node you get to the Table Browser where you can enter new coordinate values. If multiple nodes are selected and you click one of them, all the selected nodes will appear in the table.

Moving selected nodes into the same plane: If the plane is a global one you can move selected nodes into this plane easily.

- 1. Click on any of the selected nodes.
- 2. Select the entire column of the respective coordinate.
- 3. Use *Edit / Set common value* to set a common coordinate value.
- *Using pet palettes* Depending on the type of the dragged element different pet palettes appear on the screen. Their position can be set in *Settings / Preferences / Toolbar*. **See...** 3.3.9 Preferences
- *Dragging nodes* The following dragging modes can be selected:
 - 1. Dragging the node.
 - 2. Dragging the node disconnecting selected connecting lines.
 - 3. Dragging the node translating connecting lines.
 - 4. Dragging the node lengthening or shortening connecting arcs.
 - 5. Detaching a copy of the node from the original.

The 6th and 7th tool buttons determines the behaviour of connecting arcs.

- 6. Center angle remains constant.
- 7. The new arc is defined by the dragged node, the startpoint and midpoint of the original arc.



8. When detaching icon (5th item on toolbar) is active you can select the properties which will detach with the selected node./line/arc

Entering node coordinates: Clicking a node the table of nodes appears where coordinates can be changed. After selecting one or more nodes their coordinates can be edited in the property editor as well.

Examples of aligning nodes to a plane if this plane is parallel with one of the global coordinate plane:

- 1. Select nodes to align.
- 2. Enter the required coordinate value in the property editor.



4.8.17. Delete

[Del] See in detail... 3.2.7 Delete

4.9. **Finite Elements**

The commands related to the definition of the finite elements are described below.



The commands associated with the icons let you define the finite elements used for modeling. In the definition process you must define and assign different property sets.

Depending on the type of finite element, you have to define the following properties:

Properties of finite elements

Finite element	Material	Cross-section	Reference	Stiffness	Surface
Truss	•	•	0		
Beam	•	•	•	0	
Rib	•	•	0		
Membrane	•		•		•
Plate	•		•		•
Shell	•		•		•
Support			•	•	
Rigid					
Spring			0	•	
Gap				•	
Link				•	
Edge hinge				•	

o: optional

Note that some elements like springs and gaps can have nonlinear elastic stiffness properties that are taken into account only in a nonlinear analysis. In a linear analysis the initial stiffness is taken into account for the spring element, and the active or inactive stiffness depending on its initial opening for the gap element.

4.9.1. Material

Define Materials	Table Browser File Edit Format Report Hel B Szél (18) -Load cases (3) -Load Combinations (5 - Winisht Report	stru	+ ×	: 🖻 🕮 🎚 terials - Eurocode	8	2 1						-	
	LIBRARIES Material Library		Nar	ne Type	E _x [kN/cm ²]	E _y [kN/cm ²]	ν	α _T [1/°C]	ρ [kg/m ³]	Material color	Contour color	Textúra	1
	DIN	1	C12/15	Concrete	2600	2600	0,20	1E-05	2500			Concrete A	_
	Eurocode	2	C16/20	Concrete	2750	2750	0,20	1E-05	2500			Concrete A	
	-Eurocode [A]	3	C20/25	Concrete	2900	2900	0,20	1E-05	2500			Concrete A	
	- Italian code	4	C25/30	Concrete	3050	3050	0,20	1E-05	2500			Concrete A	
	-MSZ	5	C30/37	Concrete	3200	3200	0,20	1E-05	2500			Concrete A	
	-NEN	6	C35/45	Concrete	3350	3350	0,20	1E-05	2500			Concrete A	_
	-STAS	7	C40/50	Concrete	3500	3500	0,20	1E-05	2500			Concrete A	-
	Custom	8	C45/55	Concrete	3600	3600	0,20	1E-05	2500			Concrete A	-
	Rehar Steel Gra	•											•
	Editing C12/15, Material Name										ок	Canc	el

Lets you define and save material property sets or load them from a material library. If you delete a material property set, the definition of the elements with the respective material will be deleted.

Ŧ AxisVM uses exclusively isotropic materials with linear elastic behavior. Browse Material Library The material library contains material properties of civil engineering materials based on Eurocode, DIN, NEN, SIA and other specifications. The following parameters are stored:



[Ctrl+L]

Ø

If a material type is deleted all elements made of this material will be deleted.

Material Depending on the type of the finite element you must define the following material *Properties* properties:

Finite Element	E	v	α	ρ
Truss	•		•	•
Beam	•		•	•
Rib	•		•	•
Membrane	•	•	•	•
Plate	•	•	•	•
Shell	•	•	•	•
Support				
Rigid				
Diaphragm				
Spring				
Gap				
Link				

Displaying and changing material properties is described in 3.1.13 Material Library.

In AxisVM all the materials are considered to be linear elastic (Hooke's Law), and uniform isotropic or orthotropic (for beam, rib, membrane, plate, and shell elements).
 Some elements can have nonlinear elastic material (truss), or stiffness (support, gap, link, spring elements).

Nonlinear material models are taken into account only in a nonlinear analysis. In a linear analysis the initial stiffness is taken into account for the nonlinear elements.

4.9.2. Cross-Section



Lets you define and save cross-sectional property sets or load them from a cross-section library. The beam, truss, and rib elements require a cross-section. The properties are related to the element's local coordinate system.

For cross-section properties see... 3.1.14 Cross-Section Library

[©] If you delete a cross-section property set, the definition of the elements to which it was assigned will also be deleted. The lines will not be deleted.

You must enter values for all properties.

Cross section properties are defined in the coordinate system of a truss / beam / rib element.

4.9.3. Direct drawing of objects

	× - 0	Top toolbar
Туре	Shell	
Material	C25/30	
Local x Orientat	× Auto	D
Local z Reference	× Auto	Property fields
Thickness [cm]	40,0	
COBIAX	CBCM-E-270	
Auto Stiffness	▼	
f _F	0,900	
V _{Rd.Cobias} [kN/	0	
$\Box \oslash \otimes \odot$	5 —	Bottom toolbar

After clicking the icon a direct drawing toolbar and property editor appears. With the help of this window columns, beams, walls, slabs and holes can be drawn. Their properties can be set previously and changed any time during the drawing.

The top toolbar shows the type of the object to draw and the orientation of the object (for columns and walls). Property fields can be edited like in the Property Editor.

The bottom toolbar shows the drawing methods available for the object (one segment, polyline, polygon, rectangle, etc.).

Clicking a domain contour before drawing holes forces the drawing into the plane of the domain.

Object types	Column (in global Z direction)
	Beam (in global X-Y plane)
	Beam (spatial) Wall (always vertical with a constant height, i.e. its normal and upper/lower edges are parallel to the global X-Y plane)
	Slab domain (parallel to the global X-Y plane)
	COBIAX slab Slab domain (parallel to the global X-Y plane)
	Surface domain (spatial)
	Hole
Object dragpoints	Column upwards / downwards
	Wall upwards / downwards
Object geometry	Single segment beam or wall
	Beam or wall polyline
	Arced beam with centerpoint, start and endpoint
	Arced beam with three points
Object geometry Image: Polygonal beam or wall Image: Walls on a rectangle Image: Walls on a slanted rectangle

4.9.4. Domain

A domain is a planar structural element with a complex geometric shape described by a closed polygon made of lines and arcs. A domain can contain holes, internal lines and points. Polygon vertices, holes and internal lines must be



A domain has the following parameters:

in same plane.

Element type (membrane, plate, shell) Material Thickness Local coordinate system

The following parameters can be assigned to the polygon, hole edges, internal lines and points of a domain:

- point, line, and surface support rib element distributed load dead load thermal load nodal degrees of freedom (DOF)
- A domain is displayed by a contour line inside of the domain's polygon, with a color corresponding to the domain's element type (blue for membrane, red for plate, and green for shell).



Domains can be defined for floors, walls, and any other complex structural surface element.

The domain can be meshed automatically. **See...** 4.11.1.2 Mesh generation on domain

More than one domain can be used to model a structural element.



A domain can contain other (sub-) domains.

Define a domain

Select lines on the contour of the domains you want to define. If you select more lines or lines from different planes, AxisVM will find the planes and the contour polygons of the set. The program applies the parameters you entered in a dialog window.

Domains	×
Define O Modify	
Туре	
 Membrane (plane stress) Membrane (plane strain) Plate Shell 	
Material C12/15	· 🔊 .
Local x Reference × Auto	
Pick Up >>	OK Cancel

Modify a domain Select the domain (click on the contour line of the domain) you want to modify and make the changes in the dialog displayed.

Delete a domain Press the **[Del]** button, select the domains (click on the contour line of the domain) you want to delete and click OK in the dialog.

4.9.4.1. COBIAX-domain

If the package includes the COBIAX module (CBX), void formers can be placed into slabs reducing self weight and the total amount of concrete, making larger spans available. COBIAX slabs can be designed according to Eurocode, DIN 1045-1 and SIA (Swiss) design code.

COBIAX domain parameters Clicking on the checkbox beside the graphics showing a COBIAX slab we can turn the void formers on or off. This checkbox is enabled only if the material is concrete and the thickness of the slab is at least 200 mm.

Models avaliable for the given thickness are listed in the dropdown combo box. Element parameters and the schematic diagram of the slab is displayed under the combo. Void formers reduce the stiffness and shear resistance of the slab. If we choose *Automatic*, factors will be set to their default values. These can be overridden after clearing the checkbox.

Domain self weight will be automatically reduced and analysis will be performed with reduced stiffness and shear resistance.

Definition of shear resistance depends on the current design code.

Domains	X
Type Membrane (plane stress) Membrane (plane strain) Plate Shell	
Material C25/30_1 Thickness [cm] = 50,0	Cobiax
Local x Reference	Auto
cobiax	CBCM-E-405
More information	Cage support height 41,1 cm Min. slab thickness 50,0 cm
Stiffness	Spacing45,0 cmVolume reduction0,1718 m³ /m²
E _C = 0,86 · E	Concrete reduction 429,5 kg/m² Stiffness factor 0,86
Shear resistance V _{Rd,Cobiax} [kN/m] = 0	Shear resistance 0 kN/m
Pịck Up »>	OK Cancel

Eurocode, These design codes require specification of the $V_{Rd,Cobiax}$ shear resistance.

DIN 1045-1 To estimate its value build the model with solid slabs and read the $(V_{Rd,c})$ shear resistance of the slab. Sheer resistance of COBIAX slabs is about half of the solid ones.

SIA 262 Swiss design code allows two options. It is possible to enter the actual shear resistance or only the shear factor.

^(*) If more than one COBIAX domains were selected, their COBIAX parameters can only be redefined. Modifying COBIAX parameters of multiple domains is not allowed.

- ↔ Void formers appear as circles drawn in the slab plane in wireframe mode and balls placed into a partially transparent plate in rendered view.
 - Colours assigned to COBIAX-slabs and void formers can be customized by clicking on the button right to to the element type combo.

Move void formers

Void formers are positioned according to a raster depending on element type. Certain design rules are applied near holes, edges, and supports. Shifting the origin of the raster void former positions will change accordingly.

Right-clicking the domain outline choose *Move Cobiax elements* from the popup menu. Enter the base point of the translation vector then its end point. Number of the void formers in the resulting raster is displayed while moving the mouse.



Cobiax parameters in the output Table Browser shows COBIAX slabs of the model and their parameters in one table under *Elements*.

Two additional tables appear in the *Weight Report* section. A table titled *COBIAX-elements* lists elements by type with the number of void formers, the total area covered and the total weight reduction. *COBIAX Weight Report* displays and sums the weight reduction of individual slabs.

For details of COBIAX slab design see... 6.5.10 Design of COBIAX slabs

4.9.5. Hole

Holes can be defined in domains. Holes have to be inside the domain and in the domain's plane.

Select the (closed) polygons that are the edges of the holes you want to define. You can move holes from one domain to another, or change their shape.



 $_{\mbox{\footnotesize GC}}$ Holes are displayed by a contour line with the color of the domain in which they are located.

4.9.6. Domain operations

Domain contours can be changed, cut and a union of domains can be calculated.

Change domain contour

1. Click the *Change domain contour* icon on the toolbar.

ΞŤ	-	•	
		•	

- 2. Select a domain to change. Domain countour will be selected.
 - 3. Change selection to modify domain contour and click OK on the selectioin toolbar.



^{CF} Domain properties (material, thickness, local system) will be retained but the existing mesh will be removed.

If loaded areas are removed from the domain, loads will automatically be removed.

Union of Domains	Union c
	Officite

5 Union can be created from adjacent domains.

- 1. Click the Union of domains icon on the toolbar.
- 2. Select the domains and click OK on the selection toolbar.
- 3. If domains have different properties (thickness, material or local system) one of the domains has to be clicked. The union will inherit properties from the clicked domain.



Cut domains

To cut domains along en existing line:

- 1. Click the *Cut domains* icon on the toolbar.
- 2. Select the domains.
- 3. Select the cutting line and click OK on the selection toolbar.



4.9.7. Line Elements

Line elements are defined and modified in a common dialog. After choosing the element type specific truss / beam / rib element parameters can be set.

Line elements are handled as structural members and not as finite elements. Meshing a line element divides a beam or a rib into finite elements. Existing line elements can be joined to form a single element if the geometry and their properties allow it. (*Edit / Find structural members*). Numbering, labeling, listing functions will consider it to be a single structural member. Structural members can be broken apart by *Edit / Break apart structural members*) **See...** 3.2.12 Assemble structural members, 3.2.13 Break apart structural members



Truss elements can be used to model truss structures. Trusses are two node, straight elements with constant cross-section properties along the truss length. A maximum of three translational degrees of freedom are defined for each node of the elements. The elements are pin-ended (spherical hinges).



Axial internal forces N_x are calculated for each truss. The variation of the axial force is constant along the element.

i denotes the truss end with the lower node index (first node). By default the element *x* axis goes from the node (*i*), to the node (*j*). It can be changed by selecting the other orientation from **Local x Orientation**.

You must select the lines to which you want to assign the same material and cross-sectional properties in order to define truss elements.

If elements of different type are selected element definition will be activated.

Defining Materials and cross-sections can be selected from built-in libraries or from a list of the *crials and* materials/cross-sections already defined.

materials and cross-sections

Define

150 Truss

ð

ĨĬ

Allows browsing of the cross-section library to assign a cross-section to the element. The cross-section selected will be added to the cross-section table of the model.

Allows browsing of the material library to assign a material to the element. The material



Launches the Cross-section Editor. The cross-section created in the Editor will be registered in the list of model cross-sections.

 $G_{\mathcal{O}}$ The truss elements are displayed on the screen as red lines.

selected will be added to the material table of the model.

Service class If the current design code is Eurocode and a timber material is selected, the service class can be set here. **For details see**...6.7 Timber Beam Design

Local x Orientation

Local x direction of a beam can be set to point from Node *i* to Node *j* or vica versa.

 $i \rightarrow j$: local x axis is directed from the end node with a lower number to the node with the higher one

 $j \rightarrow i$: local x axis is directed from the end node with a higher number to the node with the lower one

Setting this parameter to automatic means that the program determines this orientation based on the endpoint coordinates.

The orientation can be reversed any time using the shortcut **[Ctrl+E]** or in the dialog or in the property editor window.

Cross-section In the calculation of the element stiffness, only the cross-sectional area A_x is considered from the cross-sectional properties.

Local z Reference A reference point can be assigned to define the element orientation. This allows a correct display of the cross-section on the screen. In case of selecting Auto the reference(s) will be set by the program. Affects only the display of references. **See...** 4.9.19 References

Reference angle

le Rotation of cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel with the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

Nonlinear parameters

Beam

near In a nonlinear analysis you can specify that a truss has stiffness only if it is in tension or compression. You can optionally enter a resistance value as well. A nonlinear elastic behavior is assumed for the nonlinear truss elements.

The nonlinear parameters are taken into account only in a nonlinear analysis.

The initial elastic stiffness of a truss element is taken into account if a linear static, vibration, or buckling analysis is performed, disregarding any nonlinear parameter entered.

Beam 1 🔀
Type Truss Beam Rib
Material Properties
Material C24
Service class 2 V K _{def} = 0,8 V
Variable cross-section Cross-Section
Cross-Section 400x400_oak_beam 🗹 🗊 🎛
Local x Orientation 1→1 ▼
Local z Reference
End Releases Startpoint Endpoint
Pick Up >> OK Cancel

Beam elements may be used to model frame structures.

Beams are two-node, straight elements with constant or variable (linearly changing) cross-section properties along the beam length. A reference point is used to arbitrarily orient the element in 3-dimensional space (to define the local x-z plane). A maximum of three translational and three rotational degrees of freedom are defined for each node of the elements. The ends of the elements can have arbitrary releases.

Three orthogonal internal forces, one axial and two shear (N_x, V_y, V_z) , and three internal moments, one torsional and two flexural (T_x, M_y, M_z) are calculated at each cross-section of each element.

The variation of the internal forces along the beam are: constant axial force, constant torsion, constant shear forces and linear moments.

The displacements and internal forces are calculated at intervals of at least 1/10 of the element length.

i denotes the beam end with the lower node index (first node). By default the element x axis goes from the node (i), to the node (j). It can be changed by selecting the other orientation from **Local x Orientation**.



Material, cross- Defining material, cross-section and local direction X are similar to truss elements. *section, local x*

orientation

Automatic The reference vector will be generated by the program according to the section 4.9.19 *reference* References.

The orientation of the local x axis of the element can be reversed or can be set to Auto which means that local x directions will be set automatically based on the beam end coordinates.

Reference angle ∦₿

Rotation of cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel with the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

- $G_{\mathcal{A}}$ The beam elements are displayed on the screen as blue lines.
- End releases

Ε

S You can specify releases that remove the connection between the selected elements' degrees of freedom (in the local coordinate system) and the nodes. The end-releases are set by a six code set for each end. Each code corresponds to one internal force component. By default the beam ends are considered rigidly connected (all codes are of rigid connection) to the nodes. Setting a code as hinged connection will result in the corresponding internal force component of the respective end to be released. A semi-rigid connection code can be assigned to the in-plane rotation components of the beam ends.

	End Releases	×	
	, →Hinge around y and z axis 👻	→Hinge around y and z axis 🔹	
		∎ ○ ⊕ ●	
	∯v Hinge around z axis	7 8 6 0	
	∯,	4	
	, Å→Hinge around y and z axis	y ey to t	
	Å→ Spherical hinge	X e ₂ C C	
	s ∰→Roller along the y axis	θ _x @ C	
	st⊒ Roller along the z axis	Endpoint 0 _y C C C C	
	₽2 ₽2 0 0 0 0	€2 C C C C	
	Stiffness	Stiffness	
	s _{yy} [kNm/rad] =	s _{yy} [kNm/rad] =	
	s _{zz} [kNm/rad] =	s _{zz} [kNm/rad] =	
	Resistance	Resistance	
	M _{yH} [kNm] =	✓ M _{yH} [kNm] = 220,109 ▼	
	M _{2H} [kNm] =	M _{zH} [kNm] = 107,722	
nd releases at the			End releases at the
start node		OK Cancel	end node

- Graphical symbol of a rigid connection code (the corresponding local displacement component of the beam end is transferred to the node)
- **O** Graphical symbol of a hinged connection code (the corresponding local displacement component of the beam end is not transferred to the node)
- ⊕ Graphical symbol of a semi-rigid connection code (the corresponding local displacement component of the beam end is partially transferred to the node)
- Graphical symbol of a plastic connection: the maximum value of the moment at the endpoints is calculated from the material and cross-section properties.

End Release	Symbol
Hinge in <i>x-y</i> plane. Can't transmit M ₂ moment.	
Hinge in x - z plane. Can't transmit M_y moment.	₹ ×
Hinge in $x-y$ and $x-z$ plane. Can't transmit M_z and M_y moments.	
Hinge in <i>x</i> - <i>y</i> and <i>x</i> - <i>z</i> plane and free rotation about local <i>x</i> axis (spherical hinge). Can't transmit M_x , M_y , and M_z moments.	
Free translation along local y axis. Can't transmit V _y shear force.	▲ ×
Free translation along local z axis. Can't transmit V _z shear force.	Ĵ Ţ z

The table below demonstrates the use of end releases for some common cases:

Care must be taken not to release an element or group of elements such that rigid body translations or rotations are introduced.

For example, if you specify spherical hinges at both ends (code: 000111), a rigid body rotation about element axis is introduced. In this case at one of the ends you may not release the element degree of freedom corresponding to the rotation about local x axis (e.g. i end numerical code: 000011; j end numerical code: 000111).



Semi-rigid To define semi-rigid hinges set the radio button to semi-rigid and enter the torsional stiffness of the linear elastic spring modeling the connection about the local axis y or z. The value should be the initial stiffness of the real connection M- ϕ characteristics.

The moment - relative rotation diagram of a connection is modeled by a linear or nonlinear elastic rotational spring. The nonlinear characteristic can be used only in a nonlinear static analysis. In a linear static, vibration, or buckling analysis, the initial stiffness of the connection is taken into account.



For example, in the case of steel frame structures, Eurocode 3 Annex J gives the details of application.

Moment To fixed or semi-rigid connections a moment resistance can be assigned, that is the *Resistance* maximum moment that can develop in the connection.

¢,

The moment resistance parameter is used only in case of a non-linear analysis.

Plastic hinge To define plastic hinges set the radio button to plastic. Moment resistance will be displayed but cannot be edited. If elements with different materials or cross-sections are selected no value will appear in the edit field but hinges will be defined with the appropriate moment resistance.

After completing the nonlinear analysis and displaying beam internal force diagrams hinges that got into plastic state at the current load step become red. The number beside the hinge shows the order of getting into a plastic state. Hinge with number 1 is the hinge getting plastic first. Where hinges are not red, plastic limit moment is not reached yet.

- Plastic hinges can only be used with steel beams.
- Ger If any beam end release code is of a hinged connection, the beam end is displayed on the screen as a blue circle. If it has a stiffness value a blue cross is inscribed. If the end release corresponds to a spherical hinge, it is displayed as a red circle.

The plastic hinges are displayed as solid circles. The defined beams appear as dark blue lines.

Beam 1 🛛
© <u>D</u> efine C <u>M</u> odify
Type Truss Beam Rb
Material Properties
Material C24
Service class Class 2 V K _{def} = 0,8 V
☐ ⊻ariable cross-section
Cross-Section
Cross-Section 400x400_oak_beam
Local × Orientation
Local z Reference
End Releases
Startpoint Endpoint
Eccentricity Custom eccentricity
Pick Up >> OK Cancel

Rib elements may be used, independently or in conjunction with surface elements (plates, membranes, and shells) to model ribbed surface structures. When used attached to surface elements, the ribs can be connected centrically or eccentrically to the surface elements. The properties of the corresponding surface elements are used to orient the element in the 3-dimensional space (to define the local *x*-*z* plane).

When used independently, the ribs can model frame structures in a similar way as the beam element, but it can take into account the shear deformations. A reference point or vector is required to arbitrarily orient the element in the 3D space.

Rib elements are isoparametric three node, straight elements with constant or variable (linearly changing) cross-section properties along the rib length, and with quadratic interpolation functions. Three translational and three rotational degrees of freedom are defined for the nodes of the element. Three orthogonal internal forces, one axial and two shear (N_x , V_y , V_z), and three internal moments, one torsional and two flexural (T_x , M_y , M_z) are calculated at each node of each element. The variation of the internal forces within an element can be regarded as linear.

Rib

Define	You must assign the following properties:	
Material, Cross- section, Local x orientation	Defining material, cross-section and local direction	X are similar to truss elements.
Material	The material of the rib can be different from the surface).	surface material (if it is connected to a
Cross-section	The rib element's cross-section is taken into account	t as is shown in the figure below:
Automatic reference	The reference vector will be generated by the progr	am according to the section References
Reference	Independent rib: The local coordinate system is defined as follows: the element axis defines the x local axis; the local z axis is defined by the reference point or vector; the y local axis is according to the right-hand rule.	Reference point

Rib connected to a surface element:

The local coordinate system is defined as follows: the element axis defines the x local axis; the local z axis is parallel with the z axis of the surface element; the y local axis is parallel with the plane of the surface element, oriented according to the right-hand rule. The figure below shows that when the beam is located on the edge of two surface elements that makes an angle, the local z axis is oriented by the average of normal axes of the surfaces. If more than two surfaces are connected to the edge and you select one or two of them then an automatic reference will be available when defining the rib.

The cross-sectional properties must be defined in this coordinate system.



Reference angle ⊉₿

The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel with the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

End releases End releases can be defined for ribs the same way as for beams. By default both ends are fixed.

Eccentricity You can specify eccentricity for a rib only if it is on the edge of one or two surfaces. If more than two surfaces are connected to the edge select one or two of them to define eccentricity for the rib.

The eccentricity (ecc) of a rib is given by the distance of the center of gravity of its crosssection to the plane of the model of the surface (neutral plane). It is positive if the center of gravity is on the positive direction of its local z axis.

There are four options to set the rib eccentricity. Bottom rib, top rib, rib in the midplane or custom eccentricity.

In the first three cases the actual eccentricity is calculated from the rib cross-section and the plate thickness. If the rib is made of concrete the definition of top and bottom ribs are different, so button pictures change according to the rib material. If rib cross-section or plate thickness changes the eccentricity is automatically recalculated.

If the rib is made of steel or timber, connected to a shell and is defined as a top or bottom rib, an additional axial connection stiffness can be defined.

In case of reinforced concrete plate-rib connections rib cross-section must include the plate thickness. In other cases (steel or timber structures) the cross-section is attached to the top or bottom plane of the plate.



For plates, the eccentricity of the rib will modify the flexural inertia of the rib as follows: $I_y^* = I_y + A \cdot exc^2$

For shells, due to the eccentric connection of the rib to the shell, axial forces will appear in the rib and shell.

- G_{C} Ribs appear as blue lines.
- Modifying Selecting elements of the same type and clicking the tool button Modifying will be actived. Properties of elements can be changed if the checkbox before the value is checked. If a certain property is does not have a common value its edit field will be empty. If a value is entered it will be assigned to all selected elements.
 - *Pick Up>>* Properties of another element can be picked up and assigned to the selected elements. Clicking the *Pick Up* button closes the dialog. Clicking an element picks up the value and shows the dialog again.

Only those properties will be copied where the checkbox is checked.

4.9.8. Surface Elements

Surface elements can be used to model membranes (membrane element), thin and thick plates (plate element) and shells (shell element) assuming that the displacements are small. As surface elements you can use a six node triangular or eight/nine node quadrilateral finite elements, formulated in an isoparametric approach. The surface elements are flat and have constant thickness within the elements.

^(*) It is preferable for the element thickness not to exceed one tenth of the smallest characteristic size of the modeled structural element, and the deflection (w) of a plate or shell structural element is less than 20% of its thickness (displacements are small compared to the plate thickness).

Use of elements with the ratio of the longest to shortest element side lengths larger than 5, or with the ratio of the longest structural element side length to the thickness larger than 100 are not recommended.

In some cases when the elements are used (that are flat with straight edges) to approximate curved surfaces or boundaries, poor results may be obtained.



Ø

Membrane



Membrane elements may be used to model flat structures whose behavior is dominated by in-plane membrane effects. Membrane elements incorporate in-plane (membrane) behavior only (they include no bending behavior).

œ. The element can be loaded only in its plane.

AxisVM uses an eight node Serendipity,

plane stress ($\sigma_{zz} = \sigma_{xz} = \sigma_{yz} = 0$, $\varepsilon_{xz} = \varepsilon_{yz} = 0$, $\varepsilon_{zz} \neq 0$) or

plane strain ($\varepsilon_{zz} = \varepsilon_{xz} = \varepsilon_{yz} = 0$, $\sigma_{xz} = \sigma_{yz} = 0$, $\sigma_{zz} \neq 0$),

finite element as membrane element.

The membrane internal forces are: n_x , n_y , and n_{xy} . In addition the principal internal forces n_1 , n_2 and the angle α_n are calculated.

The variation of internal forces within an element can be regarded as linear.

The following parameters should be specified:

- 1. Plane strain or plane stress
- 2. Material
- 3. Thickness
- 4. Reference (point/vector/axis/plane) for local x axis
- 5. Reference (point/vector) for local z axis



Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.

Automatic reference:

The axis of element local directions x and z can be determined by reference elements, see part 4.9.19 References or can be set automatically.

The center of the membrane elements is displayed on the screen in blue. 6

Plate

Туре			-	
C Membrane (plan	ie stress)			~
🔿 Membrane (plar	ie strain)			
Plate				
C Shell				
Managerian D			ش ا	
wateriai	.35/45	<u> </u>	Шh	
Thickness [cm] =	20.0 💌			×
Local x Refe	rence »	× Auto	-	\checkmark
Level - Defe		X Auto		\sim

Plate elements may be used to model flat structures whose behavior is dominated by flexural effects.

AxisVM uses an eight/nine node Heterosis finite element as plate element, that is based on Mindlin-Reissner plate theory that allows for transverse shear deformation effects). This element is suitable for modeling thin and thick plates as well.

Plate elements incorporate flexural (plate) behavior only (they include no in-plane behavior).

The element can only be loaded perpendicular to its plane.

The plate internal forces are: m_x , m_y , m_{xy} moments, and v_x , v_y shear forces (normal to the plane of the element). In addition, the principal internal forces: m_1 , m_2 , the angle α_m and the resultant shear force q_R are calculated.

The variation of internal forces within an element can be regarded as linear.

The following parameters should be specified:

- 1. Material
- 2. Thickness
- 3. Reference (point/vector/axis/plane) for local x axis
- 4. Reference (point/vector) for local z axis



Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.

Automatic reference:

The axis of element local directions x and z can be determined by reference elements, **see part** 4.9.19 References or can be set automatically.

 $G_{\mathcal{A}}$ The center of the plate elements is displayed on the screen in red.

face Elements					
Oefine	C Modify				
Туре —					
C Membrane	plane stress)				
C Membrane	plane strain)			- /~V 7)	
C Plate					
Shell				~	
	r		 		
Materia	C35/45	-	ø		
Thickness [cm]	= 20.0 💌			×	
Local x F	eference »	× Auto	-		
Local z F	eference »	× Auto	-	\sim	
Pick Up >>			ок	Cance	el .
Tiok op 33					

Shell elements may be used to model structures with behavior that is dependent upon both in-plane (membrane) and flexural (plate) effects.

The shell element consists of a superimposed membrane and plate element. The element is flat, so the membrane and plate effects are independent (first order analysis).

The element can be loaded in its plane and perpendicular to its plane.

The shell internal forces are: n_x , n_y , and n_{xy} forces (membrane components), m_x , m_y , and m_{xy} moments, and v_x , v_y shear forces (plate components). In addition, the principal internal forces and moments n_1 , n_2 , the angle α_n , m_1 , m_2 , the angle α_m and the resultant shear force vSz are calculated.

The variation of internal forces within an element can be regarded as linear.

The following parameters should be specified:

- 1. Material
- 2. Thickness
- 3. Reference (point/vector/axis/plane) for local x axis
- 4. Reference (point/vector) for local z axis

Shell

Modifying

ð	Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.
	Automatic reference : The axis of element local directions x and z can be determined by reference elements, see part 4.9.19 References or can be set automatically.
6.	The center of the shell elements is displayed on the screen in green.
ing	Selecting elements of the same type Modifying will be activated. Checked properties can be changed or picked up from another element. Selecting elements of different types Definiton will be activated.
Pick Up>>	See Pick Up at 4.9.7 Line Elements.

4.9.9. Nodal Support

Nodal support elements may be used to model the point support conditions of a structure. Nodal support elements elastically support nodes, while the internal forces are the support reactions. Midside nodes of surface edges cannot be supported. References are used to arbitrarily orient the *x* and *z* axes of the element. The x axis is directed from a reference point to the attachment node (the node to which it is attached).

You can specify the translational and/or rotational (torsional) stiffness values about the element axes. Nonlinear parameters can be assigned to each direction. To change the characteristics click one the three buttons (bidirectional, compression only, tension only) and set the resistance checkbox and specify a value if necessary.

Direction G Global C Referential C Beam/Rib Relative C Edge Relative	
Reference	Nonlinear Parameters Resistance
R _X [kN/m] = 1E+10 ▼	🛊 🕂 🛉 🗖 F _X [kN] =
R _γ [kN/m] = 1E+10	🛊 🕂 🛉 🗖 F _Y [kN] = 💽 并
R _Z [kN/m] = 1E+10 ▼	
R _{XX} [kNm/rad] = 1E+10	
R _{YY} [kNm/rad] = 1E+10	i + + F Γ M _Y [kNm] = →
R _{ZZ} [kNm/rad] = 1E+10 💌	i i i i i i i i i i i i i i i i i i i

The default stiffness values are 1.000E+10 [kN/m], [kNm/rad].

Ger The support elements are displayed on the screen in yellow (translational spring) or orange (rotational spring).

The support can be defined in the following systems:

- Global	- Beam/rib relative
- Reference	- Edge relative

Defines nodal support elements parallel to global coordinate axes. You must select the nodes that are identically supported, and specify the corresponding translational (R_X , R_Y , R_Z) and rotational (R_{XX} , R_{YY} , R_{ZZ}) stiffnesses.



You can define only one global support for a node. You cannot define nodal support for a midside node of a surface element.

Reference

Edge relative

Defines nodal support elements in the direction of a reference (point or vector). You must select the nodes that are identically supported, and specify the corresponding stiffness (translational R_{x} , and rotational R_{xx}).

The direction of the reference vector is defined by the element node and its reference point or reference vector in the following way:



Support elements oriented toward a reference point

Beam/rib relative Defines nodal support elements about local coordinate axes of beam / rib elements. You must select the beam / rib elements and the nodes that are identically supported, and specify the corresponding translational R_{xx}, R_y, R_z and rotational R_{xx}, R_{yy}, R_{zz} stifnesses.

Defines nodal support elements about local coordinate axes of surface element edges. You must select the surface elements and the nodes that are identically supported, and specify the corresponding translational R_{xx} , R_{yy} , R_z and rotational R_{xxy} , R_{yy} , R_{zz} stifnesses.







If one surface is connected to the edge the local coordinate axes of the edge are:

- x = the axis of the edge
- y = the axis is oriented toward inside of the surface element in its plane
- z = parallel with the *z* local axis of the surface element

If two surfaces are connected to the edge the local *z*-axis direction is bisecting the angle of surfaces. The *y*-axis is determined according to the right hand-rule.

If more than two surfaces are connected to the edge and you select one or two of them then support local system will be determined based on the selected surfaces.

- *Nonlinear* Nonlinear force-displacement characteristics can be specified for this element as follows: *behavior* compression only (very small stiffness in tension), tension only (very small stiffness in compression). A resistance value can be also be entered.
 - The non linear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account.
 - $\ref{eq:scalar}$ Nodal supports appear as brown (R_{Xr} , R_{Y} , R_{Z}) and orange (R_{XXr} , R_{YY} , R_{ZZ}) pegs in 3 orthogonal direction.



Use the *Calculate*... button to calculate the support stiffness (including the rotational stiffness) due to a column type support. The support stiffnesses are determined based on the end releases, material, and geometry of the column.

Calculating nodal support stiffness a column below and a column above the node can be specified separately. These column parameters can also be used in punching analysis (especially in the case of intermediate slabs). The columns and walls modeling the supports also appear in rendered view and the cursor can identify them.

- Modifying Selecting elements of the same type Modifying will be activated. Checked properties can be changed or picked up from another element. Selecting elements of different types Definiton will be activated.
 - *Pick Up>>* **See...** *Pick Up* at 4.9.7 Line Elements.

4.9.10. Line Support

-	-			
5	5	5	5	з
•	~	~	•	2

Global

Support 6	×
O Define O Modify	
Direction C Glgbal C Referential C Begin/Rib Relative C Edge Relative	××××
Reference	V Nonlinear Parameters
R _x [kN/m] = 1E+7	i + + F _x [kN] = →
R _y [kN/m] = 1E+7	± + + ► F _y [kN] = +
R _z [kN/m] = 1E+7	i
R _{xx} [kNm/rad] = 0	i + + + + + + + + + + + + + + + + + + +
R _{yy} [kNm/rad] = 0	i + + + F ► M _y [kNm] = + +
R _{zz} [kNm/rad] = 0	
Pick Up >> Calculation	OK Cancel

Line support elements may be used to model the line support conditions of a structure. Line support elements (Winkler type) are elastically supporting beams, ribs, or surface edges, while the internal forces are the support reactions.

You can specify the translational and/or rotational (torsional) stiffness values about the element axes. Nonlinear parameters can be assigned to each direction. To change the characteristics click one the three buttons (bidirectional, compression only, tension only) and set the resistance checkbox and specify a value if necessary.

The support can be defined in the following systems: Global Beam/rib relative Edge relative

The default stiffness values are 1.000E+07 [kN/m/m], or [kNm/rad/m].

Defines line support elements parallel to global coordinate axes. You must specify the corresponding translational (R_X , R_Y , R_Z) and rotational (R_{XX} , R_{YY} , R_{ZZ}) stiffnesses.

Beam/Rib relative Defines line support elements for beam/rib elements in their local coordinate system acting as an elastic foundation. You must specify the corresponding translational R_x, R_y, R_z and rotational R_{xx}, R_{yy}, R_{zz} stifnesses.

The beams/ribs with line supports must be divided into at least four elements.

In addition, the following condition must be satisfied:

$$L \le l_k = \frac{1}{2} \min\left(\sqrt[4]{\frac{4E_x I_z}{k_y}}, \sqrt[4]{\frac{4E_x I_y}{k_z}}\right), \text{ where } L \text{ is the beam / rib length.}$$

Ø

AxisVM warns you if the condition is not satisfied (by one or more elements). In this case the Winkler's modulus of the defined elements are set to zero, therefore you can divide the elements and repeat the definition/modification process.

If you specify line supports the internal forces are linearly interpolated between the ends of the element, therefore the division of the elements is required.

Edge relative

Defines edge support elements relative to local coordinate axes of the edges. You must specify the corresponding stiffness (translational R_x , R_y , R_z and rotational R_{xx} , R_{yy} , R_{zz}). If one surface is connected to the edge the local coordinate axes of the edge are:

- x = the axis of the edge
- y = the axis is oriented toward inside of the surface element in its plane
- z = parallel with the *z* local axis of the surface element

If two surfaces are connected to the edge the local *z*-axis direction is bisecting the angle of surfaces. The *y*-axis is determined according to the right hand-rule.

If more than two surfaces are connected to the edge and you select one or two of them then support local system will be determined based on the selected surfaces.



Nonlinear Nonlinear force-displacement characteristics can be specified for this element as follows: *behavior* compression only (very small stiffness in tension), tension only (very small stiffness in compression). A resistance value can aslo be entered.

- The non linear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account.
- & Line supports appear as brown (R_x , R_y , R_z) and orange (R_{xx} , R_{yy} , R_{zz}) lines in 3 orthogonal direction.

Support stiffness calculation

Local Line Su	pport Calcula	tion		×
V Z	Materia C20/2 L (m) d (cm)	al :5 = 3,000 = 20,0	• Ø	End Releases
	R _x [kN/m/m] =	7,12E+4	R _{xx} (kNm/ra	ad/m] = 8,55E+3
	R _y [kN/m/m] =	2,85E+3	R _{yy} [kNm/r	ad/m] = 1E+0
	R _z [kN/m/m] =	6,41E+5	R _{zz} (kNm/ra	id/m] = 1E+0
			ок	Cancel

Use the *Calculate...* button to calculate the global or edge-relative line support stiffness (including the rotational stiffness) due to a wall type support. The support stiffnesses are determined based on the end releases, material, and geometry of the wall.

4.9.11. Surface Support

Define	C Modify	
		\land
		Nonlinear Parameters
R [kN/m/m ² 1	= 1E+4	
R. [kN/m/m ²]	= 1E+4 💌	
· · · · · · · · · · · · · · · · · · ·		

Surface support

Defines a surface support element (Winkler type elastic foundation) to surface elements. You must specify a translational stiffness in the surface element local coordinate system. The surface support behaves identically in tension and compression and is considered constant within the element.

You must specify the support stiffness R_x , R_y , R_z (Winkler's modulus) about the surface element local *x*, *y*, and *z* axes.

The default stiffness values are 1.000E+04 [kN/m/m], or [kNm/rad/m].

- *Nonlinear* Nonlinear force-displacement characteristics can be specified for this element as follows: *behavior* compression only (very small stiffness in tension), tension only (very small stiffness in compression), or with resistance (the same stiffness for compression and tension).
 - [©] The non linear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account.
 - $G_{\mathcal{S}}$ Surface supports appear as an orange square-hatched fill.

4.9.12. Edge hinge

Edge hinge can be defined between domain edges or between a rib and a domain edge. Select edge and a domain. Hinge stifness can be defined in the local system of the edge of the selected domain.

e hinge © Define	C Modify		ž ž t ×
K _X [kN/m/m]	= 1E+8 •	F _x [kN/m] = 0	•
K _Y [kN/m/m]	= 1E+8 •	F _y [kN/m] = 0	•
K _Z (kN/m/m)	= 1E+8 •	F ₂ [kN/m] = 0	•
K _{xx} [kNm/rad/m]	= 0 •	$M_{\chi} [kNm/m] = 0$	Ψ.
K _{yy} [kNm/rad/m]	= 0 •	$M_y [kNm/m] = 0$	T
K., [kNm/rad/m]	= 0 •	M_{z} [kNm/m] = 0	v

4.9.13. Rigid elements

Rigid elements may be used to model parts with a rigid behavior relative to other parts of the structure. Rigid elements may be used only in a linear static analysis.

The elements can be defined by selecting the lines that connect its nodes. The selected lines that have common nodes define the same rigid element. There is no limit to the number of nodes of any element.

The degrees of freedom of the nodes of a rigid element cannot be constrained (fixed).



Define Lets you define rigid elements. You must select the lines that connect the nodes attached to rigid elements. Recall that the lines with common nodes define the same rigid element.



You can join or split rigid elements using the modify command. If you select lines that connect nodes of different rigid elements, the elements will be joined. If you deselect lines of rigid elements interrupting their continuity, the respective elements will be split.

- A finite element cannot have all of its lines assigned to the same rigid body.
 If we want to calculate the mass of the body in a vibration analysis, place a node to the center of gravity, connect it to the body and make this line a part of the rigid body. Assign the mass of the body to this node.
- $G_{\mathcal{A}}$ The rigid elements are displayed on the screen with thick black lines.

4.9.14. Diaphragm

F

Using diaphragms means simplifying the model. Diaphragms are special rigid bodies where the relative position of the element nodes remain constant in a global plane. Diaphragms considerably reduce the amount of calculation. It can be an advantage running vibration analysis of big models. Diaphragms can represent plates totally rigid in their planes.

Definition

Select lines to define diaphragms. Each set of connecting lines will form a diaphragm.

Diaphragms are displayed as thick gray lines.

If you modify the diaphragm and select lines connecting to another diaphragm the two diaphragms will be merged into a single diaphragm. Selecting several groups of lines with no connection between the groups will break apart the original diaphragm.



Diaphragm	X
Plane of operat	tion
• 2	ΩY
0)	K <u>Z</u>
() O 2	<u>Y</u> Z
OK	Cancel

After definition you must set the working plane of the diaphragm. The relative position of element nodes remain constant in this plane. For rigid plates in the X-Y plane choose XY.

4.9.15. Spring

Sp	rings		×
	C Define C Modify		
	Direction Global By Geometry By Reference Element Relative Node Relative	N N N N N N N N N N N N N N N N N N N	
	Local × Orientat	tion i→j 💌	
	Stiffness	Resistance	
	K _χ [kN/m] = 1,5E+5	✓ F _X [kN] = 600,00]
	K _γ [kN/m] = 2E+2	F _Y [kN] =	
	K _Z [kN/m] = 2E+3	F _Z [kN] =]
	K _{XX(} [kNm/rad] = 0	🔽 M _X [kNm] =]
	K _{YY} [kNm/rad] = 0	M _γ [kNm] =	
	K _{ZZ} [kNm/rad] = 0	Mz [kNm] =]
	Pick Up >>	OK Cancel	

Spring element

The spring element connects two nodes of the model. The element has its own coordinate system. You can specify the translational and/or rotational (torsional) stiffness values about the element axes. The element can have nonlinear elastic stiffness properties.

The support can be defined in the following systems: Global / By Geometry / By Reference / Element relative / Node relative

Define You must select the nodes that are connected, and specify the corresponding stiffness (translational Kx, Ky, Kz and rotational Kxx, Kyy, Kzz). If a nonlinear elastic spring is to be defined, you can specify resistance values, for each internal force component.



Resistances will be taken into account only in a nonlinear static analysis, otherwise they will be ignored.

The nonlinear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account (that stay constant during the analysis).

4.9.16. Gap

Active In Tension	C In Compression
Active Stimess (kivm	
Inactive Stiffness [kN/m] 1E+0 🔽
Initial Opening [m] 0 🗖 By Geo
Auto Active Stiff	ness Adjustment
Penetration Allo	
Minimum I	nm] 15.5 _
wiii iiriidii ir	
Maximum [r	mm] 1E-2
Adjustment R	atio 100 👻
Adjustment it	

Gap element

The gap element is used to model point-to-point contact. The element has two states:

- one *active*, when it has a large stiffness value (simulates that a contact is achieved)
- one *inactive*, when it has a small stiffness value (simulates that no contact is achieved).

This contact model is approximate.

The gap element can be active in tension or compression. Typical force-displacement diagrams of gaps active in tension and compression are shown below correspondingly.



The gap element is a nonlinear element that can impose difficulties to the solution of the nonlinear problem, due to large changes of element stiffness when it changes status (active/inactive).

If the element is used to model regular contact problems, you may allow the element to auto adjust its stiffness, in order to smooth the large stiffness variations (at status changes) that can cause even divergence of the iterative solution process.

-*Р*_н

You must specify with two nodes:

Defining local x orientation is the same as for beam elements.

Active: The active state that can be tension (a tension bolt connection) or compression (contact of two plates)

Orientation (from one of its node to its other node)

Active stiffness: By default it is 1E+8 kN/m.

Inactive stiffness: By default it is 1E-2 kN/m.

Initial opening**penetration**: By default it is 0. The initial opening can be set based on element geometry as well (Check *By Geometry*). The initial opening is a positive or zero value. While the initial opening does not close, the gap is considered inactive.

Auto active stiffness adjustment:

If no adjustment is selected, the values below are not taken into account.

Minimum allowed penetration: You can set a minimum value for the penetration of the contact condition that is allowed. By default is 1E-05.

Maximum allowed penetration: You can set a maximum value for the penetration of the contact condition that is allowed. By default is 1E-05.

Maximum adjustment ratio: If the penetration is below the minimum, the active stiffness is softened by a maximum ratio entered here. If the penetration is between the two limits, no action is taken. If the penetration exceeds the allowed maximum, the active stiffness is hardened by a maximum ratio entered here. The default value is 100. In this case, the value of the adjustment ratio is the taken as: 1/100, 1/10, 1, 10, or 100.

[©] If the gap element is used in an analysis different from a nonlinear static analysis, the element will be taken into account as a spring with a stiffness corresponding to its initial opening. If the initial opening is zero, the active stiffness will be taken into account.

4.9.17. Link



Link elements

Link elements connect two nodes (N-N) or two lines (L-L) and have six stiffness components (defined in their coordinate system) that are concetrated on an interface (located between the connected nodes/lines). Its position can be entered relative to one node/line that is considered as reference.

Link elements can have a nonlinear parameter called limit resistance that limits the force they are able to transfer.



Node-to-Node (N-N) Link

Connects two nodes. The stiffness components are defined in the global coordinate system. Assigning zero value to a component the corresponding force or moment will not be transferred from one node to the other. The position of the interface can vary from 0 to 1 relative to the master node (selected by the user). If the location of the interface is = 0 the interface is at the master node. If it is = 1 the interface is at the opposite node. For any value greater than 0 or lower than 1 the reference is between the nodes.

Local z ReferenceInterface Location = 0.500Stiffness K_X [kN/m] = 1E10 K_Y [kN/m] = 1E10 K_Z [kN/m] = 1E10 K_Z [kN/m] = 1E10 K_Z [kN/m/rad] = 0 K_{YY} [kN/m/rad] = 0 K_{YY} [kN/m/rad] = 0 K_{ZZ} [kN/m/rad] = 0 K_{ZZ} [kN/m/rad] = 0	Direction © Global © By Geometry	2 8 × 8
Interface Location = 0.500 (*)StiffnessK_X [kN/m] = 1E10 •K_Y [kN/m] = 1E10 • $\ddagger + 4$ $F_X [kN] = $ K_Z [kN/m] = 1E10 • $\ddagger + 4$ $F_Z [kN] = $ K_{XX} [kN/m/rad] = 0 • $\ddagger + 4$ $F_Z [kN] = $ K_{YY} [kN/m/rad] = 0 • $\ddagger + 4$ $M_X [kN/m] = $ K_{ZZ} [kN/m/rad] = 0 • \checkmark $\ddagger + 4$ $m_X [kN/m] = $ K_{ZZ} [kN/m/rad] = 0 • \checkmark $\ddagger + 4$ $m_X [kN/m] = $	Local z Reference	V
Stiffness Resistance K_X [kN/m] = 1E10 \checkmark K_Y [kN/m] = 1E10 \checkmark K_Z [kN/m] = 1E10 \checkmark K_{XX} [kN/m/rad] = 0 \checkmark K_{YY} [kN/m/rad] = 0 \checkmark K_{ZZ} [kN/m/rad] = 0 \checkmark K_{ZZ} [kN/m/rad] = 0 \checkmark K_{ZZ} [kN/m/rad] = 0 \checkmark	Interface Location = 0.500	
K_X [kN/m] = 1E10 Image: state stat	Stiffness	Resistance
K_Y [kN/m] = 1210 Image: space state stat	K _χ [kN/m] = 1E10 💌	i + 4
K_Z [kN/m] = 1E10 Image: the system of the system o	K _Y [kN/m] = 1E10 💌	i
K_{XX} [kNm/rad] = 0 Image: Harmonic field of the second s	K _Z [kN/m] = 1E10 ▼	i + 4 □ F _Z [kN] =
K_{YY} [kNm/rad] = 0 Image: Marked for the second s	K ₃₀₀ [kNm/rad] = 0	i + + - + - ↓ M _X [kNm] =
K _{ZZ} [kNm/rad] = 0 ▼	K _{YY} [kNm/rad] = 0	i + + - + + − − M _γ [kNm] =
	K_{ZZ} [kNm/rad] = 0	↓ + + I M _Z [kNm] =

Typical applications are: main girder-purlin connection; some types of grillage connections; St. Andrew bracing connections; etc.

Example: A main girder-purlin connection (**see**... SteelFrame.axs in the examples folder) Let assume that the vertical axis is Z being parallel to the local z axis. The main girder is an IPE-400 in X-Z plane, the purlin is an I-200. You would like to transfer forces from the purlin to the main girder but not the moments.



These elements are represented by their line of gravity. The link has to be placed between these two axes at their point of intersection (if seen from above). Therefore, this link has to be assigned to a vertical line having a length equal to the distance of axes i.e. 30 cm (40/2 + 20/2). Select the node on the main girder to be the master node of the link. The inter-face always has to be placed at the actual point of contact. In this case the interface is located 20 cm far (40/2) from the master node (i.e. the main girder axis). So the interface position is 20/30 = 0.666. You assume that the connection is fixed against displacements but can rotate. Therefore, you enter 1E10 for translational stiffnesses and 0 for rotational ones. If the purlins are supported only by these links you have to enter KYY=0.001 or a similar small value to eliminate rotation around the main girder axis.

Nonlinear parameters

inear Nonlinear parameters can be assigned to each nonzero stiffness component. To change the characteristics click one the three buttons (bidirectional, compression only, tension only) and set the resistance checkbox and specify a value if necessary.

Line-to-Line Link

Connects two lines with three nodes each that can be rib elements and/or edges of surface elements. A line-to-line link has 6 nodes. The stiffness components are defined in the local coordinate system of the link that is in the plane of the link element with the x local axis parallel to the master line, and the local z axis oriented toward the other line in the plane of the link and is orthogonal to the local x axis.



Assigning zero value to a component the corresponding force or moment will not be transferred from one node to the other. The position of the interface can vary from 0 to 1 relative to the master line (selected by the user).

If the location of the interface is 0, the interface is at the master line (at the start point of the arrow).

If it is 1 the interface is at opposite line (at the end point of the arrow). For any value greater than 0 or lower than 1 the interface is between the lines.

Typical applications are: floor-wall hinged connections; semi-composite / full-composite layered beams; Semi-rigid rib-shell connections; etc.

Interface Location = 0.500		onlin + +	iear I 4 4	Param	neters Re F _x [kN/m] =	sistance	
Stiffness K _x [kN/m/m] = 1E8 K _y [kN/m/m] = 1E8 K _z [kN/m/m] = 1E8	· ···· ····	onlin + +	iear I 4 4	Param	$\mathbf{F}_{\mathbf{X}} [kN/m] = \begin{bmatrix} \mathbf{F}_{\mathbf{X}} \\ \mathbf{F}_{\mathbf{X}} \\ \mathbf{F}_{\mathbf{X}} \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{\mathbf{X}} \\ \mathbf{F}_{\mathbf{X}} \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{\mathbf{X}} \\ \mathbf{F}_{\mathbf{X}} \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{\mathbf{X}} \\ \mathbf{F}_{\mathbf{X}} \end{bmatrix}$	sistance T	
K _x [kN/m/m] = 1E8 K _y [kN/m/m] = 1E8 K ₂ [kN/m/m] = 1E8		+ +	4 4		$F_{x} [kN/m] =$		
K _y [kN/m/m] = 1E8		÷	4	Г	E [kblóp] -		
K _z [kN/m/m] = 1E8					y fusionij -	T	
	ş.	÷	4		F _z [kN/m] =	-	
K _{xx} [kNm/rad/m] = 0		$_{ op}^{\perp}$	4	Г	M _x [kNm/m] =	~	
K _{yy} [kNm/rad/m] = 0		$^{\perp}_{\top}$	4	Г	M _y [kNm/m] =	~	
K _{zz} [kNm/rad/m] = 0		$\stackrel{\perp}{\top}$	4	Γ	M _z [kNm/m] =	~	
Overwrite other existing elemnts with a link element							

Example: A floor-wall hinged connection.

Let's assume that the vertical axis is Z, the wall is in Y-Z plane, the floor is parallel to the X-Y plane and walls are represented by shell elements. Floor thickness is 15 cm. You would like to transfer forces from the floor to the wall but not the moments.



Elements are represented by their middle plane. The wall has to reach until the bottom plane of the floor. Links have to be placed between the upper wall edge and the floor edge. In this case the link elements have to be in the plane of the wall. The distance between the edges is 7.5 cm (15/2). Select wall edge nodes to be the master nodes. The interface has to be at the actual point of contact which is in the bottom plane of the floor and is 0 cm far from the master node. Therefore enter 0 for the interface position. You assume that the connection is fixed against displacements but can rotate. Therefore, you enter 1E10 for translational stiffnesses and 0 for rotational ones.

Nonlinear A limit resistance can be specified for each corresponding component with non-zero *parameters* stiffness.

When used in conjunction with domains the following steps can be followed to define lineto-line link elements:

- 1. Define the domains (See... 4.9.4 Domain) and connect the cor-responding opposite nodes of the domains with lines (the number of nodes on the edges of the domains should be equal).
- 2. Select the quadrilateral between the domains. Click **OK** on the Selection Toolbar.
- 3. Select the master line of the link element. Click **OK** on the Selection Toolbar.
- 4. Define the link stiffness, and set the interface location. By default the interface is in the midpoint of the link element. The link element(s) are created.
- 5. Now you can mesh the domains. **See...** 4.11.1.2 Mesh generation on domain
- 6. Link elements are divided according to the domain mesh.



4.9.18. Nodal DOF (Degrees of Freedom)

Lets you constrain the six nodal degrees of freedom that are: translations (e_X , e_Y , e_Z and rotations (θ_X , θ_Y and θ_Z).

In the default setting no nodes have constrained degrees of freedom.

In the calculations, equilibrium equations will only be written in the direction of the free displacements (translations/rotations).

Any combination of the six nodal degrees of freedom (e_X , e_Y , e_Z , θ_X , θ_Y and θ_Z) can be selected. However, in many cases typical combinations of degrees of freedom can be used. In these situations, you can quickly apply a predefined setting by selecting it from the list box.

The following particular structures are listed:

Plane truss girder / Space truss / Plane frame/ Grillage / Membrane / Plate

Define a nodal DOF

Use the buttons to set the degrees of freedom. Button captions will reflect the current value. Changes will be applied only to those nodal DOF which have their corresponding check-box checked. Unchecked components will retain their original values in the selection.

You have two options to change nodal DOF.

Overwrite

The new setting overwrites the existing degrees of freedom settings of the selected nodes.

Union

Performs a union set operation with the set of the new degrees of freedom codes and the set of existing degrees of freedom codes of the selected nodes. This option is useful in the definition of symmetry conditions.

Nodal Degrees of Freedor	m X
Overwrite C	Union
Free node Fixed node Truss girder in Plane X-Y Truss girder in Plane X-Z	<u> </u>
Space truss	•
	Prescribed Displacement 👻
✓ e _X Constrained ✓ e _Y Constrained	z
▼ e _Z Free	¢ ¢ v
V θ _X Free	€×
₩ ^θ Z Constrained	
Pick Up >>	OK Cancel

Example of union

	ex	еч	ez	θx	θγ	θz
initial code:	free	constr.	free	constr.	free	constr.
new code:	free	free	free	constr.	constr.	constr.
resulting code:	free	constr.	free	constr.	constr.	constr.

The six nodal degrees of freedom (e_X , e_Y , e_Z , θ_X , θ_Y and θ_Z) are set by a six digit code comprised of f (free) and c (constrained) symbols.

Each digit corresponds to one degree of freedom component. By default the nodes are considered free (all digits are f-free symbols). By setting a digit to c (constrained) the corresponding degree of freedom component is constrained.

The default DOF code of a node is [f f f f f f].

- ^(*) The loads that apply in the direction of a constrained degree of freedom are not taken into account. Loads in the direction of the constrained degrees of freedom will appear in the table of unbalanced loads.
- $G_{\mathcal{O}}$ The nodes with DOF different from [f f f f f] are displayed on the screen in cyan.

Notations: \uparrow free translation, \clubsuit free rotation about the specified axis.

Degrees of	Free	Degrees of	Free
Freedom	displacements	Freedom	displacements
Truss girders			
Truss girder in	Z	Truss girder in	Z
X-Y plane	Y Y	X-Z plane	Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ Υ
Truss girder in	Z	Space truss	Z
Y-Z plane	× ×		× ×
Frames			
X-Y plane frame	Z X X	X-Z plane frame	Z X X
Y-Z plane frame	Z Y X X		

1	2	3	4	5	6
ex	еч	ez	θx	θγ	θz

Degrees of	Free	Degrees of	Free
Freedom	displacements	Freedom	displacements
Grillages			
Grillage in X-Y	Z	Grillage in X-Z	Z
plane	× x x x x x x	plane	X X
Grillage in Y-Z	Z		
plane	×××		
Membranes			
Membrane in X-Y	Z	Membrane in X-Z	Z
plane	X X	plane	Ϋ́ς Υ΄
Membrane in Y-Z	Z		
plane	Y X		
Plates			
Plate in X-Y plane		Plate in X-Z plane	Z Y X X
Plate in Y-Z plane	Z * Y X		
Symmetry			
X-Y symmetry plane	Z	X-Z symmetry plane	
Y-Z symmetry plane	Z Y		

Pick Up>> Degrees of freedom can be picked up from another node and assigned to the selected nodes.

4.9.19. References



Lets you define reference points, vectors or axes, and planes. The references determine the orientation of the local coordinate systems of the finite elements in the 3D space. The local coordinate system of the elements defined with the references is used to define cross-sectional properties and to interpret results.

The element properties are defined and the internal forces $(N_x, V_y, V_z, T_x, M_y, M_z)$ for beams, m_x, m_y, m_{xy} for plates, n_x, n_y, n_{xy} for membranes, etc.) are computed in that local system.

Quick modify: Clicking on the symbol of a reference the Table Browser is invoked displaying the table of the references. The reference vector and axis can be defined by two points, the reference plane by three points. When closing the table the reference vectors, and axes are normalized with respect to 1.

```
Gev Color codes: x = \text{red}, y = \text{yellow}, z = \text{green}.
```

The following references can be used:

Automatic Automatic references for truss and beam elements:

references A reference vector is generated and assigned to the truss and beam elements as follows:

If the axis of the element is parallel with the global Z axis the reference vector will be parallel to the global X axis.

In any other case it will be parallel with the global Z axis.

For arcs: if the arc plane is parallel to the global X-Y plane, automatic reference is perpendicular to it and points to the +Z direction. If the arc is in a different plane its reference vector is in the arc plane and points outwards from the arc centerpoint.

Automatic references for rib elements:

If the rib is independent the reference vector will be generated and assigned to the element as for the beam elements.

If the rib is connected to a surface element, the generation of the reference vector is as follows:

The reference vector will be parallel to the bisector of the local z axes (normal to the surfaces) of the surfaces that have the rib element attached.

Automatic references for domains and surface elements:

Reference vectors will be generated and assigned to the surfaces as follows:

Local x-axis reference

If the plane of the surface is parallel with the X-Y plane the reference vector for the x local axis will be generated as a vector parallel with the global X axis.

In any other case, it will be parallel with the intersection line of the surfaces and X-Y plane. *Local z-axis reference*

If the plane of the surface element is parallel to the Z axis, the generated reference will be a vector oriented toward the origin of the global XYZ system. In any other case it will be parallel with the global Z axis.

The *Edit / Convert automatic references* menu item converts automatic references into reference vectors.

Reference point

Reference point is used to define the orientation (local coordinate system) of beam, rib, support, and spring elements or to define the positive local x and z axes of surface elements. The reference points are defined (by its coordinates) in the global coordinate system.

 G_{C} The reference points are displayed on the screen as small red + symbols.

Beams, ribs, and springs:

The reference point and the element's local x axis defines the local x-z plane. The positive local y and z axis direction is determined by the right-hand rule.



Surface elements:

The positive local z axis is oriented toward the half-space in which the reference point is located, and is perpendicular to the element's plane. Once the local x-axis is defined local y-axis is determined according to the right hand-rule.



The local *x* axis will be oriented in the direction of the reference point.

In the case of a surface element the reference point must be located in the plane of the element.



Supports:

In the case of a support element you can use a reference point to define local x axis.



Reference vector

Lets you define the local x axis for surface, support, and spring elements. Also defines the orientation of local z coordinate axis of beam, rib and spring elements.



 $G_{\mathcal{O}}$ The reference vectors are displayed on the screen as red arrows.

Surfaces:

The local x axis will be parallel with the reference vector. In the case of a surface element the reference vector must be parallel with the plane of the element.

The orientation of local *z*-axis can also be defined by a reference vector.



Supports:

In the case of a support element you can use a reference vector to define local x axis.



Beams, ribs, and springs:

The reference vector and the element's local x axis defines the local x-z plane. The positive local y and z axis direction is determined by the right-hand rule.







 $G_{\mathcal{O}}$ The reference axises are displayed on the screen as red arrows.

Reference Plane Reference plane is used to define the local x axis of surface elements, that will be parallel to the intersection line of the reference plane and the plane of the element. The reference plane must not be parallel with the plane of the element.



Reference angle

Yβ

Rotation of truss / beam / rib cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel with the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

 $G_{\mathcal{S}}$ The reference plane is displayed on the screen as a red triangle.

4.9.20. Creating model framework from an architectural model

	/	λ.	٦
17			Ξ

This icon starts the conversion operation of the architectural model if previously an IFC file (*.IFC) was loaded by *File / Import* (See ... 3.1.6 Import.) as a background layer.

Architectural model	×
I Slab I Wall Column I Beam I Roof	✓ Stories ✓ 5 ↓ 4 □ 3 ✓ 2 ✓ 1 ✓ 0
🗖 Refresh All	
Create Model Framework >	
Delete Objects >	Close

Display Select architectural project stories and element types you want to be displayed.

The Use the built-in Filter to enhance selection.

If you create model framework or delete objects and nothing is selected the Selection Toolbar appears. Click the Property Filter icon to select beams and columns within a certain range of section size according to their minimum side length or select walls or slabs within a certain range of thickness.

If you want to restore the whole range click the button at left bottom.

If the *Only objects without static model* is checked only elements not having static model will be selected.

Delete Objects Click this button to delete selected architectural model objects.

Beleting an architectural object having a static model will not delete its associated static model.

Create Model Framework Model framework will be created from selected layer elements. Columns will be reduced to their axis, walls, slabs and roofs will be reduced to their center plane. Framework nodes and lines become part of the AxisVM model and are independent of the background layer.

Parts will automatically be created for levels and object types and the elements created for the static model will be included in the appropriate parts.

Hinged wall connections can be modeled using edge hinges when creating a model framework from the architectural model.

Slab	Create Model Framework	Floors can be defined as plates or shells. Assign a material and a thickness. For layered floors, the thickness of the layers will appear in the layer list. You can select the layers that you want to take into account.
	Slab 8 elements OK Cancel	

You can assign properties to the selected architectural objects as follows:

Wall



Walls can be defined as membranes or shells. Assign a material and a thickness. For layered walls you can choose to apply the thickness of the load bearing layer, the total thickness or a

Apply bottom support:

custom value.

You can automatically assign a support to the bottom edge of the selected walls.

Convert walls to supports: You can convert wall objects to supports by enabling this checkbox. The support will be placed at the top edge of the corresponding wall. The support stiffness will be computed based on the top and bottom end releases.

Column



Column objects are always converted to beam elements. Assign a material and a cross-section. If Auto is selected the cross-section is created based on the geometrical description of the architectural object.

You can assign a support to the bottom of the column.

Convert	columns to	o supports:	The select	ted colum	ın objects	s can be	converted	to	suppo	rts.
Support s	stiffness is	established	based on t	the end re	eleases. Su	apports	will be plac	ed a	at the	top
of the col	umn.									

×

Beam

Roof

	r Touc	i Frameu Y	vork	1	,	
Slab	Wall	Column	Beam	Roof		
	/pe					
	Rib					
) Beam					
Mat	erial					
C2	0/25		· [6]			
Cro	ss-Secti	on				
A	ıto.	•	ាញាំ	표 🗌		
Bear	n			OK		Cancel

Create Model Framework

Material

C20/25

Roof

7 elements

Object Layers

Slab Wall Column Beam Roof

Thickness [cm] = 15,00 💌

✓ 15,00 cm: [Load Bearer Core]

J 🔊

OK

Cancel

Beam objects are always converted to beam elements. Assign a material and the crosssection.

If Auto is selected the cross-section is created based on the geometrical description of the architectural object.

Roof objects are always converted to shell elements. Assign a material and a cross-section. For layered roofs, the thickness of layers will appear in the layer list.

You can select the layers that you want to take into account.

4.9.21. Modify

Lets you modify the definition of the selected elements.

- 1. Holding the [Shift] key down, select the elements to modify. You can use the Selection icon as well.
- 2. Click the element's icon on the Elements Toolbar.
- 3. In the element's dialog window check the properties you want to modify. Property fields show the common value in selection. If selected elements have different values the field is empty.
- 4. Modify the respective properties as desired.
- 5. Click the **OK** button to apply the modifications and exit the dialog window.
- [©] In fact, the modification is similar to the element definition, but does not assign properties to undefined geometrical elements and allows access to a specific property without altering others. You can switch to the element definition radio button to define all properties of all the selected elements, lines or surfaces.
- *Immediate mode* If the Geometry or Elements tab is active click a finite element to modify its properties. If more finite elements have been selected they can be immediately modified by clicking one of them. If you click an element which is not selected, selection disappears and you can modify the element you clicked. If you click on a node its nodal degrees of freedom can be edited immediately.

You can also modify the properties using of Property Editor. **See...** 3.5.1 Property Editor

4.9.22. Delete

[Del] See... 3.2.7 Delete
4.10. Loads



Lets you apply various static loads for static and buckling analysis, and define concentrated masses for vibration analysis.

4.10.1. Load Cases, Load Groups

Load Case

Lets you set the current, create new, and modify or delete existing load cases. Any load you create will be stored in the current load case. In the professional version the number of load cases is not limited. In the standard version a maximum of 99 cases can be created. Load groups can also be created from the different load cases.

Load Groups Load Cases	×
B Ungrouped UI ST1 UI ST2 PERM1	Load Case New Case IIII III IIIIIIIIIIIIIIIIIIIIIIIIIIII
	(for timber design)
	Load Group (Eurocode) New Group Image: Second Se
X Delete	OK Cancel

New Case You must assign a different name to each case. The following are the three types of load cases that you can choose from when you want to create a new load case:

1. Static

The static load case can be applied to static, vibration, and buckling analysis. In case of vibration analysis, the loads can also be taken into account as masses.

The load case can be included into a load group. When calculating the critical load combination, the load case will be taken into account according to the parameters of the load group to which it belongs.

Critical combination can be determined only from the results of a linear static analysis.

2. Influence line

Lets you apply a relative displacement load to obtain the influence line of a result component, of a truss or beam element.

When the influence line load case type is selected you can apply only the influence line load

3. Seismic

When selecting seismic load case type you can specify the parameters for calculation of earthquake loads. Prior to creating an seismic load case, you must perform a vibration analysis. Based on the mode shapes, and on the structural masses, AxisVM generates seismic loads case, in a k+2 number, where k is the number of available smallest frequencies. The two additional cases corresponds to the signs +, and -, that contain the critical combinations. **See**...4.10.20 Seismic Loads

When selecting seismic load case the only icon available on the Toolbar will be Seismic parameters.

4. Pushover

When selecting pushover load case type you can specify parameters for generating load distributions that can be used in pushover analyses. Prior to creating a pushover load case, you must perform vibration analysis. Based on specified mode shapes AxisVM generates nodal forces on each node of the model. A total of four load cases are generated initially. They represent a uniform (U) and a modal (M) distribution in the direction of each of the horizontal axes (X and Y by default). The uniform load distribution option generates nodal forces proportional to the masses assigned to each node in the model. The modal load distribution uses the mode shape weighed by the masses at each node to generate the nodal force distribution. In both cases the sum of forces generated is 1 kN in the same horizontal direction.

See details... 4.10.21 Pushover loads

When selecting pushover load case the only icon available on the Toolbar will be Pushover parameters.

5. Tensioning

If tensioning calculation according to the current design code is supported, tensioning load cases can be created. These load cases always get into a tensioning load group. After defining a load case with the name *name*, two load cases will be created. *name-T0* will contain the equivalent load calculated for the end of tensioning process, *name-T1* will contain long term values of the equivalent load. Any of these load cases can be selected to define tensioning. After definition just loads for *name-T0* will be calculated as static analysis results are required to determine the long term equivalent loads. **See details...** 4.10.22 Tensioning

When selecting tensioning load case the only icon available on the Toolbar will be Tensioning.

6. Moving load

In this type of load case only moving (line or surface) loads can be defined. When defining a moving load a group of new load cases will be created. The number of these load cases is equal to the number of steps specified in the moving load definition dialog. Their name is created automatically like MOV_xx. As they get into a load group the most unfavourable effect of the moving load can be checked displaying the result of the critical combination. These auto-created load cases can be moved together only and only into another moving load group.

If more than one moving load is applied in the same load case the number of steps (and auto-created load cases) will be equal to the maximum number of steps specified. If the maximum number of steps is k, and another moving load has i steps (i < k), then this load will remain at the end of the path in steps i+1, i+2, ..., k.

See details... 4.10.23 Moving loads



৶

+-	t	+	

4MM

- When selecting moving load case the only icon available on the Toolbar will be Moving Load.
 - 7. Dynamic load case

Dynamic load cases can be used only if DYN module is available. After defining a dynamic load case and selecting it the *Loads* tab will allow definition of dynamic loads and nodal acceleration.

See details...4.10.24 Dynamic loads (for time-history analysis)

- *Dynamic load cases cannot be included in load groups and load combinations. Loads within dynamic load cases will be applied only in Dynamic analysis.*
- Load-durationTimber design module requires information on the load duration. So if a timber material has
been defined in the model load case duration class can be entered. (Permanent: > 10 years;
Long term: 6 months–10 years; Medium term: 1 week–6 months; Short term: < 1 week; In-
stantaneous; Undefined)
 - *Duplicate* Lets you make a copy of the selected load case under another name. You must specify the new name, and a factor that will multiply the loads while copying. The factor can be a negative number as well.
 - Selected loads can be copied or moved to another load case by changing load case during the copy or move process.
 - *Delete* Lets you delete the selected load case.

You can change the current load current case by selecting from the drop down list near the load case icon. Selection can be moved using the up and down arrow keys. This is the best way to overview moving load cases.



The name of the selected load case will appear in the Info window and the loads you define will get to this load case.

In case of choosing Tensioning load case only the Tensioning Icon will be active on the toolbar. Click on it then select the proper beam or rib elements, so the Tensioning Dialog will appear. **See...** 4.10.22 Tensioning



Click right mouse button over the list, select Order of load cases to get to a dialog setting the load case order. This dialog is also available in the Table Browser (*Format / Order of load cases*).

Order of The display order of load cases and groups can be changed by dragging the load case or load group within the tree. Display order is also used in the load combination table and reand groups sult load case combo boxes.

Right-clicking the tree displays a popup menu allowing other ordering options (alphabetical or creation order). Position of the ungrouped load cases in the tree can also be set.

Load group You can select the load group you want to place the current load case in from the dropdown list. Load case will immediately be moved to its new position in the tree view. You can also drag and drop load cases between load groups by mouse.

Setting the current load case :

向

Click on any existing load cases from the list which is on the left side of the Load Groups & Load Cases dialog window. The chosen load case will become the current case. Any subsequently defined loads will belong to the newly selected load case.

Safety Class Select the safety class of the building from the combo box. Changing the safety class may require changes in the incidental group factors $\gamma_{f,a}$, Ψ and Ψ_t .

183

Load Group

NL C

Load groups are used when generating of critical (design) values of the results.

New Group Lets you define a new load group. You must specify the name and the type (permanent, incidental, exceptional) of the load group, and the corresponding factors according to the current design code. Later you can specify which load cases belong to a specific load group. Clicking any icon within the New Group group box will create a new group in the tree and you can specify a name for it. Existing load group names will be rejected. After creating a load group you have to specify the value of its paremeters (like the partial factor, dynamic factor, simultaneity factor, etc.). A load case can be assigned to a load group by choosing a group from the dropdown list or dragging the load case under a load group in the tree. **See...** 4.10.2 Load Combination

The following load groups are allowed depending on the design code:

1. Permanent

Includes dead load, permanent features on the structure...

Include all load cases in combinations

All load cases from the group will be taken into account in all load combinations with their upper or lower partial factor.

Include the most unfavourable load case only

Only the most unfavourable load case will be taken into account from the load group with its upper or lower partial factor.

2. Incidental

Includes live load, wind load, snow load, crane runway load...

Can be simultaneous with exceptional goups

If checked load case(s) from the group can act together with a load case from an exceptional group in critical combinations.

Simultaneous load cases

Any number of load cases from the group can act simultaneously in critical combinations.

Mutually exclusive load cases

In a critical load combination only one load case from the group will be taken into account at one time.

3. Exceptional

Includes earthquake, support settlements, explosion, collision... Only one load case from the group will be taken into account in a load combination at one time. That load case must have the simultaneity factor of $\alpha = 0$.

4. Seismic load group (Eurocode, SIA 26x, DIN 1045-1, STAS and Italian code)

Only one load case from the group will be taken into account in a load combination at one time. That load case must have the simultaneity factor of $\alpha = 0$.

5. Tensioning load group (if tensioning can be calculated according to the current design code)

Tensioning load group is handled as a permanent load group. It can contain only tensioning load cases. Both load cases for the same tensioning (name-T0 and name-TI) cannot be included in any load combination.

6. Moving load groups

Auto-created load cases for the moving loads in a moving load case get into a moving load group.

Load types

The following loads can be applied to the elements:

Load	Element
Concentrated	node, beam
Line (distributed)	beam, rib, plate, membrane, shell
Edge (distributed)	plate, membrane, shell
Dead load	truss, beam, rib, plate, membrane, shell
Temperature	truss, beam, rib, plate, membrane, shell
Fault in length	truss, beam
Tension/Compression	truss, beam
Forced support displacement	support
Fluid	plate, shell
Seismic	node
Influence line	truss, beam
Tensioning	beam,rib
Moving	Beam, rib, plate, shell

4.10.2. Load Combination



 poids Beam Loads 		+ x 1	ð Cl	III <i>8</i>								
- Distributed Load Beam Self Weigh	Loa	d Combination	s									
⊕ neige (389)		Name	Туре	poids	neige	vf1	vf2	vf3	vo1	vo2	vo3	Comment
± vf1 (456)	14	Co #14		1.35	1.50	0	0	0	0	0	0.80	
the test of test	15	Co #15	-	0.80	0	1.50	0	0	0	0	0	
± vf3 (467)	16	Co #16	-	0.80	0	0	1.50	0	0	0	0	
+ V01 (416)	17	Co #17	-	0.80	0	0	0	1.50	0	0	0	
± vo2 (417)	18	Co #18	-	0.80	0	0	0	0	1.50	0	0	
- Load cases (8)	19	Co #19	-	0.80	0	0	0	0	0	1.50	0	
-l oad Groups (8)	20	Co #20	-	0.80	0	0	0	0	0	0	1.50	
Load Combinations (276)	21	Co #21	ULS	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Incidental combinations	22	Co #22	ULS	1.35	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Weight Report	23	Co #23	ULS	1.00	1.35	1.00	1.00	1.00	1.00	1.00	1.00	
IBRARIES	24	Co #24	ULS	1.35	1.35	1.00	1.00	1.00	1.00	1.00	1.00	
Material Library	25	Co #25	ULS	1.00	1.00	1.35	1.00	1.00	1.00	1.00	1.00	
🗄 Cross-Section Library 💌	26	Ço #26	ULS	1.35	1.00	1.35	1.00	1.00	1.00	1.00	1.00	
► 1												

Lets you define load combinations of the defined load cases. You can specify a factor for each load case in a load combination.

The results of a load combination will be computed as a linear combination of the load cases taking into account the specified load case factors. A zero factor means that the respective load case does not participate in the load combination.



Inserts a load combination table to the current report.

Calculates all critical combinations based on load groups and transfers them into the load combination table.

^{CF} You can also define load combinations after you have completed a linear static analysis. Then, when required the postprocessor computes the results of these load combinations. In case of nonlinear static analysis, AxisVM first generates the combination case, and then performs the analysis (a load combination at a time).

Automatic load combination

The program builds all possible combinations depending on the load groups parameters and the equations of the current design code.

The minimum and maximum result values of these combinations are selected as critical (design) values.

Internal forces (Eurocode) ULS

Permanent and Incidental:
$$\sum_{i=1}^{n} \sum_{j=1}^{n} \sum_{i=1}^{n} \sum_{j=1}^{n} \sum_$$

$$\sum \gamma_{Gi} G_{ki} + \gamma_{Qj} Q_{kj} + \sum_{i \neq j} \gamma_{Qi} \Psi_{0i} Q_{ki}$$

Exceptional:

$$\sum G_{ki} + A_d + \Psi_{1j}Q_{kj} + \sum_{i \neq j} \Psi_{2i}Q_{ki}$$

Seismic loads:

Eurocode, SIA and DIN

STAS

Rare (SLS1):

 $\sum G_{ki} + Q_{kj} + \sum_{i \neq j} \Psi_{0i} Q_{ki}$

Common (SLS2):

$$\sum_{i \neq j} G_{ki} + \Psi_{1j}Q_{kj} + \sum_{i \neq j} \Psi_{2i}Q_{ki}$$
Quasipermanen (SLS3):

$$\sum_{i \neq j} G_{ki} + \sum_{i \neq j} \Psi_{2i}Q_{ki}$$
STAS:
$$\sum_{i \neq j} G_{kj} + 0.6\gamma_{I}A_{Ek} + \sum_{i \neq j} \Psi_{2i}Q_{ki}$$

Critical load combination method for internal forces and for displacements are selected automatically. Critical load combination method for displacements depends on the type of structure you are modeling. Click Result Display Parameters on the Static Toolbar then click Select. If your current design code is Eurocode you can set the critical combination formula at the bottom of the dialog.

 $\sum G_{ki} + A_{Ed} + \sum \Psi_{2i} Q_{ki}$

 $\overline{\sum} G_{ki} + \gamma_I A_{Ek} + \sum \Psi_{2i} Q_{ki}$

Italian code

 Φ

Seismic loads: see above at Internal forces

Combination of seismic loads with other load types:

	$\sum G_K +$	$\gamma_I \cdot E + \sum_i \left(\Psi_{ji} \cdot Q_{Ki} \right)$
here:	γ_I	importance factor
	Ε	seismic load
	G_K	characteristic value of permanent loads
	Q_{Ki}	characteristic value of variable loads
	ψ_{ji}	ψ_{2i} (ULS) combination factor for quasipermanent Q_i
	ψ_{0i}	(DLS) combination factor for rare Q_i

4.10.3. Nodal Loads

W

Lets you apply forces / moments to the selected nodes. You must specify the values of the load components F_X , F_Y , F_Z and M_X , M_Y , M_Z , in the global coordinate system.

If you apply a nodal load to a node that is already loaded, you can overwrite or add it to the existing load.



Nodal loads on Node 203	×
C Define C Modify	
C Global C Referential	Z ^M 4† ↑ F ² _ = [™] M ₂
C Overwrite 💿 Add	F _x M _x
Reference	
F _X [kN] = 5,00	$M_{\chi} [kNm] = 0$
F _Y [kN] = 4	M _Y [kNm] = 0
F _Z [kN] = -10 ▼	M _Z [kNm] = 0
Pick Up >>	OK Cancel

Modify position

The positive directions are according to the positive directions of global coordinate axes.

Modify nodal loads You can select, move, copy or modify the load independently of the node.

- 1. Select the loads you want to move together.
 - 2. Grab any of them by pressing the left mouse button.
 - 3. Move them to their new position.
 - 4. Click the left mouse button or use a command button. (Enter or Space).
- *Modify value* 1. Select the load.
 - 2. Click the Nodal Load icon on the Toolbar.
 - 3. Change the values
 - Nodal loads can be moved onto a beam, a rib or a domain.

Signs of the load values are calculated according to the right hand rule.

- *Constrained Components applied in the direction of a constrained degree of freedom will be not taken into account in the analysis.*
- $\mathop{{\rm G}}\nolimits{{\rm C}}$ The forces are displayed on the screen as yellow arrows, the moments as green double arrows.

4.10.4. Concentrated Load on Beam

Lets you apply concentrated forces/moments to the selected beam finite elements. You must specify the values of the load components Fx, Fy, Fz, Mx, My, Mz in the local or global coordinate system.

If you apply a concentrated load to a node that is already loaded, you can overwrite or add it to the existing load. Concentrated loads can be

selected, moved, copied, modified independently of the beam. Modify load values like in case of nodal loads.

Concentrated Loads o	n Beams	X
Define	C Modify	
Direction © Global © Local		
C Overwrite	Add	
Position C By Length € By <u>R</u> atio	a = 0,5	•
F _X [kN] = 0	•	M _X [kNm] = 0
F _Y [kN] = 0	•	M _Y [kNm] = 0
F _Z [kN] = -15,0		$M_Z [kNm] = 0$
Pick Up >>		OK Cancel

The positive directions are in accord with the positive directions of the local or global coordinate axes.

 $_{\mbox{GC}}$ The forces are displayed on the screen as yellow arrows, the moments as green double arrows.

4.10.5. Point Load on Domain

Applies a point (concentrated) load at the location of the cursor if it is over a domain. You can also enter the location of the load by its coordinates. You can place loads by clicking the left mouse button or pressing any of the command buttons. **See...**4.7.2 Entering Coordinates Numerically

The direction of the load can be:

- Global (with respect to the global coordinate system)
- Local (with respect to the local (element) coordinate system)
- Reference (with respect to a reference)



Modify point load on You can modify the location and value (intensity) of the load: domain

Modify position

- 1. Select the load with the cursor (a load symbol appears beside the cursor).
- 2. Keep left mouse button depressed.
- 3. Move the mouse or enter the relative coordinates to move the load to a new location.
- 4. Release left mouse button to set the load in its new location.

Modify value

шIJ

Ŧ

- 1. Select the load with the cursor.
 - 2. Click the left mouse button.
 - 3. Enter the new load values in the dialog.
 - 4. Click on the Modify button to apply the changes and close the window.

The load value can also be changed in the Table Browser.

Modifying domain mesh leaves the concentrated loads (applied on the domains) unchanged.

4.10.6. Distributed line load on beam/rib

Lets you apply constant or linearly distributed forces and torque to the selected beam/rib finite elements. You can apply multiple distributed loads to a beam/rib in the same load case.

Line loads can be selected, moved, copied, modified independently of the beam or rib. Modify load values like in case of nodal loads.

You must specify the distribution, the location and the values of the load components in the local or global coordinate system as follows:

Distributed Loads	on Beam 1 O Modify	ز
Direction © Global A © Global P © Local	Jong Element rojective L _{max} = 14.013 m	
C Overwr Type	te $\bigcirc Add$ Position $\bigcirc B_{Y} Length$ $\bigcirc B_{Y} Ratio$ $d_{1} [m] = 3.503$	d ₂ (m) = 10.510
4	Startpoint p _{X1} [kN/m] =0 p _{Y1} [kN/m] =0	Endpoint p _{X2} [kN/m] =0 p _{Y2} [kN/m] =0
	p _{Z1} [kN/m] =-3	p _{Z2} [kN/m] = -2



Statung value: p_{x1} , p_{y1} , p_{z1} , m_{TOR1} End location: x_2 relative to the *i*-endEnd value: p_{x2} , p_{y2} , p_{z2} , m_{TOR2}

If the load is projective, the value of the load that is applied to the beam/rib is $p \cdot \sin \alpha$, where α is the angle of the load direction and the beam/rib axis.

For rib elements you can apply line loads distributing along the entire length of the rib only.

4.10.7. Edge Load

Ŵ

Lets you apply distributed (constant) loads to the selected edges of the selected surface elements.

If more than two finite elements are connected to the edge or they have different local coordinate systems you have to select both the edge and the finite element when you specify the local load. Load will be defined in the local system of the selected element.

Edge Load on Plate 177	×
Define C Modify	
Direction Gobal On Surface Gobal Projective Cocal	
© Overwrite C Add p _x [kN/m] = 0	
p _y [kN/m] = 0 v p _z [kN/m] = -100 v	
Pick Up >>	OK Cancel



In the case of shell elements, the load that is applied in global coordinate directions can have a projective distribution. If the load *p* is projective, the value of the load that is applied to the shell is $p \cos \alpha$, where α is the angle of the load direction and the element plane normal.

4.10.8. Domain Line Load

IIII

Applies a uniform or linear distributed line load over a domain.

The direction of the load can be global projective, global along element, edge relative or surface relative. The m_x is always the torsional moment (around the application line of the load). Set load components and placement method then draw the load (or click the lines) to place it.

Direction	p _{X1} [kN/m] = 0	▼ p _{X2} [kN/m] =	0 🔹	Pick Up >>
Global Along Element	p _{Y1} [kN/m] = 0	▼ p _{Y2} [kN/m] =	0 🔹	
Global Projective	p _{Z1} [kN/m] = -1	▼ p _{Z2} [kN/m] =	-1.00 💌	
Edge Relative Surface Relative	m _{tor1} [kNm/m] = 0	✓ m _{tor2} [kNm/m] =	0 🔹	Close
)OOOJ * 1	5		



During definition of a complex polygon a pet palette appears with several geometry functions. These are: drawing a line, drawing a line as a tangent of an arc, drawing an arc with centerpoint, drawing an with a midpoint, drawing an arc with the tangent of the previous polygon segment, drawing an arc with a given tangent, picking up an existing line.

Distributed line load on an existing line or arc *Distributed line load on an existing line or arc.* Click any line or arc on the domain boundary or within the domain to apply the load previously defined. This type of load is associative. Moving the boundary or the internal line moves the load as well. Deleting the line deletes the load.

1



Line load by selection. Similar to the previous function but the load will be applied to the selected lines.

Modify of the load	You can modify the location and value (intensity) and any vertex of the load polyline:
Modify location	 Select the load with the cursor. Keep left mouse button depressed. Move the mouse or enter the relative coordinates to move the load to a new location. Release left mouse button to set the load in its new location.
Modify shape	 Move the cursor above the vertex (a load polyline vertex symbol appears beside the cursor). Click the left mouse button Drag the vertex to its new position after pressing the left mouse button. Click the left mouse button.
Modify value	 Select the load with the cursor (a load symbol appears beside the cursor). Click the left mouse button. Enter new load values in the dialogue window. Click on the Modify button to apply the changes and close the window.
	The load value can also be changed in the Table Browser.
Delete	Select the loads you want to delete and press Delete .
(F	Modifying domain mesh leaves line loads (applied on the domains) unchanged.

4.10.9. Surface Load

 \bigcirc

Distributed Load	on Shells		×
Define	C Modify		
Direction © Global C © Global F © Local © Overwith P _X [kN/m ²] = P _Y [kN/m ²] = P _Z [kN/m ²] =	n Surface rojective	Z WX	
Pick Up >>		OK Cancel	

The intensity of a distributed load on a surface element is constant.

Lets you apply distributed loads to the selected surface elements or domains.

^C Modifying domain mesh leaves the loads (applied on the domains) unchanged.

Element	Load in Local Directions	Load in Global Directions
	(in Local Coordinate System)	(in Global Coordinate System)
Membrane	x Abigaro to	
	y Burgerson and a second	
Plate		
Shell	X ABGEORGIA	X Z Z Z Z Z Z Z Z Z Z Z Z Z Z Z Z Z Z Z
	y Bolemon end	Y Z Z Z Ry TRY
	Z Bigging of the state	Z Z

4.10.10. Domain Area Load

Applies a mesh-independent area load to a domain.

The domain element type determines the load type and direction as follows.

For a membrane domain the load must be in the plane of the domain. For a plate domain the load must be perpendicular to the plane of the domain. For a shell domain any load direction is acceptable.

The load can be a global load on surface, a global projective load or a local load and the components will be interpreted accordingly.

You can select between constant or linear load intensities and set if loads disapper over holes or are distributed on the edge of the hole.



The first icon represents the option that loads over holes are not applied to the structure. The second one represents the option that loads over holes are distributed on the edge of the hole.

193

Loads disappear/ allowed on holes



2. Enter two diagonal end points of the rectangle by clicking or by coordinates. (This function is available only on the X-Y, Y-Z and X-Z planes)





- 1. Enter load components (p_x, p_y, p_z)
- 2. Enter three corners of the rectangle by clicking or by coordinates.



- 1. Enter load components (p_x, p_y, p_z)
- 2. Enter polygon vertices by clicking or by coordinates. In this latter case press an extra Enter after specifying the last position. If you enter the polygon by clicking on the domain close the polygon by clicking on the first vertex again or by double-clicking at the last vertex. Instead of the left mouse button you can also use Space or Enter key to enter polygon vertices.



Enter load components (p_x, p_y, p_z)
 Click on the domain

Distributed domain load

¹ The load will be distributed over the domain. The shape of this type of load will automatically follow any change in the domain geometry. Within a load case you can apply only one load of this type on a domain. New distributed domain load definition always overwrites the previous one.

Linear load

Steps of load definition in case of linear load:



The plane of the load intensity can be specified by load intensity values (p_1, p_2, p_3) at three points [(1), (2), (3)] in the plane of the domain. These points are the load value reference points. If you want to use the same reference points and values to many loads of different shape and position you can lock the reference points and values by clicking the Lock button. Loads are applied by entering an area.



Modify intensity 1. Place the mouse above the load contour (the cursor will identify the load).

- 2. Click the left mouse button. The area load windows appears.
- 3. Change the load intensity values.
- 4. Click on the Modify button to confirm the changes.

Multiple loads can be selected and modified this way.

Area load intensity and shape can also be changed in the Table Browser by changing the appropriate values in the load table.

Delete Select the loads to delete and press [Del]

Mesh-independent loads are not affected by removing or re-creating a meshes on domains.

4.10.11. Surface load distributed over line elements

Homogenous surface load can be placed over line elements (trusses, beams and ribs). Loads over trusses will be converted into loads on the truss end nodes.

1. Click the icon and select the load distribution range in the dialog.

Auto distributes the load over the elements under the load. Any new truss, beam or rib defined under the load will redistribute the load.



To selected elements only distributes the load over the selected elements only. Select lines using the selection toolbar. Distribution remains the same if a new beam or rib is defined under the load.



2. Define load polygon the same way as for a constant or linearly changing domain area load.



Load direction can be global on surface, global projective or local. Local directions are defined like automatic references for domains **See...** 4.9.19 References.

Enter load values into the edit fields. (p_X , p_Y , p_Z)

The load polygon can be a rectangle, a skewed rectangle or any closed polygon. The fourth method on the icon tollbar is to click lines of a closed beam/rib polygon. This way the load becomes associative. Moving the elements or their end nodes changes the load polygon accordingly.



Edit / Convert surface loads distributed over beams menu item converts loads created this way to individal beam loads.

4.10.12. Fluid Load

Lets you apply pressure loads characteristic to fluids to the selected plate or shell elements. The actual load is calculated from values computed at the corner of the elements.

Fluid loads created with the same definition will be handled as one load. So if you specified a fluid load on more than one element and click on the load contour on any of these elements the load will be selected on all of them and you can easily change the load parameters.

To change a fluid load only on certain elements use partial selection, i.e. draw a selection frame around the elements.

Load Vari	ation Directi	on		
С×	ΟY	€Z	P(Z ₄)	
Overw	rite 🔿 /	Add	P(Z ₂)	
(n] = 8	,00	p(Z ₁) [kN/m	²] = 0	
		 p(7)[kN/m	21 = 80.00	-
2 to 4 1 -		1-1-2710-000	· ·]	



4.10.13. Dead Load

- Lets you take the dead load of the elements (that have materials assigned) and domains into account in the analysis. The dead load is computed based on the cross-sectional, the mass density of the material, the gravitational acceleration *g*, and the length or area of the element. The load is applied as a distributed load in the direction of the gravitation vector.
- A dashed line is drawn along line elements or surface/domain contours. If load intensity labels are turned on a light blue G appears.

4.10.14. Fault in Length (Fabrication Error)

<u>≙L</u>∺

G

This load type is used when a structural beam element is shorter or longer than required due to a fault in manufacturing.

Lets you apply the load, which is required to force the shorter/longer beams to fit the distance of the corresponding nodes, to the selected elements.

You must specify the value of the manufacturing fault, dL [m]. A positive dL means that the beam is longer by dL.

Fault In Length of E	Beam 1	X
Define	C Modify	
dL [m] = 0,02	J. J.	1
Pick Up >>	OK Cano	el

The load has the same effect as the $dT^{-} = dL/(\alpha \cdot L)$ thermal load.

4.10.15. Tension/Compression

+<u></u>+

Lets you define an initial axial internal force in truss/beam elements. The load has the same effect as a $dT^{=} = -P/(\alpha \cdot E \cdot A)$ thermal load.



4.10.16.Thermal Load on Line Elements

ם]

Lets you apply temperature loads to the selected line elements (truss, beam, and/or rib). You must specify values for the following parameters:

Tension/Compression	n on Trusses and Beams	×
Define	C Modify	
P [kN] = 0	▼ →← +P	
Pick Up >>	OK Cancel	

Thermal Load On Beam 1	×
Define C N	lodify
Temperature Variati © In plane x-z © In plane x-y T1 [*C] = -25,00 T2 [C] = 40,00	$x_{y} T_{1}$ $T_{2} y$ $T_{1} T_{2}$ $T_{2} y$
Pick Up >>	OK Cancel

TrussTref -reference temperature (corresponding to the initial unstressed state)T-the temperature assumed for the analysis

dT = T - T_{ref} is the temperature variation that is taken into account in the analysis. A positive dT means a warm up of the truss.

Beam/Rib $T_{ref:}$ - reference temperature (corresponding to the initial unstressed state) $T_1:$ - the temperature of the top cord (in the corresponding local direction) $T_2:$ - the temperature of the bottom cord (in the corresponding local direction)

 $dT^{=}=T - T_{ref}$ is the uniform temperature variation that is taken into account in the analysis, where *T* is the temperature of the cross-section in its center of gravity.

in local y direction: $T = T_2 + (T_1 - T_2) \frac{y_G}{H_y}$

in local z direction: $T = T_2 + (T_1 - T_2) \frac{z_G}{H_z}$

where,

 y_G , z_G , and H_y , H_z are properties of the cross-section.

A positive dT^{-} indicates a temperature increase of the beam.

 $dT^{\#}=T_1 - T_2$ is the non-uniform temperature variation that is taken into account in the analysis.

4.10.17. Thermal Load on Surface Elements

F

Lets you apply temperature loads to the selected surface elements. You must specify values for the following parameters:

Thermal Load on S	urface Elements	×
Define	C Modify	
$T_1 [^{\circ}C] = 50,0$ $T_2 [^{\circ}C] = -10,0$		
	T _{reference} [°C] = 20,00	
Pick Up >>	OK Cancel	

Tref: - reference temperature (corresponding to the initial unstressed state)

 $T_{1:}$ - the temperature of the top cord (in the positive local z direction)

T_{2:} - the temperature of the bottom cord (in the negative local z direction)

 $dT^{=}=T$ - T_{ref} is the uniform temperature variation that is taken into account in the analysis, where T is the temperature in the center of gravity of the cross-section.

 $dT^{\#}=T_1 - T_2$ is the non-uniform temperature variation that is taken into account in the analysis.



For membranes only dT⁼ *is taken into account. For plates only* dT[#] *is taken into account.*

upport Displace

Define

4.10.18. Forced Support Displacement

Lets you apply forced displacements to the selected support elements. You must specify the values of the forced displacement components (translational: e [m]; rotational: θ [rad]).

AxisVM approximates the problem, by applying a force $P_{support}$ in the direction of the support element so as to produce the forced displacement *e*.

 $P_{\sup port} = K_{\sup port} \cdot e$

 $e_{x} [mm] = 0 \qquad \forall \\ e_{y} [mm] = 0 \qquad \forall \\ e_{y} [mm] = 0 \qquad \forall \\ e_{z} [mm] = -50 \qquad \forall \\ \varphi_{x} [rad] = 1 \qquad \forall \\ \varphi_{y} [rad] = 0 \qquad \forall \\ \varphi_{z} [rad] = 0 \qquad \forall \\$

nts At Support 2

C Modify

where $K_{support}$ is the corresponding support stiffness.

If the stiffness of the support element is large enough, the secondary deflections due to other loads will be negligible. Therefore, you may apply forced displacements only to the supports stiff enough relative to the stiffness of the structure (at least 10^3 times larger) in the corresponding direction.

Check this assumption every time, by checking the displacement results and verifying the displacement at the respective node.

A positive forced displacement moves the node in the positive direction of the local axis.



4.10.19. Influence Line

<u>+†-</u>

Lets you apply a relative displacement load to obtain the influence line of an internal force component, on the selected truss/beam elements.

You must specify the value of the relative displacement e as +1 or -1.

My Influence Line Load on I	Beam 2 🔀
Position C By Length C By Ratio a = 0.5 ♀ Length = 2,500 m	y 2 dyy
Relative displacement	
C e _x = 0	Ο θ _χ = 0
C e _y = 0	$\Theta_y = 1$
• e ₂ = 0	Ο θ _z = 0
	OK Cancel

[©] You can define influence line load, only in an influence line type load case.

See... 4.10.1 Load Cases, Load Groups

Truss You can specify the value of the relative displacement e_x as +1 or -1.



Beam You can specify the value of the relative displacement e_x / e_y / e_z / θ_x / θ_y / θ_z as +1 or –1.



4.10.20. Seismic Loads

տիտ

The seismic loads are taken into account according to the Response Spectrum Analysis method. This method requires a previously calculated number of undamped free vibration frequencies and the corresponding mode shapes.



Based on these vibration mode shapes AxisVM generates equivalent static loads (for each vibration mode shape) which are then applied to the model in a static analysis. Then internal force results obtained for each mode shape are summed using to the method described in design code specifications.

Seismic analysis can be performed based on the following design codes:

Design codes

- Eurocode 8 (EN 1998-1:2004)
- Swiss code (SIA 261:2003)
- German code (DIN 4149:2005-04)
- Italian code (OPCM 3274)

The program performs only the analysis described below. Any other supplementary analysis required by the design codes must be completed by the user. AxisVM can calculate extra torsional moments due to random eccentricities of masses and check the sensitivity of stories to second order effects.

Check the table of seismic equivalence coefficients in X, Y, Z directions in the Table

These are the steps of creating seismic loads and setting response spectrum parameters:

Browser. Vibration results will appear only if you the Vibration tab is selected.

Seismic load generation, setting parameters

Results Vibration (first-order)	l 😔 🕂	× P		4 8		
	Seismic Equ	iivalence	Coefficien	ts (I.) [ST1]		
Mode 1 (1.34 HZ)		f [H ₇]	5.	F.,	5-	Ac
- Mode 3 (2.01 Hz)		. [9%	Sy.	*2	-10
- Mode 4 (4 10 Hz)	1	1.34	0.837	0.003	0	
- Mode 5 (4 98 Hz)	2	1.63	0.010	0.591	0	•
Mode 6 (6.24 Hz)	3	2.01	0.003	0.242	0	•
-Mode 7 (6.94 Hz)	4	4.10	0.100	0	0	
Mode 8 (9.24 Hz)	5	4.98	0	0.064	0	•
- Mode 9 (9.58 Hz)	6	6.24	0.005	0.039	0	•
Mode 10 (10.47 Hz)	7	6.94	0.023	0.007	0	
- Mode 11 (10.86 Hz)	8	9.24	0.008	0	0	
- Mode 12 (12.61 Hz)	9	9.58	0	0.029	0	
All Mode Shapes (12)	10	10.47	0.001	0.003	0.001	
-Frequencies (12)	11	10.86	0	0	0.518	
Seismic Equivalence Coeffic	12	12.61	0	0.003	0.003	
🖃 LIBRARIES 📃 👻	1 12/12		0.989	0.982	0.522	

1. Calculate the first n vibration mode shapes and frequencies.

[©] Each design code requires that the mode shapes must represent a certain ratio of the total mass. E.g. In Eurocode 8 the requirement is $\varepsilon \ge 0.9$ (the sum of the coefficients must represent at least 90% in each direction) and every mode shape having a coefficient larger that 5% in any direction must be included.

The individual mode shapes can be turned on or off. Mode shapes turned off are not used when calculating seismic loads.

After right-clicking any cell of the the *Active* column the popup menu appears. Choosing *Turn on/off mode shapes* displays a dialog.

Mode shapes under user-defined threshold values can be turned off. The program can be set to reapply this filtering based on ε_X , ε_Y and ε_Z automatically after every vibration analysis.

Turn on/off mode shapes
C _urn on all mode shapes C _urn off all mode shapes ⊙ _urn off mode shapes under threshold values
$\epsilon_{\chi} \le 0.003$ $\epsilon_{\gamma} \le 0.003$ $\epsilon_{Z} \le 0.003$
Reapply after every vibration analysis
OK Cancel

2. Create a new seismic load case.

The program will create multiple load cases:

⊟ Ungrouped	Load Case New Case
Permanent Distributed	+++ 🗠 🦢 🛲 🚧
Self Weight Snow Hi Snow Wind	Self Weight Dyplic contains 60 loads.
WoodWind	Load Group Permanent
- 通 MOV1-001 - 通 MOV1-002 - 通 MOV1-003	Load Group (Eurocode)
	New Group
一週 MOV1-005	

a.) Without extra torsional effects:

Load cases with endings X, Y and Z. The result of these cases will contain the maximum displacements and forces summed up from seismic effects in X, Y or Z direction.

Load cases with endings + and –. The results of these cases will contain the positive and negative maximum displacements and forces summed up from seismic effects in X, Y and Z direction.

b.) With extra torsional effects:

Load case with endings Xa, Xb, Ya, Yb. The results of these load cases will contain the maximum forces and displacements calculated from the seismic effect in X or Y direction and the torsional effect with a + eccentricity (Xa and Ya) or with a – eccentricity (Xb and Yb).

Load case with ending Z. The results of this load case will contain the maximum forces and displacements calculated from the seismic effect in Z direction.

Load cases with endings 1+ and 1-. The results of these load cases will contain the maximum forces and displacements calculated from the sum of Xa, Ya and Z with a + or - sign.

Load cases with endings 2+ and 2-. The results of these load cases will contain the maximum forces and displacements calculated from the sum of Xa, Yb and Z with a + or - sign.

Load cases with endings 3+ and 3-. The results of these load cases will contain the maximum forces and displacements calculated from the sum of Xb, Ya and Z with a + or - sign.

Load cases with endings 4+ and 4-. The results of these load cases will contain the maximum forces and displacements calculated from the sum of Xb, Yb and Z with a + or - sign.

Select any of these cases.

The effect of seismic forces in Z direction will be taken into account only if a vertical response spectrum is defined.



3. Setting seismic parameters

Clicking this button you can set the response spectrum and other parameters.

- T.		<u>A</u> nalysis	Case	
<u></u>		Linear	T ST1	¥
Parameters (Eur	ocode)			
		γ ₁ = 1		
Spectrum (horiz	ontal) Spectre	um (vertical) Torsiona	al effect Combinat	ion methods
		<u>D</u> esign spe	ctrum	
	$a_{aR} [m/s^2] = 1$,000 <parametr< td=""><td>ic shape></td><td>-</td></parametr<>	ic shape>	-
Ground type	a = [2	A	0 1 1 1 2 1	
A Type 1		<u> </u>	Sd [m/s≁]	
B Type 1	S = 1	1,250		
C Type 1	т (е) - Го	1150		
D Type 1	, B [9] - Lo	1,150		
A Time 2	T _C [s] = 0	,400 0.007		
B Type 2	E	U,00/ \		0.000
C Type 2	1 _D [s] = 2	2,000		0,200
D Type 2	β = 0),2 LLL		TIel
E Type 2				1[5]
I		0		4,000

The parameters required depend on the actual design code (see details below).

Closing this dialog futher load cases will be created:

Load cases with endings 01X, 02X,nX, 01Y, 02Y,nY, 01Z, 02Z,nZ.

These are the seismic forces in X, Y or Z direction coming from individual mode shapes. Load cases with endings 01tX, 02tX,ntX, 01tY, 02tY,ntY.

These are the extra torsional forces due to the seismic effects in X or Y direction.

4.10.20.1. Seismic calculation based on Eurocode 8

Eurocode 8 Design response spectrum (EN 1998-1:2004) S (T) (for linear analysis)

 $\mathbf{S}_{d}(T)$ (for linear analysis)

The program uses two different spectra for the horizontal and vertical seismic effects. You can create a spectrum in two ways

- 1. Define a custom spectrum.
- 2. Define a parametrical spectrum based on Eurocode 8 EC8 EN1998-1 (4.2.4.)

Parametrical design response spectrum for horizontal seismic effects:



$$0 \le \mathbf{T} < \mathbf{T}_{\mathrm{B}}: \qquad S_d(T) = a_g \cdot S \cdot \left[\frac{2}{3} + \frac{T}{T_B} \left(\frac{2.5}{q} - \frac{2}{3}\right)\right]$$

$$T_B \leq T < T_C$$
: $S_d(T) = a_g \cdot S \cdot \frac{2.5}{q}$

$$\mathbf{T}_{\mathrm{C}} \leq \mathbf{T} < \mathbf{T}_{\mathrm{D}}: \quad S_{d}(T) = a_{g} \cdot S \cdot \frac{2.5}{q} \left[\frac{T_{\mathrm{C}}}{T} \right] \geq \beta \cdot a_{g}$$

$$T_D \leq T$$
: $S_d(T) = a_g \cdot S \cdot \frac{2.5}{q} \left[\frac{T_C \cdot T_D}{T^2} \right] \geq \beta \cdot a_g$

where **S**, T_B , T_C , T_D , is defined in EC8 EN 1998-1(Table 3.2,3.3.)

The default values of these parameters depend on the soil class and the type of spectrum.

Type 1 spectra									
Subsoil	S	T _B	T _C	T _D					
class		[s]	[s]	[s]					
А	1,0	0,15	0,4	2,0					
В	1,2	0,15	0,5	2,0					
С	1,15	0,20	0,6	2,0					
D	1,35	0,20	0,8	2,0					
Е	1,40	0,15	0,5	2,0					
	Type 2 spectra								
Subsoil	S	T _B	T _C	T _D					
class		[s]	[s]	[s]					
А	1,0	0,05	0,25	1,2					
В	1,35	0,05	0,25	1,2					
C	1,50	0,10	0,25	1,2					
D	1,80	0,10	0,30	1,2					
Е	1.60	0.05	0.25	1.2					

The above parameters can be changed when defining the parametric spectrum.

 \mathbf{a}_{g} : design ground acceleration

- **β**: lower limit for the horizontal design spectrum (the recommended value is 0.2).
- **q** : behaviour factor for horizontal seismic effects. It depends on the type and material of the structure. This factor connects the linear analysis results and the nonlinear (elastic-plastic) behaviour of the structure.

Parametrical design response spectrum for vertical seismic effects: EC8 EN 1998-1 (3.2.2.5.)

Vertical design parametric spectrum is calculated from the horizontal spectrum, but \mathbf{a}_{g} and \mathbf{q} is replaced by \mathbf{a}_{gv} and \mathbf{q}_{v} ,

default values of **S**, **T**_B, **T**_C, **T**_D are:

Type 1				
a_{vg}/a_{g}	S	TB	T _C	T _D
		[s]	[s]	[s]
0,90	1,0	0,05	0,15	1,0
Туре 2				
a_{vg}/a_{g}	S	T _B	T _C	T _D
		[s]	[s]	[s]
0,45	1,0	0,05	0,15	1,0

 \mathbf{a}_{gv} : vertical design ground acceleration

 $\mathbf{q}_{\mathbf{v}}$: behaviour factor for vertical seismic effects

Torsional effects (optional) EC8 EN 1998-1 (4.3.3.3.3.)

AxisVM calculates extra torsional forces around a vertical axis due to random eccentricities of masses for every story and modal shape using the maximum X and Y sizes of stories:



Extra torsional moments due to seismic effects in X or Y direction are

$$\begin{split} M_{tXi} &= F_{Xi} \cdot (\pm 0.05 \cdot H_{Yi}) \\ M_{tYi} &= F_{Yi} \cdot (\pm 0.05 \cdot H_{Xi}) \end{split}$$

where

 F_{Xi} and F_{Yi} are the horizontal forces belonging to a modal shape of the *i*th story due to seismic effects in X or Y direction. Torsional moments will be taken into account with both (+ and –) signs but always with the same sign on all stories.

Seismic forces are

1

$$P_{kr} = S_D(T_r) \cdot m_k \cdot \eta_{kr}$$

where

 η_{kr} is the mode shape ordinate reduced according to its seismic coefficient

- index of degree of freedom **k**:
- index of modal shape r:

Analysis

Seismic effects are analysed in global X and Y direction (horizontal) and optionally in global Z direction (vertical).

Seismic effects in X and Y direction are considered to be coexistent and statistically independent effects.

æ

Combination of modal responses in one direction, EC8 EN 1998-1-2 (3.3.3.2.)

Force and displacement maximum values can be calculated according to two different methods:

SRSS method (Square Root of Sum of Squares):

$$E = \sqrt{\sum_{i} E_i^2}$$

CQC method (Complete Quadratic Combination):

 $E = \sqrt{\sum_{i} \sum_{j} E_{i} \cdot r_{ij} \cdot E_{j}}$

where *E* is a displacement or force component value at a certain point.

Combination of spatial components

Resultant maximum displacement and force values can be calculated from the coexisting effects in X, Y and Z direction according to two different methods:

1. Quadratic mean:

$$E = \sqrt{E_X^2 + E_Y^2 + E_Z^2}$$

2. Combination with 30%:

$$E = \max \begin{pmatrix} E_X"+"0.3E_Y"+"0.3E_Z\\ 0.3E_X"+"E_Y"+"0.3E_Z\\ 0.3E_X"+"0.3E_Y"+"E_Z \end{pmatrix}$$

where

 $\mathbf{E}_{Xr} \mathbf{E}_{Yr} \mathbf{E}_{Z}$ are the maximum values of independent seismic effects in X, Y, and Z direction.

Calculating displacements

Displacements coming from nonlinear behaviour are calculated this way:

 $E_s = q_d \cdot E$

where

 q_d : behaviour factor for the displacements

E : maximum displacement form the linear analysis

Usually $q_d = q$.

Check of second order seismic sensitivity EC8 EN 1998-1 (4.4.2.2.)

At the end of a seismic analysis AxisVM checks the second order seismic sensitivity of each story. The sensitivity factor θ is calculated from the seismic effects in X or Y direction:

$$\theta = \frac{P_{tot} \cdot d_r}{V_{tot} \cdot h} , \text{ where}$$

 \mathbf{P}_{tot} is the total gravitational load above and on the story

- $\mathbf{d}_{\mathbf{r}}$ is the interstory displacement calculated from the differences of average displacements between stories with a seismic effect in X or Y direction.
- \mathbf{V}_{tot} is the total seismic shear force above and on the story coming from a seismic effect in X or Y direction.



h is the interstory height

The program finds the section of walls at the story level then determines the shear center (*S*) using the calculation method for cross-sections. It converts the loads of the load case used for vibration analysis to masses, then finds their center of gravity for each story (G_m). Calculates the total mass of stories (*M*) and the inertia at the center of gravity about an axis in Z direction (I_{mz}). Results can be find in the table of *Seismic sensitivity of stories*. This table appears among the results only if the *Static* tab is selected.



Seismic parameters, response spectra and combination methods can be set in a dialog.

Spectral functionSetting the Design spectrum typeeditorcombo from Parametric to Customand clicking on the SpectralFunction Editor icon a dialogappears.Spectrum can becreated/modified as a functionconsisting of linear segments.Segment points listed on the lefthand side can be edited.



On the third tab page you can choose the combination methods.

Combination methods	Spectrum (horizontal) Spec	ctrum (vertical) Torsional effect Combination methods
	Combination of modal responses	• Auto • $E = \sqrt{\sum_i E_i^2}$ (SRSS) • $E = \sqrt{\sum_{i,j} E_i r_{ij} E_j}$ (CQC) $\xi' = 0.05$
	Combination of the components of seismic action	• $E_{\max} = \sqrt{E_X^2 + E_Y^2 + E_Z^2}$ • $E_{\max} = \max \begin{pmatrix} E_X + 0.3E_Y + 0.3E_Z \\ 0.3E_X + E_Y + 0.3E_Z \\ 0.3E_X + 0.3E_Y + E_Z \end{pmatrix}$

Combination of modal responses

It is possible to let the program choose the combination method of modal responses by turning on the *Automatic* radio button. If $T_j/T_i < 0.9$ is true for all vibration mode shapes (i.e. the modal responses can be considered to be independent) then the program choose SRSS method. In other cases the CQC method will be chosen.

Combinations of the components of seismic action The quadratic formula or the 30%-method can be chosen.

4.10.20.2. Seismic calculation based on Swiss Code

Swiss code (SIA 261:2003)

Design response spectrum

 $S_d(T)$ for linear analysis

AxisVM uses two spectra for the analysis: one for horizontal seismic effects and one for vertical ones.

A design response spectrum can be defined as a user-defined diagram or in a parametric form based on SIA 261:2003 (16.2.4.)





$$0 \le T < T_B: \qquad S_d(T) = \gamma_f \cdot a_{gd} \cdot S \cdot \left[0.67 + \left(\frac{2.5}{q} - 0.67\right) \frac{T}{T_B} \right]$$

 $T_B \leq T < T_C$: $S_d(T) = 2.5 \cdot \gamma_f \cdot a_{gd} \cdot \frac{S_d}{q}$

$$T_{C} \leq T < T_{D}$$
: $S_{d}(T) = 2.5 \cdot \gamma_{f} \cdot a_{gd} \cdot S \cdot \left[\frac{T_{C}}{T \cdot q}\right]$

$$\mathbf{T}_{\mathrm{D}} \leq \mathbf{T}: \qquad \qquad S_d(T) = 2.5 \cdot \gamma_f \cdot a_{gd} \cdot S \cdot \left[\frac{T_C \cdot T_D}{T^2 \cdot q}\right] \geq 0.1 \cdot \gamma_f \cdot a_{gd}$$

where

 \mathbf{a}_{gd} : horizontal design ground acceleration

- γ_f : importance factor of the building
- **q** : behaviour factor for horizontal seismic effects which depends on the type and material of the structure. **q** is the link between the linear calculation and the nonlinear (elastic-plastic) behaviour of the structure.

S, T_B , T_C , T_D : the default values of these parameters depend on the soil class based on SIA 261:2003 (Table 25)

Design response spectrum				
Subsoil	S	T _B	T _C	T _D
class		[s]	[s]	[s]
А	1,0	0,15	0,4	2,0
В	1,2	0,15	0,5	2,0
С	1,15	0,20	0,6	2,0
D	1,35	0,20	0,8	2,0
E	1,40	0,15	0,5	2,0

The design spectrum is not normalized with g.

Parametric design response spectrum for vertical seismic effects: SIA 261:2003 (16.2.4.)

The vertical parametric design response spectrum is based on the horizontal one. a_{gd} and q must be replaced by a_{gdv} and $q_{v\prime}$

where

 \mathbf{a}_{gdv} : vertical design ground acceleration, ($\mathbf{a}_{gdv} = 0.7 \mathbf{a}_{gd}$)

 $\mathbf{q}_{\mathbf{v}}$: behaviour factor for vertical seismic effects

Torsional effects (optional) SIA 261:2003 (16.5.3.4.)

AxisVM calculates extra torsional forces around a vertical axis due to random eccentricities of masses for every story and modal shape using the maximum X and Y sizes of stories:



Extra torsional moments due to seismic effects in X or Y direction are

$$\begin{split} M_{tXi} &= F_{Xi} \cdot (\pm 0.05 \cdot H_{Yi}) \\ M_{tYi} &= F_{Yi} \cdot (\pm 0.05 \cdot H_{Xi}) \end{split}$$

where

 F_{Xi} and F_{Yi} are the horizontal forces belonging to a modal shape of the *i*th story due to seismic effects in X or Y direction. Torsional moments will be taken into account with both (+ and –) signs but always with the same sign on all stories.

Seismic forces are

$$P_{kr} = S_D(T_r) \cdot m_k \cdot \eta_{kr}$$

where

- η_{kr} is the mode shape ordinate reduced according to its seismic coefficient
- k: index of degree of freedom
- r: index of modal shape

Analysis

Seismic effects are analysed in global X and Y direction (horizontal) and optionally in global Z direction (vertical).

Seismic effects in X and Y direction are considered to be coexistent and statistically independent effects.

Combination of modal responses in one direction

Force and displacement maximum values can be calculated according to two different methods:

SRSS method (Square Root of Sum of Squares):

 $E = \sqrt{\sum_{i} E_i^2}$

CQC method (Complete Quadratic Combination):

$$E = \sqrt{\sum_{i} \sum_{j} E_{i} \cdot r_{ij} \cdot E_{j}}$$

where *E* is a displacement or force component value at a certain point.

Combination of spatial components

Resultant maximum displacement and force values can be calculated from the coexisting effects in X, Y and Z direction according to two different methods:

1. Quadratic mean:

$$E = \sqrt{E_X^2 + E_Y^2 + E_Z^2}$$

2. Combination with 30%:

$$E = \max \begin{pmatrix} E_X "+"0.3E_Y "+"0.3E_Z \\ 0.3E_X "+"E_Y "+"0.3E_Z \\ 0.3E_X "+"0.3E_Y "+"E_Z \end{pmatrix}$$

where

 $E_{X\!\prime}\;E_{Y\!\prime}\;E_{Z}$ are the maximum values of independent seismic effects in X, Y, and Z direction.

Calculating displacements

Displacements coming from nonlinear behaviour are calculated this way:

 $E_s = q_d \cdot E$

where

 q_d : behaviour factor for the displacements

E : maximum displacement form the linear analysis

^G Usually $q_d = q$.

Check of second order seismic sensitivity EC8 EN 1998-1 (4.4.2.2.)

At the end of a seismic analysis AxisVM checks the second order seismic sensitivity of each story. The sensitivity factor θ is calculated from the seismic effects in X or Y direction:

$$\theta = \frac{P_{tot} \cdot d_r}{V_{tot} \cdot h}$$
 , where

 $\boldsymbol{P}_{tot}~$ is the total gravitational load above and on the story

- dr is the interstory displacement calculated from the differences of average displacements between stories with a seismic effect in X or Y direction.
- \mathbf{V}_{tot} is the total seismic shear force above and on the story coming from a seismic effect in X or Y direction.
- **h** is the interstory height





Seismic parameters, response spectra and combination methods can be set in a dialog.

Spectral function editor Setting the Design spectrum type combo from *Parametric* to *Custom* and clicking on the Spectral Function Editor icon a dialog appears. Spectrum can be created/modified as a function consisting of linear segments. Segment points listed on the left hand side can be edited.



Combination methods

Spectrum (horizontal)	Spectrum (vertical) Torsional effect Combination methods
Combination of modal responses	• Auto • $E = \sqrt{\sum_i E_i^2}$ (SRSS) • $E = \sqrt{\sum_{i,j} E_i \tau_{ij} E_j}$ (CQC) $\xi' = 0.05$
Combination of the components of seismic action	$ \mathbf{\bullet} E_{\max} = \sqrt{E_X^2 + E_Y^2 + E_Z^2} $ $ \mathbf{\bullet} E_{\max} = \max \left\{ \begin{aligned} E_X + 0.3E_Y + 0.3E_Z \\ 0.3E_X + E_Y + 0.3E_Z \\ 0.3E_X + 0.3E_Y + E_Z \end{aligned} \right\} $

Combination of modal responses

It is possible to let the program choose the combination method of modal responses by turning on the *Automatic* radio button. If $T_j / T_i < 0.9$ is true for all vibration mode shapes (i.e. the modal responses can be considered to be independent) then the program choose SRSS method. In other cases the CQC method will be chosen.

Combinations of the components of seismic action

The quadratic formula or the 30%-method can be chosen.

2005-04

4.10.20.3. Seismic calculation based on German Code

DIN 4149: Design response spectrum

 $S_d(T)$ (for linear analysis)

The program uses two different spectra for the horizontal and vertical seismic effects. You can create a spectrum in two ways

- 1. Define a custom spectrum.
- 2. Define a parametrical spectrum based on DIN 4149:2005-04 (5.4.3)

Parametrical design response spectrum for horizontal seismic effects:



$0 \leq T < T_B$:	$S_d(T) = a_g \cdot \gamma_I \cdot S \cdot \left[1 + \frac{T}{T_B} \left(\frac{\beta_0}{q} - 1 \right) \right]$
	ße

 $T_B \leq T < T_C:$ $S_d(T) = a_g \cdot \gamma_I \cdot S \cdot \frac{\rho_0}{q}$

$$T_{C} \le T < T_{D}$$
: $S_{d}(T) = a_{g} \cdot \gamma_{I} \cdot S \cdot \frac{\beta_{0}}{q} \frac{T_{C}}{T}$

$$T_D \leq T$$
: $S_d(T) = a_g \cdot \gamma_I \cdot S \cdot \frac{\beta_0}{q} \cdot \frac{T_C \cdot T_D}{T^2}$

where **S**, T_B , T_C , T_D , is defined in DIN 4149:2005-04 (Table 4) The default values of these parameters depend on the soil class.

Resp	onse spec	ctrum		
Soil Class	S	TB	T _C	T_D
5011 C1035		[s]	[s]	[s]
A-R	1,0	0,05	0,2	2,0
B-R	1,25	0,05	0,25	2,0
C-R	1,5	0,05	0,3	2,0
B-T	1,0	0,1	0,3	2,0
C-T	1,25	0,1	0,4	2,0
C-S	0,75	0,1	0,5	2,0

The above parameters can be changed when defining the parametric spectrum.

a_g : Ground acceleration

 γ_{I} : Importance factor for buildings DIN 4149:2005-04 (Table 3)

- β_0 : Spectral acceleration factor (Refrence value $\beta_0 = 2,5$)
- **q** : Behaviour factor for horizontal seismic effects. It depends on the type and material of the structure. This factor connects the linear analysis results and the nonlinear (elastic-plastic) behaviour of the structure.

Parametrical design response spectrum for vertical seismic effects: DIN 4149:2005-04 (Table 5)

Vertical design parametric spectrum is calculated from the horizontal spectrum, but a_g and q is replaced by a_{gv} and q_{v} ,

default values of S, T_B, T_C, T_D are:

Response spectrum				
Soil Class	S	T _B [s]	T _C [s]	Τ _D [s]
A-R	1,0	0,05	0,2	2,0
B-R	1,25	0,05	0,2	2,0
C-R	1,5	0,05	0,2	2,0
B-T	1,0	0,1	0,2	2,0
C-T	1,25	0,1	0,2	2,0
C-S	0,75	0,1	0,2	2,0

 \mathbf{a}_{gv} : vertical design ground acceleration ($\mathbf{a}_{gv} = 0.7 \mathbf{a}_{g}$) \mathbf{q}_{v} : behaviour factor for vertical seismic effects

Torsional effects (optional) DIN 4149:2005-04 (6.2.2.4.3)

AxisVM calculates extra torsional forces around a vertical axis due to random eccentricities of masses for every story and modal shape using the maximum X and Y sizes of stories:



Extra torsional moments due to seismic effects in X or Y direction are

$$\begin{split} M_{tXi} &= F_{Xi} \cdot (\pm 0.05 \cdot H_{Yi}) \\ M_{tYi} &= F_{Yi} \cdot (\pm 0.05 \cdot H_{Xi}) \end{split}$$

where

FXi and FYi are the horizontal forces belonging to a modal shape of the ith story due to seismic effects in X or Y direction. Torsional moments will be taken into account with both (+ and –) signs but always with the same sign on all stories.

Seismic forces are

$$P_{kr} = S_D(T_r) \cdot m_k \cdot \eta_{kr}$$

where

 ηkr is the mode shape ordinate reduced according to its seismic coefficient

- k: index of degree of freedom
- r: index of modal shape

Analysis

Seismic effects are analysed in global X and Y direction (horizontal) and optionally in global Z direction (vertical).

Combination of modal responses in one direction

Force and displacement maximum values can be calculated according to two different methods:

SRSS method (Square Root of Sum of Squares):

$$E = \sqrt{\sum_{i} E_{i}^{2}}$$

CQC method (Complete Quadratic Combination):

$$E = \sqrt{\sum_{i} \sum_{j} E_{i} \cdot r_{ij} \cdot E_{j}}$$

where *E* is a displacement or force component value at a certain point.

 $[\]mathcal{F}$ Seismic effects in X and Y direction are considered to be coexistent and statistically independent effects.

Combination of spatial components

Resultant maximum displacement and force values can be calculated from the coexisting effects in X, Y and Z direction according to two different methods:

1. Quadratic mean:

 $E = \sqrt{E_X^2 + E_Y^2 + E_Z^2}$

2. Combination with 30%:

 $E = \max \begin{pmatrix} E_X "+"0.3E_Y "+"0.3E_Z \\ 0.3E_X "+"E_Y "+"0.3E_Z \\ 0.3E_X "+"0.3E_Y "+"E_Z \end{pmatrix}$

where

EX, EY, EZ are the maximum values of independent seismic effects in X, Y, and Z direction.

Calculating displacements

Displacements coming from nonlinear behaviour are calculated this way: $E_s = q_d \cdot E$ where qd: behaviour factor for the displacements

E : maximum displacement form the linear analysis

[©] Usually qd=q.

Check of second order seismic sensitivity DIN 4149:2005-04 (7.2.2.(2))

At the end of a seismic analysis AxisVM checks the second order seismic sensitivity of each story. The sensitivity factor [] is calculated from the seismic effects in X or Y direction:



Seismic parameters, response spectra and combination methods can be set in a dialog.



Spectral functionSetting the Design spectrum typeeditorcombo from Parametric to Customand clicking on the SpectralFunction Editor icon a dialogappears. Spectrum can be created /modified as a function consisting oflinear segments. Segment pointslisted on the left hand side can beedited.



On the third tab page you can choose the combination methods.

Combination of modal responses	• Auto • $E = \sqrt{\sum_{i} E_{i}^{2}}$ (SRSS) • $E = \sqrt{\sum_{i} E_{i}^{2}}$ (COC) 81 = 0.05
Combination of the components of seismic action	

Combination of modal responses

It is possible to let the program choose the combination method of modal responses by turning on the Automatic radio button. If Tj / Ti < 0.9 is true for all vibration mode shapes (i.e. the modal responses can be considered to be independent) then the program choose SRSS method. In other cases the CQC method will be chosen.

Combinations of the components of seismic action The quadratic formula or the 30%-method can be chosen.

4.10.20.4. Seismic calculation based on Italian Code

Italian code Design response spectrum

 $S_d(T)$ for linear analysis

AxisVM uses two spectra for the analysis: one for horizontal seismic effects and one for vertical ones.

A design response spectrum can be defined as a user-defined diagram or in a parametric form based on the Italian code.

Parametric design response spectrum for horizontal seismic effects:



Combination methods

$$\begin{split} 0 &\leq \mathrm{T} < \mathrm{TB}: \qquad S_d(T) = a_g \cdot S \cdot \left[1 + \frac{T}{T_B} \left(\frac{2,5}{q} - 1 \right) \right] \\ \mathrm{T}_\mathrm{B} &\leq \mathrm{T} < \mathrm{T}_\mathrm{C}: \qquad S_d(T) = a_g \cdot S \cdot \frac{2,5}{q} \\ \mathrm{T}_\mathrm{C} &\leq \mathrm{T} < \mathrm{T}_\mathrm{D}: \qquad S_d(T) = a_g \cdot S \cdot \frac{2,5}{q} \left[\frac{T_\mathrm{C}}{T} \right] \geq 0.20 \cdot a_g \\ \mathrm{T}_\mathrm{D} &\leq \mathrm{T}: \qquad S_d(T) = a_g \cdot S \cdot \frac{2,5}{q} \left[\frac{T_\mathrm{C} \cdot T_\mathrm{D}}{T^2} \right] \geq 0.20 \cdot a_g \,, \end{split}$$

Where

the default values of S, T_B , T_C , T_D depend on the subsoil class.

Subsoil	S	TB	TC	TD
class		[s]	[s]	[s]
Α	1,0	0,15	0,40	2,0
B, C, E	1,25	0,15	0,50	2,0
D	1,35	0,20	0,80	2,0

 a_g : design ground acceleration

q : behaviour factor for horizontal seismic effects. It depends on the type and material of the structure. This factor connects the linear analysis results and the nonlinear (elastic-plastic) behaviour of the structure.

Parametrical design response spectrum for vertical seismic effects:

$0 \le T < T_B$:	$S_{vd}(T) = 0.9 \cdot \alpha \cdot S \cdot \left[1 + \frac{T}{T_B} \left(\frac{3.0}{q_v} - 1 \right) \right]$
$T_B \leq T < T_C$:	$S_{vd}(T) = 0.9 \cdot \alpha \cdot S \cdot \frac{3.0}{q_v}$
$T_C \le T < T_D$:	$S_{vd}(T) = 0.9 \cdot \alpha \cdot S \cdot \frac{3.0}{q_v} \left[\frac{T_C}{T} \right]$
$T_D \leq T$:	$S_{vd}(T) = 0.9 \cdot \alpha \cdot S \cdot \frac{3.0}{q_v} \left[\frac{T_C \cdot T_D}{T^2} \right]$
$a_{gv} = 0,9 \cdot a_g$	

If no detailed results are available $q_v = 1,5$ for all type of structure and all materials.

Seismic forces are

$$P_{kr} = S_D(T_r) \cdot m_k \cdot \eta_{kr}$$

where

 η_{kr} is the mode shape ordinate reduced according to its seismic coefficient

Analysis

Seismic effects are analysed in global X and Y direction (horizontal) and optionally in global Z direction (vertical).

P

Seismic effects in X and Y direction are considered to be coexistent and statistically independent effects.

Combination of modal responses in one direction

Force and displacement maximum values can be calculated according to two different methods:

SRSS method (Square Root of Sum of Squares): CQC method (Complete Quadratic Combination):

$$E = \sqrt{\sum_{i} E_{i}^{2}}$$

 $E = \sqrt{\sum_{i} \sum_{j} E_{i} \cdot r_{ij} \cdot E_{j}}$

where *E* is a displacement or force component value at a certain point.

Combination of spatial components

Resultant maximum displacement and force values can be calculated from the coexisting effects in X, Y and Z direction according to two different methods:

1. Quadratic mean:

$$E = \sqrt{E_X^2 + E_Y^2 + E_Z^2}$$

2. Combination with 30%:

$$E = \max \begin{pmatrix} E_X "+"0.3E_Y "+"0.3E_Z \\ 0.3E_X "+"E_Y "+"0.3E_Z \\ 0.3E_X "+"0.3E_Y "+"E_Z \end{pmatrix}$$

where

 $E_{X\prime}$ $E_{Y\prime}$ E_{Z} are the maximum values of independent seismic effects in X, Y, and Z direction.

Displacements coming from nonlinear behaviour are calculated this way:

 $E_s = q_d \cdot E$, where

 q_d : behaviour factor for the displacements

E : maximum displacement form the linear analysis



Seismic parameters, response spectra and combination methods can be set in a dialog.
User's Manual

Spectral function editor

Setting the Design spectrum type combo from *Parametric* to *Custom* and clicking on the Spectral Function Editor icon a dialog appears. Spectrum can be created/modified as a function consisting of linear segments. Segment points listed on the left hand side can be edited.



Combination methods

Spectrum (horizontal)Spectrum (vertical)Torsional effectCombination methodsCombination of modal
responsesImage: Auto
Image: Combination of the
components of seismic
actionImage: Combination of the
components of seismic
actionCombination of the
components of seismic
actionImage: Combination of the
Combination of the
components of seismic
actionImage: Combination of the
Combination of the
components of seismic
actionImage: Combination of the
components of seismic
actionImage: Combination of the
Combination of the
Combination of the
Combination of the
Combination of the
components of seismic
actionImage: Combination of the
Combination of the
C

Combination of modal responses

It is possible to let the program choose the combination method of modal responses by turning on the *Automatic* radio button. If $T_j / T_i < 0.9$ is true for all vibration mode shapes (i.e. the modal responses can be considered to be independent) then the program choose SRSS method. In other cases the CQC method will be chosen.

Combinations of the components of seismic action

The quadratic formula or the 30%-method can be chosen.

4.10.21. Pushover loads

Pushover loads are generated according to the regulations of Eurocode 8 (EN 1998-1:2004) by default. The load generation uses undamped free vibration frequencies and corresponding mode shapes of the model, therefore loads can only be generated if a vibration analysis has already been performed.

Pushover load generation steps

The following description shows how to create pushover load cases and set their properties before performing a nonlinear static analysis.

1. Calculate vibration mode shapes and frequencies.

When running the vibration analysis be sure to use the convert loads to masses option with the appropriate load case if there are loads defined that need to be considered static. Check the table of seismic equivalence coefficients in the Table Browser. Vibration results will appear only if the Vibration tab is selected.



Although there is no requirement in Eurocode 8 for the minimum value of seismic equivalence coefficient, it is strongly advised to perform standard pushover analysis only on structures having clearly dominant mode shapes in each horizontal direction. The coefficients for each mode shape are listed in the Seismic Equivalence Coefficients table (see Figure above). Unlike Seismic loads, standard pushover load generation uses a single vibration mode shape for each load case, therefore the sum of seismic equivalence coefficients is not important. Thus there is no need to calculate a large number of modes if the dominant ones are among the first few.

Pushover load cases can be created, renamed and deleted in the Load Cases Load Groups dialog window. The initial configuration of four load cases is created by clicking on the Pushover Load button.

Ungrouped	Load Case New Case					
	444 :	<u>1-</u>	亚	\checkmark		follor
TPUShover X U TPUShover X M 亚 Pushover Y U 亚 Pushover Y M	Pushover contains 144 I	XM oads.			1	Dyplicat
	Load Group]			
	Load Group (E	urocode)				
	New Group					
	New Group	Ð Ð	•	₽		1
	New Group	Ŧ	Ð	₽]
	New Group	Ð.	Ð	Ð]
	New Group	J J		¢]
	New Group	J	•	æ]
	New Group	¥ ¥		Ð]
	New Group	¥ ¥		Ð]

3. Setting pushover load parameters.

爭

After creating the load cases the parameters for the loads can be set up by clicking on the Pushover Analysis button in the toolbar of the Loads tab.

X direction		V dire	ction	
Analysis 2	nd order 🛛 💌	1	Analysis	2nd order
Load case	. Tk 💌]	Load case	1. Tk
🔽 Uniform load distri	oution	🔽 Unifi	orm load dis	tribution
🔽 Modal load distribu	tion	Mod	al load distr	ibution
Mode 🔽	Dominant		Mode	🔽 Dominant
N	12 (0,950) 🛛 🔻	1		M1 (0,980)
Stor	les			~
	Stories	Z[m]	Imz =	U
				Add
				Delete
			-	
				Pick Up 🔊
				1 10 10 10 10 10 10 10 10 10 10 10 10 10

The parameters for load generation can be set up at the top, while the story levels used for interstory drift calculation are specified at the bottom part of the window. (Previously defined story data is also available here)

Load generation for a specific direction can be disabled using the topmost checkboxes. This is useful in case the model is two dimensional. For each direction the vibration analysis type and the assigned load case needs to be selected first. The checkboxes below turn the uniform and modal load generation on or off respectively. The uniform load distribution option generates nodal forces proportional to the masses assigned to each node in the model. The modal load distribution uses the mode shape weighed by the masses at each node to generate the nodal force distribution. In both cases the sum of forces generated is 1kN in the same horizontal direction.

If modal loads are to be generated it is possible to override the dominant mode shape used for load generation. It is important to emphasize that this option is only for advanced users and Eurocode 8 requires the use of dominant mode shape for analysis. The number in brackets by each mode number shows the corresponding seismic equivalence coefficient. Pushover loads are generated only after closing the dialog window. Unnecessary load cases are also removed at this time.

4. Run a Nonlinear Static Analysis

After defining loads for pushover load cases the pushover analysis shall be run using the Nonlinear Static Analysis button under the Static tab of the main window. Setting the solution control to Pushover lets the user define a parametric and a constant load case. The parametric load case is typically a pushover load case, however AxisVM does allow users to define other load cases as parametric too. The constant load case represents gravitational loads in most cases. The other settings of this dialog window are explained in Static Analysis chapter.

The control node shall be one of the nodes at the top of the structure. It is important to set the direction of the analysis according to the direction of the parametric load case. The stability of the analysis can be increased significantly by increasing the number of increments. Following geometric nonlinearity is recommended for pushover analyses. The analysis is started by clicking the OK button.

Generation of capacity curves and related results are explained in 6.1.4 Pushover capacity curves chapter.

4.10.22. Tensioning

些

Tendons can be assigned to a continuous selection of beam or rib elements. After defining tendon properties and the tensioning process AxisVM determines the immediate losses of prestress and the equivalent loads for the end of tensioning (load case *name-T0*). After completing a static analysis it determines the time dependent losses of prestress and the long term equivalent loads from the result of quasi-permanent combinations (load case *name-TI*). Tendon trajectory tables can be generated with user-defined steps.

Tendons

The first tab is to define tendon parameters and geometry.



Icons on the vertical toolbar beside the tendon list are

Add new tendon. Geometry for the new tendon can be defined using the toolbar beside the diagram.

 $\langle \langle |$

Geometrical tansformations of tendons



Tendons selected in the tree can be translated or mirrored. Tendons can be copied or just moved. Copied tendons inherit the original parameters and the tensioning process assigned to them.

Delete tendon. Deletes the selected tendon.

Parameters of the selected tendon appear beside the tendon list. Parameter values can be edited.

- E_p modulus of elasticity of tendon steel
- A_p cross-section area of the tendon
- f_{pk} characteristic tensile strength of tendon steel
- μ coefficient of friction between the tendon and its sleeve
- *k* unintentional angular displacement for internal tendons per unit length. Shows the precision of workmanship.
 - Ususally 0,005 < k < 0,01.
- R_{min} Minimum radius of curvature. Where the radius of curvature is smaller than this limit tendons are displayed in red.

To draw tendon geometry click the icons on the vertical toolbar beside the drawing and enter base points. AxisVM determines the trajectory passing through these base points as a cubic spline to minimize curvature. For each basepoint the angles of tangent can be specified by setting the α (top view) and β (side view) values in the table. Enter values between -180° and 180°. Initial values are 0°. Existing base points can be dragged to a new position using the mouse.

Draw tendon in 2D. Base points can be created by clicking the diagram or using the coordinate window. Double-click or *Mouse Right Button/Complete* to make the base point the last one. The tendon position within the cross-section has to be specified only at the first base point. Further base points will be in the local *x*-*z* plane containing the first base point. *Steps of drawing a tendon in 2D:*

1. Select the postion of the cross-section where you want to define the tendom basepoint.

Settle the tendon onto the proper position in the cross-section view.

You can position the tendon onto the top or at the bottom of the cross-section considering the concrete cover.

Position the tendon onto an optional point



.

Position the tendon onto the neutral axis

- Position the tendon onto the top of the cross-section
- Position the tendon onto the bottom of the cross-section
- 2. Following the first location you can position the other points of the tendon onto the longitudinal section.

Draw tendon in 3D. The tendon position within the cross-section has to be specified at every basepoint. You can close a tendon geometry with using Mouse Right Button/Complete. *Steps of drawing a tendon in 3D:*

- 1. Select the postion of the cross-section where you want to define the tendom basepoint.
- 2. Settle the tendon onto the proper position in the cross-section view.

Following the first location repeat the step 1. and step 2. to define all basepoint.

Add new base point. Click the cable to add a new base point. In case of several tendons this function only works with the active tendon.

Delete base point. Clicking an existing base point deletes it. After deleting the second base point the tendon geometry is deleted. In case of several tendons this function only works with the active tendon.

Table of baseBase point properties can be edited in the table. Use the toolbar beside the table to add basepointspoints or remove the selected lines.

	Base points - T1							
+		× [m]	y [m]	z [m]	Tangent	a [°]	β ["]	
$\underline{\mathbf{N}}$	1	0	-2,250	0,100	~	0	357,25	
	2	25,000	-2,250	-0,500				
	3	50,000	-2,250	0,100	~	0	2,75	

Options. Grid and cursor settings of the longitudinal and the cross-section diagram can be set. **See...** 2.15.15.1 Grid and Cursor

Tensioning process

The second tab is to define the tensioning process for tendons by determining the order of certain operations.



Possible operations and parameters:



Tensioning from left / right / both side Release from left / right / both side

Anchor on left / right / both side

Deletes the last operation from the list.

Force as a fraction of the characteristic value of tendon steel tensile strength (f_{pk}).

Wedge draw-in of the anchorage device

Concrete

The third tab is to check the material properties of the concrete. $e_{cs}(\infty)$ is the long term value of the concrete shrinkage strain. Its value can be entered here.



Results If valid parameters, geometry and tensioning process is assigned to every tendon, result diagrams are displayed on the fourth tab. If one tendon is selected in the tree two diagrams are shown. The first one is the actual tension along the tendon (f_p / f_{pk}), and the equivalent load for the tendon (F). If more than one tendon is selected the diagram shows the resultant equivalent load for the selected tendons only.



Immediate losses of tension

1. Tension loss due to friction between tendons and their sleeves at position x measured from the anchorage point along the tendon is calculated as

$$\sigma_{\mu}(x) = \sigma_{\max}(1 - e^{-\mu(\Theta + kx)})$$

where

 σ_{max} is the maximum tension in the tendon

 Θ is the sum of the absolute angular displacements over a distance *x*

2. Losses due to the instantaneous deformation of concrete are calculated as

$$\Delta P_{el} = A_p E_p \sum \left[\frac{j \Delta \sigma_c}{E_{cm}} \right],$$

where

 $\Delta \sigma_{\rm c}$ is the variation of stress at the centre of gravity of the cross-section *j* = (n-1)/2n, where *n* is the number of stressing steps *E*_{cm} is the secant modulus of elasticity of concrete

3. Losses at anchorage are due to wedge draw-in of the anchorage devices.

Long term loss of tension

Long term loss of force due to shrinkage and creep of the concrete and the relaxation of the tendon is calculated as

$$\Delta P_{c+s+r} = A_p \Delta \sigma_{c+s+r} = A_p \frac{\varepsilon_{cs} E_p + 0.8\Delta \sigma_{pr} + \frac{E_p}{E_{cm}} \varphi \sigma_{c,QP}}{1 + \frac{E_p}{E_{cm}} A_c} (1 + \frac{A_c}{I_c} z_{cp}^2) [1 + 0.8\varphi]},$$

where

 $\begin{array}{ll} \Delta \sigma_{c+s+r} & \text{is the tension loss due to the effects above} \\ E_{cm} & \text{is the secant modulus of elasticity of concrete} \\ \Delta \sigma_{pr} & \text{is the long term absolute tension loss due to the relaxation of tendons} \\ & \text{in case of 2nd relaxation class :} \end{array}$

$$\Delta \sigma_{pr} = \sigma_{\max} \cdot 0.66 \,\rho_{1000} \,e^{9.1\mu} \cdot 500^{0.75(1-\mu)} \cdot 10^{-5} \,,$$

in case of 3rd relaxation class :

$$\Delta \sigma_{pr} = \sigma_{\text{max}} \cdot 1,98 \rho_{1000} e^{8\mu} \cdot 500^{0.75(1-\mu)} \cdot 10^{-5}$$

where $\rho_{1000} = 2,5\%$ is the relaxation loss at a mean temperature of 20°C at 1000 hours after tensioning

φ	final value of creep coefficient
$\sigma_{c,OP}$	is the stress in the concrete adjacent to the tendons, due to self-weight and ini-
~~~	tial prestress and other quasi-permanent actions where relevant.
$A_p$	is the total cross-section area of tendons
A _c	is the cross-section area of the concrete
Ic	is the second moment area of the concrete section
$Z_{cp}$	is the distance between the centre of gravity of the concrete section and the
,	tendons

# Trajectory table The last tab is to build a trajectory table for the selected tendons with the desired increment and optional shift of origin. The trajectory table consists of the local y and z coordinates of the selected tendons at the calculated x positions.

The defined basepoints are always displayed in the Trajectory Table.

Tension	ning between No	de 1 and 4				<u>_0×</u>
Elle Edit	Window					
						<u>C</u> lose Cancel
Tendons	Tensioning proce	ss Con	crete Re	sults T	rajectory table	· · · · · · · · · · · · · · · · · · ·
Traject	tory table			Р	roperties	
⊡ ⊒ Al	l tendons				[1] T1	
	2 [1] T1				ength [m] 50.019	
	[2] T1/1				[kN/cm ² ] 19500	
	[3] T1/3				[cm ² ] 30.00	
	[4] T1/4			t t	. [kN/cm ² 1 186.00	
	[5] 11/5 161 T4 /6				0.200	
	171 T1 7			k	0.005	
	[1] 11/8 [18] T1/8					
	[9] T1/9			Ir	jected yes	
	[10] T1/10					
	[11] T1/11					
	[12] T1/12			-		
				L		
		Coordin	ates			
Increment	[m] = 1.000		[1] T1	[1] T1		<u> </u>
		x[ni]	y[m]	z[m]		
		11,000	-2,250	-0,312		
Shift of or	rigin	12,000	-2,250	-0,338		
		13,000	-2,250	-0,362		
dX	[m] = 0	14,000	-2,250	-0,384		
dY	[m] = 0	15,000	-2,250	-0,404		
		16,000	-2,250	-0,422	_	
dZ	[m] = 0	17,000	-2,250	-0,439		
		18,000	-2,250	-0,453	_	
		19,000	-2,250	-0,465		
		20,000	-2,250	-0,476		
		21,000	-2,250	-0,485		
		22,000	-2,250	-0,491		
		23,000	-2,250	-0,496		-
		24,000	-2,250	-0,499		<u> </u>

## Main toolbar

The main toolbar has two buttons.

Copy diagram Ctrl+C Copies the drawing on the active tab to the Clipboard as a Windows metafile. This way the diagram can be pasted to other applications (e.g. Word).



Prints a report of the tensioning using diagrams and tables. Tendons and report items can be selected. You can choose the position of the drawing (landscape or portrait) and set the scale of it (*Print options for drawings*).

Print		×
Print           Tendons         Cross-sections           □ □ □ 11/1         □ 11/1           □ □ 11/3         □ 11/3           □ □ 11/4         □ 11/5           □ 11/6         □ 11/6	Tendons Tendon loss Equivalent load Tgtal equivalent load	Report     Tendons - [1] T1     Total equivalent load - 1 tendons     Base points - [1] T1     Tendon parameters - 1 tendons
- 11/7 - 11/8 - 11/8 - 11/9 - 11/10 - 11/11 - 11/12 - 11/13	Tase points     Iensioning process     Tendon parameters     Concrete Properties     Trajectory table	
Select tendons and related data to print	Print options for drawings	
		OK Cancel

Cross-sections can be selected to print cross-section diagrams.

rint								×
Tendons	Cross-sections				Rep	ort		
	t basepoints to prin I tendons √ 11 √ 0 m √ 25,000 m 111 0 m 50,000 m 113 10 m 50,000 m 114	cross-section diagram	s ↓ + poptions for dra A ⊂ V	<b>Wings</b> M = 1 : 100		Tendons - [1] T1 Totel equivalent load - 1 Base points - [1] T1 Tendon parameters - 1	tendons	
				]	1		ок	Cancel

Menu

You can reach the following functions via the menu:

Eile		
6	Print	Ctrl+P

Print See Main toolbar / Print

```
Edit <u>Eile Edit Window</u>
```

File

Ē	<u>C</u> opy diagram	Ctrl+C			
$\square$	Geometrical transformations on tendons				
	Join connecting tendons				

*Undo/Redo* Undoes the effect of the previous command./ Executes the command which was undone.

Copy diagram See... Main toolbar / Copy diagram

See... Tendons / Geometrical transformations of tendons

Geometrical transformations of tendons Join connecting tendons

*ting* If more than one beam or rib element has been selected and these elements contain connecting tendons this function joins the connecting tendons. The joining works in case of single element, too.

Window	<u>F</u> ile	<u>E</u> dit	<u>W</u> in	dow
			¥.	Coordinates
			•	<u>S</u> tatus

- *Coordinates* Editing of the longitudinal and cross-section diagrams is made easier by a coordinate window. The display of this window can be turned on and off.
  - *Status* On diagrams an information window appears displaying diagram-specific information. The display of this window can be turned on and off.

## 4.10.23. Moving loads



Moving loads allow modeling of a drifting load with a constant intensity like a vehicle crossing a bridge or a crane carriage moving along its runway.

To define a moving load a moving load case must exist. It can be created on the Loads tab clicking the *Load cases and load groups* icon. **See...** 4.10.1 Load Cases, Load Groups. Moving load icons will be enabled only if the current load case is a moving load case. After defining the load new load cases will be created automatically according to the number of steps specified. Auto-created load cases cannot be deleted or moved into another load group individually. Increasing the number of load steps will create additional load cases. Decreasing this number will make certain load cases useless. These excess load cases will be removed only before saving the model.

Moving load symbols can be displayed in two ways. The first option is to draw the current phase only. The second one is to draw other phases in gray.

## 4.10.23.1. Moving loads on line elements



Moving load on line elements is a load pattern moving on a user-defined load path in *N* steps.

The load pattern can contain any combination of concentrated and distributed loads. Individual loads in the pattern can be local or global and their position and intensity components can be set. This way the vertical load of a crane carriage and the horizontal forces can be applied together on the runway.

Loads can be added to the pattern by clicking the plus icon and filling out the fields in the row. Selected rows can be deleted by clicking the *Delete* icon under the plus icon. Load patterns can be saved under a name and reloaded.

After load pattern definition it is necessary to select the load path. It must be a continuous sequence of beams or ribs. After selecting the elements constituting the load path the startpoint and endpoint has to be selected. These points must be nodes along the path. Beside the load path button the value of *N* can be set. It determines the number of steps the

Beside the load path button the value of *N* can be set. It determines the number of steps the load pattern will make evenly along the path.

The local z direction of the load pattern will always be the local z direction of the line elements it is placed on.

Lengthening, shortening or breaking a line element of the path will lead to an automatic recalculation of the load phases.

In the first phase the load with the lowest coordinate in the pattern will be placed over the



*Crane runway mode* startpoint. In the last phase the load with the highest coordinate in the pattern will be placed over the endpoint.

**P** → F Bridge mode In the first phase the load with the highest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the lowest coordinate in the pattern will be placed over the endpoint.



One way: Load moves from startpoint to endpoint in N steps.

Round trip: Load moves from startpoint to endpoint and back in 2N steps.

4.10.23.2. Moving loads on domains



This load type is convenient when vehicle loads has to be defined. The load pattern consists of concentrated or rectangular surface loads pairs representing the wheels on the axles.

*u* is the vehicle gauge, *a* and *b* refers to the rectangle dimensions. Axle load *F* will be distributed evenly on the two wheels . Load patterns can be saved under a name and reloaded.

The load type and direction switches on the left determines the properties of all loads entered into the table.

Loads can be added to the pattern by clicking the plus icon and filling out the fields in the row. Selected rows can be deleted by clicking the *Delete* icon under the plus icon.

After load pattern definition it is necessary to select the load path. It must be a continuous polyline running through domains.

The load path does not have to stay in the same plane and can cross holes or empty areas between domains.

Path startpoint and endpoint is the first and last point of the polyline.

Each phase will contain only the loads actually falling on a domain. The local z direction of the load pattern will be the local z direction of the domain it is placed on. In case of a path running along the edge of two or more domains in different planes only the domains in the active parts are taken into account. The local z direction will be chosen finding the domain with the minimum angle between local z and global Z directions.

Beside the load path button the value of *N* can be set. It determines the number of steps the load pattern will make evenly along the path.

In the first phase the load with the lowest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the highest coordinate in the pattern will be placed over the endpoint.

Changing domain geometry will lead to an automatic recalculation of the load phases.



In the first phase the load with the lowest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the highest coordinate in the pattern will be placed over the endpoint.

mode

In the first phase the load with the highest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the lowest coordinate in the pattern will be placed over the endpoint.



*One way:* Load moves from startpoint to endpoint in *N* steps.

*Round trip:* Load moves from startpoint to endpoint and back in 2N steps.

## 4.10.24. Dynamic loads (for time-history analysis)

Dynamic nodal loads and acceleration functions can be defined for time-history analysis. Acceleration functions can be used for seismic analysis. In this case it is recommended to obtain proper seismic accelerograms and assign these functions to support nodes to analyse the effects of the earthquake. This method provides more exact results than the response spectrum analysis and can be used even if nonlinear elements are defined in the model (nonlinear supports, tension-only trusses, etc.). Its disadvantage is that it cannot be combined with other load types automatically.

*To define nodal loads or acceleration functions the current load case must be a dynamic load case.* **See...**4.10.1. Load Cases, Load Groups



Dynamic loads and accelerations are defined by functions which describe the parameter in time. Function editor is available from the dynamic load definition dialogs.

Functions must be entered as value pairs in a table. *Plus sign icon* adds a new row, *Delete icon* deletes selected rows. Functions are plotted automatically and can be printed. Functions can be reused. In order to make them available later, save them into the function library. Saved functions can be reloaded, edited and saved under a new name. Functions are saved into separate *.*dfn* files in a *dfn* folder created under the main folder of the program.



## **Table editing functions**

Adds a new row to the table.

 $\mathbf{\times}$  Deletes selected rows from the table.

 $\square$  Copies the selected cells to the Clipboard.

Insert the content of the Clipboard into the table.

*f(t)* Formula editing.

The f(t) load function can be entered as a formula. The follwing operators and functions are available: +, -, *, /, (, ), sin, cos, tan, exp, ln, log10, log2, sinh, cosh, tanh, arcsin, arccos, arctan, arcsinh, arccosh, arctanh, int, round, frac, sqr, sqrt, abs, sgn, random. random(t) returns a random number between 0 and 1.

A machine rotating about the Y axis has a dynamic load function with the following X and Z components:

 $fx(t) = a^* \cos(\omega t + \varphi)$  and  $fz(t) = a^* \sin(\omega t + \varphi)$ 

As functions are represented as a series of values a  $\Delta t$  step and a  $T_{max}$  total time must be specified.

## **Diagram and report functions**

Prints the diagram and the table.



Copies the diagram and the table to the Clipboard.

Starts the Report Maker.



Saves the diagram into the Gallery. See... 2.10.4 Gallery

Mexico 1985 EVV

A function previously saved to the library can be loaded by selecting its name from the dropdown list.



•

Renames the current function.



Saves the current function to the library.



Loads a function from the library.





The first point of functions must be at t=0. This value pair cannot be changed or deleted. If the load is applied only at T > 0, the function value must be zero between 0 and T.

DynamicTo define dynamic nodal loads selectnodal loadnodes and set the parameters in adialog.For each component you can assign an<br/>intensity and a dynamic load function<br/>describing the time-dependence of the

load factor. To use an existing function from the library click the first icon beside the combo. To edit the load function click the second icon.

The load directions can be the global X, Y and Z directions or the direction can be determined by a chosen reference. In this latter case there is just one force and moment component.

Dynamic nodal loads	x
	fy
Direction G Global C Referential Reference	$\begin{array}{c} & M_{2}^{2} \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & & \\ & &$
	Dynamic load functions
F _X [kN] = 0 ▼	<static></static>
F _Y [kN] = 0	<static> 🖬 🚔 📐</static>
F _Z [kN] = -1 ▼	DynLoadZ 💽 🚅 📐
M _X [kNm] = 0	<static> 💽 🚅 📐</static>
M _Y [kNm] = 0	<static> 💽 🚔 📐</static>
M _Z [kNm] = 0	<static> 🖬 🖆 📐</static>
Pick Up >>	OK Cancel

It is possible to define a constant (time-independent) load by selecting <Static> from the *Dynamic load functions* combo.

- The actual value of a load component in t will be calculated as  $F_i(t) = F_i \cdot f(t)$ , i.e. the load intensity is multiplied by a time-dependent load factor.
- *If a dynamic load is defined for a support with an existing dynamic load the existing load will be overwritten.*

Modify, delete

Dynamic loads can be modified or deleted the same way as static loads.

Get Dynamic loads are displayed as dashed yellow arrows.

Dynamic support acceleration

÷....+

Acceleration function can be assigned to any nodal support in the model. For each component you can assign an acceleration intensity and a dynamic load function describing the time-dependence of the load factor. The actual value of the acceleration at t will

The actual value of the acceleration at t will be calculated as

$$a_i(t) = a_i \cdot f(t) ,$$

i.e. the acceleration is multiplied by a timedependent load factor.

Dinamikus csomóponti támaszgy	orsulás 🔀
<u>D</u> efiniálás     C <u>M</u> ódos	itás
Irány       © Globális       © Referencia irányú       Referencia	Z az av
$a_{\chi} [m/s^2] = 0$ $a_{\gamma} [m/s^2] = 0$ $a_{Z} [m/s^2] = 0$	Dinamikus teherfüggvények
Eelvesz »	OK Mégsem

- *Acceleration acts at the bottom of the support string. The acceleration of the supported node can be different depending on the support stiffness.*
- *If acceleration is defined for a support with an existing acceleration load the existing load will be overwritten.*

Modify, delete

If multiple nodal supports are attached to a node, acceleration acts on all supports.

Dynamic support acceleration can be modified or deleted the same way as a static load.

Ger Dynamic support acceleration is displayed as a circle and a yellow arrow.

Dynamic nodal acceleration

Nodal acceleration can be assigned to any node in the model. For each component you can assign an acceleration intensity and a dynamic load function describing the timedependence of the load factor. The actual value of the acceleration in *t* will be calculated as  $a_i(t) = a_i \cdot f(t)$ , i.e. the acceleration is multiplied by a time-dependent load factor.

)ynamic nodal acceleration	x
● Define C Modify	
Direction	z az dar az ay ay ay
$a_{\chi} [m/s^2] = 9.810$ $a_{\gamma} [m/s^2] = 9.810$ $a_{Z} [m/s^2] = 0$	DynFunction1
Pick Up >>	OK Cancel

*If acceleration is defined for a support with an existing acceleration load the existing load is overwritten.* 

*To specify ground acceleration for seismic analysis nodal support accelerations must be defined.* 

Modify, delete

Dynamic nodal acceleration can be modified or deleted the same way as a static load.

↔ Dynamic nodal acceleration is displayed as a circle and a yellow arrow.

۳ ا

## 4.10.25. Nodal Mass

In a vibration analysis the masses are concentrated at nodes that you can take into account by their global components Mx, My, Mz. In second-order vibration analysis, the loads due to the nodal masses are applied on the model, as well as the masses due to the applied loads. If mass is the same in each direction it is enough to specify one value after checking *Apply the same mass in each direction.*.

Nodal Mass		×
Define	$\mathbf{C}$ Modify	
• Overwrite	C <u>A</u> dd	
Apply the same n	nass in each directio	n
m _X [kg] = 0	-	
m _Y [kg] = 0	-	
m _Z [kg] = 0	-	
Pick Up >>	OK	Cancel

In dynamic analysis nodal masses and nodal accelerations result in dynamic loads causing displacements and forces in the model.

Ger The nodal mass is displayed on the screen as two dark red concentric circles.

## 4.10.26. Modify

*Modify* To modify loads:

- 1. Press the **[Shift]** key and select loads you want to modify (or the loaded elements). You can also select by drawing a selection frame or using the Selection Toolbar.
- 2. Click the load type icon on the Toolbar.
- 3. Check the checkboxes beside the values you want to change.
- 4. Enter new values.
- 5. Close the dialog with **OK**.
- *Immediate mode* If the Loads tab is active click a finite element to modify its loads. If the element has more than one load only one of them will come up. If you have placed different concentrated and distributive loads on a beam and click the beam the load nearest to the click position will come up. If more finite elements have been selected their loads can immediately be modified by clicking one of them. If you click an element which is not selected, selection disappears and you can modify the element load you clicked.
  - ^(*) In fact, load modification is similar to the load definition, but does not assign loads to elements not being loaded and allows access to a specific load property without altering others. You can switch to the Define radio button to place loads on all the selected elements, lines or surfaces. If we select elements with loads not matching the load type we choose these loads remain unchanged.

## 4.10.27. Delete

[Del] See... 3.2.7 Delete

## 4.11. Mesh



Clicking the mesh tab mesh toolbar becomes available with mesh generation for line elements and domains, mesh refinement functions and a finite element shape checking.

## 4.11.1. Mesh Generation

Automatic detection of overlapping lines and missing intersections during meshing reduces the errors in model geometry.

Support of multiple core processors can reduce the time of meshing.

## 4.11.1.1. Meshing of line elements

Mesh parameters for line elements Finite element analysis uses linear elements with constant cross-section so arced and variable cross-section (tapered) line elements must be divided into parts. This is called line element meshing. The accuracy of the solution depens on the mesh density.

This mesh can be removed or modified just like a domain mesh. Removing a mesh does not delete loads and properties assigned to the line element.

A mesh can also be defined for linear elements with constant cross-section. It is useful in nonlinear or vibration analysis when it is required to divide line elements to achieve a higher accuracy.

Mesh parameter for line elements	×
Meshing criterion	
C Maximum Deviation From Arc	d [m] =
C Maximum element size	d [m] = 0,500 👻
Division into N segments	N = 4 💌
C By angle	d ["] =
	OK Cancel

Mesh generation can be performed according to different crteria:

#### Maximum deviation from arc

Chord height cannot exceed the value specified.

## Maximum element size

Length of the mesh lines cannot exceed the value specified.

Division into N segments

Line elements are divided into N parts.

By angle

Central angle of arced mesh segments cannot exceed the value specified.

#### 4.11.1.2. Mesh generation on domain

A mesh of triangular surface elements can be generated on the selected domains by specifying an average surface element side length for the mesh. Meshing will take into account all the holes, internal lines and points of the domain. Meshes can also follow loads above a certain intensity.

Meshing paramaters	Meshing Parameters	×
for domains		
	Mesh size Average Mesh Element Size [m] = 0,500	
	Fit mesh to loads	
	Image: Point loads   ≥ [kN] =         0	
	I Line loads   ≥ [kN/m] =	
	✓ Surface loads   ≥ [kN/m ² ] = 0	
	Contour division method	
	Uniform mesh size	
	○ Adaptive mesh size	
	Create mesh only for unmeshed domains	
	Calculation of domain intersections	
	OK Cancel	

Mesh size An average mesh element size can be specified. The actual mesh can contain smaller and larger elements as well.

Fit mesh to loads Meshes will follow checked loads if load intensity exceeds the value specified. Point loads will create mesh nodes, line loads will create mesh lines.

#### Contour division Uniform mesh size

method

Domain boundaries and inner lines will be divided according to the mesh size to ensure the given element size.

Adaptive mesh size

Adaptive meshing follows domain geometry and refine the mesh by reducing element size wherever it is necessary.

If Create mesh only for unmeshed domains is checked no mesh will be created for domains already meshed.

If Calculation of domain intersections is turned on domain intersections are automatically calculated before meshing.

The progress of the mesh generation process can be monitored in a window, and can be canceled any time with the Abort button.

The mesh generator uses only the end-points of beam elements that are in the plane of the domain, and disregards their corresponding line segments. Rib elements are incorporated with their line segments because they can be defined on surface edges as well.

If there are existing quadrilateral or triangular meshes within the domain, the mesh generator will not change these meshes, and will integrate them in the new mesh.



Before Meshing

After Meshing

If a mesh is generated over an existing domain mesh (with a different average element side length), the new mesh will replace the existing one.

## 4.11.2. Mesh Refinement



Lets you refine the finite element mesh of surfaces. The elements in the refined mesh have the same properties (material, cross-section / thickness, references, etc.) as those in the coarse mesh.

^{CF} You have to manually set the nodal degrees of freedom of the newly generated mesh that were not set automatically during the process of mesh generation.

#### The following options are available:



Generate Uniform Mesh	×
Maximum element side length [m] =	0,500
ОК	Cancel

Lets you refine the entire selected mesh. You must specify the maximum side length of a surface element in the refined mesh.





After mesh refinement

Bisection







Quadrilateral element







Triangular element

Lets you refine the mesh around the selected nodes (locally around columns, nodal supports). You must specify a division ratio (0.2-0.8). The command refines the mesh dividing the elements connected to the respective nodes by the defined ratio.

1							
			ſ				
1		•		•	•	•	
1					 	 	 
ĺ			[			-	
ĺ							

Before mesh refinement



After mesh refinement

## User's Manual

Edge relative

▦

æ,



Lets you refine the mesh along the selected edges (locally along edge supports / loads). You must specify a division ratio (0.2-0.8). The command refines the mesh dividing the elements connected to the respective edges by the defined ratio.

## 4.11.3. Checking finite elements

Program checks the minimum angle of surface finite elements ( $\alpha$ ).

A triangular finite element is distorted if  $\alpha \le 15$ . A qudrilateral finite element is distorted if  $\alpha \le 30$ . This page is intentionally left blank.

## 5. Analysis

AxisVM lets you perform linear and nonlinear static, linear and nonlinear dynamic, vibration and buckling analysis. It implements an object-oriented architecture for the Finite Element Method.

The instructions included in this User's Manual assume a preliminary knowledge of the finite element method and experience in modeling. Note that the finite element analysis is only a tool, not a replacement for engineering judgment.

🗊 Linear analysis of Shopping Mall.axs			×
Spring element strain evaluation			
Save		Cancel	
16:28:18 Beam element strain evaluation			•
16:28:18 Beam element stress evaluation 16:28:18 Beam element equilibrated load evaluation			
16:28:18 Beam element equilibrated load assembly			
16:28:18 Spring element strain evaluation			Ţ
10.20.10 Opting demon of an orthograph		_	
Number of Equations: 6504		Truss	240
Equations Memory: 4.7 M	- M	Rih	/55
Estimated Memory Requirement: 34.8 M		Spring	_
Solver block size 1 M		Gap	_
Largest available memory block: 116 M		Link	_
Analysis block size 5 M		Edge hinge	_
Available physical memory: 1.081 G		Membrane	-
Total physical memory: 1.999 G		Plate	-
CPU: Intel(R) Pentium(R) 4 CPU 3.00GHz (2x)		Shell	-
Multiple cores / threads 3010 MHz		Diaphragm	-
Model optimization	00:00	Load case	13
Model Verification	00:09		
Analysis	00:05		
Result File Generation			

Each analysis consists of three steps:

- 1) Model optimization
- 2) Model verification
- 3) Performing the analysis
- 4) Result file generation

The actual running times of each step, and details of the model can be displayed by pressing the Details button.



Parameters of the latest analysis is saved into the model file and can be studied in the Model Info dialog. **See...** 2.15.16 Model Info.

Model optimization	Optimization reduces the band width of the system stiffness matrix by iterative node re- numbering. Changes in the memory requirement is displayed real-time. The duration of the optimization process and the final memory requirement depends on the size of the sys- tem and the available memory.
F	The system of equations can be solved the most efficiently if the whole system fits into the physical memory. If the system does not fit into the physical memory but its largest block does, the running time will be moderate. If the largest block does not fit into the physical memory, the necessary disk operations can slow down the solution considerably.
Model verification	The input data is verified in the first step. If an Error is found a warning message is displayed and you can then decide whether to cancel or continue the analysis
Performing the analysis	AxisVM displays the evolution of the solution process by two progress bars. The bar on the top displays the current step performed, while the other displays the overall progress of the analysis process.
	The equilibrium equations in the direction of constrained degrees of freedom are not included in the system of equations. Therefore to obtain support reactions you must model the support conditions using support elements.
	The Cholesky method is applied to the solution of linear equilibrium equations. The eigenvalue problems are solved with the Subspace Iteration method.
Error of the solution	Solution error is calculated from the solution of a load case with a known result. It is a good estimation of the order of errors in displacement results for other load cases. Info palette shows this error as E(EQ).
	If the value of E (Eq) is greater than 1E-06 the reliability of the computed results is questionable. It is expected, that the Error of the displacements is of the same order.
Result file generation	During the processing of the results the program sorts the results according to the original order of the nodes and prepares them to graphical display.
	In the following chapters we 'll show the setting of the parameters of the each calculation methods.

## 5.1. Static Analysis

The term *static* means that the load does not vary or the variation with the time can be safely ignored.

- *Linear static* Performs a linear static analysis. The term *linear* means that the computed response (displacement, internal force) is linearly related to the applied load.
  - All the load cases are solved in the analysis. Through the geometric linearity, it is assumed that the displacements remain within the limits of the small displacement theory. Through the material linearity, it is assumed that all materials and stiffness characteristics are linear-elastic. The materials assigned to surface elements can be othotropic.
  - See the description of the gap, and spring elements in Chapter 4, on how to use these elements in a linear analysis.

## Nonlinear static

۳<u>ل</u>

Performs a nonlinear-elastic static analysis. The term *nonlinear* means that the computed response (displacement, internal force) is nonlinearly related to the applied load. This can be due to the use of gap, link or non-linear support, truss or spring elements, or taking into account the geometric nonlinearity of truss, beam, rib and shell elements.

Select load cases or combinations in the tree view. AxisVM will perform nonlinear analysis for the selected load cases and shows a progress dialog.

🚡 Nonlinear Static Analysis	_ 🗆 🗙
Load cases	
Parametric load case	ST1
Constant load case	JS11
Solution Control	
C Eorce	Node » 1
C <u>D</u> isplacement	Direction: X
Pushover	Maximal Displacement: [mm]
C Equal increments	Load Factor
Number of increments:	10 1,0000
Increment function	
<equal increments=""></equal>	
,	0 10
Convergence Criteria	
Maximum Iterations:	20
I Displacement	0,001
Force	0,001
Vork	1E-6
Continue even if no converg	ence has been achieved
Use reinforcement in calcu	Ilation
C Actual Reinforcement	C Calculated Reinforcement
J✓ Follow geometric nonlinearity (	of beams, trusses, ribs and shells
Store last increment only	
	OK Cancel

Load case	Status	Started at	Verification	Analysis	Processing	Total time	Finished a
Self Weight	Finished	17:09:52	0	00:10	0	00:11	17:10:05
Surface Load	Finished	17:10:05	0	00:10	0	00:11	17:10:18
Axial 1	Finished	17:10:18	0	00:10	0	00:11	17:10:24
Axial 2	Finished	17:10:24	0	00:10	0	00:11	17:10:38
Axial 3	42%	17:10:38	-	00:04			

## **Solution control**

#### Force

When the Force control is selected, the increments are applied as equal fractions of the loads (as one parameter load).

## Displacement

When displacement control is selected, the increments are applied as equal fractions of the displacement component of the node specified.

## Pushover

Pushover control is a special type of displacement control that allows the use of a constant load case while having another parametric load case that is increased incrementally. This is essential for pushover analyses to model  $P-\Delta$  effects appropriately.

After selecting pushover control, the top of the dialog changes to accommodate the drop-down boxes for parametric and constant load cases. **See...** 4.10.21 Pushover loads for details on load definition and recommended analysis settings.

#### Load factor

Load factor can be used to multiply loads of the selected load case or combination for the nonlinear analysis.

## Number of increments

There are two methods to define the number of increments:

- 1. **Equal increments.** Specify the number of increments. The default value is 10. When highly nonlinear behavior is analyzed, you may specify a greater value in order to achieve convergence.
- 2. **Increment function.** Loads are not increasing in a linear way but follow a predefined function. Using an increment function it is possible to reduce the number of increments where the behaviour of the structure is linear and increase the number of increments where the behaviour is nonlinear.

*Increment function must be monotonous (loads cannot decrease).* 

## **Convergence** criteria

Based on the convergence tolerances you specify, AxisVM will determine if the nonlinear solution has reached the required accuracy (convergence). Therefore it is important that the convergence tolerances to be set properly. During the iteration process, the norm of the unequilibrated load and/or of the iterational displacement increment vector must vanish (to approach zero).

#### Maximum iterations

You can set the maximum number of the iterations based on the specifics of your model, and of the incremental solution parameters. By default the value is set to 20. If the convergence is not achieved within the maximum number of iterations, no results will be obtained.

## Displacement/Load/Work/Convergence criteria

In case of a nonlinear calculation you can specify multiple criteria, in terms of load, displacement, and work, for monitoring the convergence of the nonlinear solution. At least one criteria has to be selected. The criteria expressed in terms of work can be adequate for most problems. However, you may encounter a small Error in your unequilibrated load while the Error in displacements is still large, or vice-versa.

Factors of convergence criteria has the following default values: 0.001 for displacements, 0.001 for force, and 0.000001 for work.

The relative errors at the end of the iteration process appear in the info window.

E(U): relative error of the displacement convergence

E(P): relative error of the force convergence

E(W): relative error of the work convergence

#### Use reinforcement in calculation

When analyzing reinforced concrete plates it is possible to take the calculated or actual reinforcement into account.

Displacements and internal forces of reinforced concrete plates are calculated according to the moment-curvature diagram of the reinforced cross-section of the plate. These results show the actual plate deflection and forces in the plate.

#### Include geometric nonlinearity

The equilibrium is established with respect to the deformed line elements. Geometric nonlinearity can be taken into account only for truss, beam, rib and shell elements. If your model does not include nonlinear finite elements (gaps, springs, supports, and/or links), this check-box is automatically enabled. If nonlinear elements are included in the model, by enabling this check-box, you may or may not include the geometrical nonlinearity for the above mentioned line elements (truss, beam, rib and shell).

P The beam elements must be divided in at least four parts when geometric nonlinearity is taken into account.

## Store last increment only

Allows you to reduce the size of the results file when an incremental nonlinear analysis is performed with multiple increments (load or displacement) when just the results of the last increment are of interest to you. You can enable this checkbox when you do not need the results of previous increments.

æ You should disable this check-box if you want to trace the load-displacement or other *(nonlinear) response of the structure.* 

AxisVM applies a Newton-Raphson iteration technique to the iterational solution of each increment. The technique is known in different variants, depending on the update of the system (stiffness) matrix.

The following example shows the behavior of a one degree of freedom spring system with load control:





Nonlinear-elastic spring element







If n=1 (default), the system stiffness matrix is updated in each iteration. The method is known as the classical Newton-Raphson technique.

If n > MaxIterations, the system stiffness matrix is updated only once, in the first iteration of each increment. The method is known as the Modified Newton-Raphson technique.

If 1 < n < MaxIterations, a variant of the Modified Newton-Raphson technique is obtained. In the figure above the iterative process is shown for the case n=2.



The stiffening systems, usually lead to more significant numerical solution problems (than the softening systems), and solutions with n>1 can lead to divergence. This is why, when gap elements change their state (from active to inactive or vice-versa), a system stiffness matrix update is triggered, even though it would not be required based on the value specified for n.

The softening systems, and the so-called *snap-through* phenomenon cannot be analyzed with load controlled increments. You must apply a *displacement control* to pass through the peak points.

*Displacement* This figure shows a load control applied to a nonlinear *control* system. The incremental solution fails in the 5th increment.

To find the peak value of the load-displacement characteristics of the system, you must apply a displacement control technique.



## 5.2. Vibration

Lets you determine the lowest natural frequencies and mode shapes corresponding to the free vibration of an undamped linear structure when no externally applied loads are computed. AxisVM verifys whether the required number of the lowest eigenvalues has been determined.

The system mass matrix has a diagonal structure and includes only translational mass components.

The solution technique applied to the associated generalized eigenvalue problem is designed to find the lowest real and positive eigenvalues. It is not suitable to find eigenvalues that are zero or nearly zero.

lution C	ontrol			1
Vibration	(first-order) Vibart	ion (secon	d-order)	ļ
lumber of	Mode Shapes			9
lase	SelfiAr			
	100004			
Conve	t Loads to Masses			
	ncentrated Masses			
	vert Concentrated	Masses to I		
C <u>M</u> asse	s only			
	ment Masses			
Co	overt Masses to Loa			
nclude ma ⊽m _X 1ass matr	ss components	mz		
nclude ma ✔ m _X Mass matr ● <u>D</u> iagor ○ Con <u>s</u> is	ss components	m _Z		
nclude ma m _X Mass matr Diagor Con <u>s</u> is apbragn	ss components v m _Y v x type al ent (only if justified)	m _Z		
nclude ma ▼ m _X Mass matr ● <u>D</u> iagor ○ Consis aphragn ⊂ Conve	ss components To my To x type al t slabs to diaphragm	m _Z		
nclude ma max fass matr Diagor Consis conver conver converger	ss components To my To ( x type al tent (only if justified) t slabs to diaphrage ce Criteria	m _Z		
nclude ma magnetic magnetic m	ss components v type al t slabs to diaphrage ce Criteria Maximum	mz ns tterations:	30	
nclude ma ✓ m _X Mass matr ● <u>D</u> iagor ○ Con <u>s</u> is aphragn ⊂ Conver onverger	ss components v type al t slabs to diaphragm ce Criteria Maximum Eigenvalue Con	Ing Ing Iterations: wergence	30	
nclude ma ✓ m _X Mass matr ● Diagor ○ Consis aphragn □ Conver ponverger	ss components To my To the x type al t slabs to diaphrage ce Criteria Maximum Eigenvalue Com	^m z 15 Iterations: ivergence	30 1E-10	
nclude ma ✓ m _X Mass matr ● Diagor C Consis aphragn C Conve convergen	ss components To m _Y To the at term (only if justified) t slabs to diaphrage ce Criteria Maximum Eigenvalue Con Eigenvector Con	^m z Is Iterations: wergence wergence	30 1E-10 1E-5	
nclude ma ▼m _X Mass matr Piagor Consis aphragm Conver pnvergen	ss components v type al t slabs to diaphragm ce Criteria Maximum Eigenvalue Con Eigenvector Con	m _Z Ins Iterations: wergence wergence	30 1E-10 1E-5	

#### Solution control

Lets you specify the parameters of the incremental solution process:

#### **First-order**

The solution does not include the effect of axial forces of truss/beam elements on the system stiffness.

## Second-order

The solution include the effect of axial forces of truss/beam elements on the system stiffness.

Tension axial forces have a stiffening effect, while the compression axial forces have a softening effect. These effects influence the free vibrations of the structure.

## Case

Lets you select a case. The loads are converted into masses. If a second-order analysis is selected, the results of a linear (first-order) static analysis, that precedes the vibration analysis, will be accounted too.

## Number of mode shapes

Lets you specify the number of the vibration mode shapes you want to evaluate. A maximum number of 99 can be requested. The default value is 6. The value specified here can not be larger than the number of the system's mass degrees of freedom.

## Convert loads to masses

You can enable the conversion of the gravitational loads into masses, and take these concentrated masses into account.

#### Masses only

You can analyze models without loads, but with masses, and take element masses into account.

#### Include mass components

Only checked mass components will be used in the analysis. It is useful when calculating modal shapes only in a certain direction.

### Mass matrix type

*Diagonal*: smaller mass matrix but without centrifugal intertias

*Consistent* (only if justified): complete mass matrix with centrifugal intertias

## Diaphragm

When running a vibration analysis with the option *Convert slabs to diaphragms* checked, all slabs (horizontal plates) will be temporarily replaced by diaphragms.

The running time is reduced if the model contains only columns and slabs. If structural walls are included, the number of equations will be reduced but the bandwidth will be increased. The resultant running time may be greater than without diaphragms.

## **Convergence** criteria

Based on the convergence tolerances you specify, AxisVM will determine if the calculated eigenvalues and eigenvectors have the required accuracy. Therefore it is important that the convergence tolerances be set properly.

#### Maximum number of iterations

You can set the maximum number of the iterations based on the specifics of your model, and the number of eigenvalues requested (more iterations for more eigenvalues). By default the value is set to 20. If the convergence is not achieved within the maximum number of iterations, no results will be obtained.

## **Eigenvalue convergence**

Lets you specify the convergence tolerance for the eigenvalues.

The default value is 1.0E-10.

## **Eigenvector convergence**

Ø

Lets you specify the convergence tolerance for the eigenvectors. The default value is 1.0E-5.

The program uses a diagonal mass matrix by default. Due to the lumped mass modeling technique to achieve the required accuracy the elements must be divided into more elements (by refining the mesh). Usually at least four finite elements must correspond to each half wave. A good rule-of-thumb is that beams must be divided into at least eight elements.

The mode shapes are normalized with respect to the mass:

 $\{U\}^T \cdot [M] \cdot \{U\} = 1$ 

## 5.3. Dynamic Analysis

			Dynamic		
P <mark>‡¢∮\∕∳+</mark> t	₩ DYN2 [6] (0,005)	eX [mm]	Diagram	▼ 1	★ max ★ min



Dynamic analysis determines time-dependent displacements and forces due to dynamic loads or nodal accelerations.

Dynamic analysis can be performed on linear or nonlinear models.

Dynamic analysis	
Load cases Static load case or combination ST1 Dynamic load case or combination DYN1	Honlinearity         Follow nonlinear behaviour of materials         Eollow geometric nonlinearity of beams, trusses, ribs and shells
Solution Control           Time increment [s] = 0,020         Total time [s] = 3,000         Total time [s] = 0,020         N           0         150         150         150	Convergence Criteria  Perform with equilibrium iterations Maximum iterations: Disglacement 0,001  Force 0,001  Vwork 1E-6
Image: Save all steps       Rayleigh damping constants         Image: Save at regular intervals       Rayleigh damping constants         Image: At [s] = 0,020       a [Hz] = 0,43       b [s] = 0,002         Image: Consider loads and nodal masses       Image: Consider loads and nodal masses	Continge even if no convergence has been achieved
Nodal Masses         Convert Loads to Masses         Convert Additional Convert Masses to Loads         Element Masses         Convert Masses to Loads	
Mass matrix type C Diagonal	OK Cancel

#### Load cases Static load case or combination

Select the static load case or combination to apply during the analysis. Select 'None' to apply dynamic loads only.

#### Dynamic load case or combination

Select the dynamic load case or combination.

Solution control Analysis can performed in equal increments or according to a custom time increment function. Predefined functions can be loaded or a new function can be created using the function editor.

If **Equal increments** is selected two parameters are required: **Time increment** and **Total time**. Analysis uses the value of **Time increment** as the increment between time steps and **Total time** defines the total time of the analysis.

Due to the considerable result file size result saving options are introduced: Checking **Save all steps** means that all result will be saved. **Save at regular intervals** saves results only at certain model time coordinates reducing file size.

#### Rayleigh damping constants (a, b)

Damping matrix is determined from the damping contants according to the following formulas:

$$\underline{\underline{M}}\underline{\ddot{u}} + \underline{\underline{C}}\underline{\dot{u}} + \underline{\underline{K}}\underline{u} = P(t)$$
$$\underline{\underline{C}} = a\underline{\underline{M}} + b\underline{\underline{K}}$$

If *Consider loads and nodal masses* is checked another matrix will be added to  $\underline{M}$  representing loads and nodal masses.

Nodal masses Nodal masses will be taken into account like in a vibration analysis.

## Nonlinearity Follow nonlinear behaviour of materials

If nonlinear elements are defined (e.g. a tension-only truss) here you can activate or deactivate the nonlinear behaviour.

#### Follow geometric nonlinearity of beams, trusses, ribs and shells

If this option is activated loads will be applied to the displaced structure in each step.

*Convergence* Convergence criteria has to be set and will be taken into account like in a nonlinear static *criteria* analysis.

*Solution method* Linear or nonlinear equilibrium equations are solved by the Newmark-beta method. If  $\Delta t$  is the time increment, in  $t + \Delta t$  we get:

 $\underline{K} \cdot \underline{U}_{t+\Delta t} + \underline{C} \cdot \underline{\dot{U}}_{t+\Delta t} + \underline{M} \cdot \underline{\ddot{U}}_{t+\Delta t} = P(t) ,$ 

where  $\underline{C}$  is the damping matrix,  $\underline{M}$  is the mass matrix,  $\underline{K}$  is the stiffness matrix.

$$\begin{split} \underline{U}_{t+\Delta t} &= \underline{U}_t + \Delta t \cdot \underline{\dot{U}}_t + \frac{\Delta t^2}{2} \left( (1 - 2\beta) \, \underline{\ddot{U}}_t + 2\beta \underline{\ddot{U}}_{t+\Delta t} \right) \\ & \underline{\dot{U}}_{t+\Delta t} = \underline{\dot{U}}_t + \Delta t \left( (1 - \gamma) \, \underline{\ddot{U}}_t + \gamma \underline{\ddot{U}}_{t+\Delta t} \right). \end{split}$$

AxisVM uses  $\beta = 1/4$ ,  $\gamma = 1/2$ .

The differential equation of the motion is solved by the method of constant mean acceleration. This step by step integration is unconditionally stable and its accuracy is satisfying. AxisVM assumes that no dynamic effect is applied in t=0. Time-limited loads appear in t>0.  $\underline{C}$  is calculated from the Rayleigh damping constants:

$$\underline{C} = a \cdot \underline{M} + b \cdot \underline{K}$$

Where *a* and *b* should be calculated from the damped frequency range (between  $\omega_i$  and  $\omega_j$ ) and the damping ratio according to the following figure:



## 5.4. Buckling

<u>+</u>		H				
IЛ		D	F	2		
r.		r	1	ç		r
	 -		-	•	-	•

Lets you determine the lowest (initial) buckling load multipliers and the corresponding mode shapes.

AxisVM verifys whether the required number of the lowest eigenvalues has been determined. The buckling load multiplier  $n_{\alpha} = \lambda_{\alpha}$  is computed, solving the eigenvalue problem.  $\lambda_{\alpha}$  is the smallest eigenvalue and the corresponding eigenvector is the buckling mode shape.

The Sturm sequence check is applied to verify whether the computed eigenvalues are the lowest.  $\lambda_{cr} < 0$  means that buckling occurs for the

Buckling Analysis			X	
Solution Control				
Case Partial2			-	
Number of Buckling Mode	Number of Buckling Mode Shapes			
Convergence Criteria				
Maximum Iterations:	20			
Eigenvalue Convergence	1E-10			
Eigenvector Convergence	1E-5			
	ОК	0	ancel	

opposite load orientation and  $\lambda_{cr}^{effectiv} \geq |\lambda_{cr}|$ .

The solution technique applied to the associated generalized eigenvalue problem is designed to find the lowest real and positive eigenvalues. It is not suitable to find eigenvalues that are zero or nearly zero.

#### Solution control

Lets you specify the parameters of the incremental solution process: **Case** 

Lets you select a case that will be taken into account. A linear (first-order) static analysis, that precedes the buckling analysis, will be performed.

## Number of mode shapes

Lets you specify the number of the vibration mode shapes you want to evaluate. A maximum number of 99 can be requested. The default value is 6. The lowest positive eigenvalue is of main importance.

#### **Convergence criteria**

See... 5.2 Vibration/Convergence criteria

*Beams/ribs* The buckling of beams/ribs is considered as in-plane buckling (flexural buckling), which means that the deformed shape of the element remains in a plane and the cross-section does not warp.

For buckling analysis the beam cross-section must be defined by specifying its principal moments of inertia.

- The beam elements must be divided into at least four elements.
- *Trusses* The flexural buckling of truss elements are not considered by the program. You must calculate the buckling load of each truss manually, or by modeling the trusses by four beam elements with the corresponding end releases.
  - If  $\lambda_{cr} > 0$  the instability is caused by loads in the reverse direction and the critical load parameter for the given case is  $\lambda_{cr}^{efectiv} \ge |\lambda_{cr}|$
  - ^(*) If the model contains trusses the critical load parameter of global structural buckling will be computed only. Buckling of individual trusses is not analysed.

## 5.5. Finite Elements

All finite elements may be used in a linear static, nonlinear static, vibration, buckling and dynamic analysis. Note that *not* all elements have geometric stiffness.



The directions in the local coordinate system in which an element has stiffness, and the corresponding local displacement components are summarized below:

Finite	ex	ey	ez	$\theta_x$	$\theta_y$	$\theta_z$					
element	u	v	W								
Truss	*										
	2-node	e, linear,	isopara	metric e	lement						
Beam	*	*	*	*	*	*	$e_x$ $e_y$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$				
	Euler-	Euler-Navier-Bernoulli type, 2-node, cubic Hermitian element									
Rib	* * * * * * * ^V						$e_x$ $e_y$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$ $e_z$				
	Timos	henko ty	ype, 3-no	ode, qua	dratic, is	oparam	etric element				
Membrane	*	*									
	Serence	Serendipity type, 8-node, quadratic, isoparametric element									

Finite	ex	ey	ez	θx	$\theta_y$	$\theta_z$				
element	u	v	W							
Plate			*	*	*		$\begin{array}{c} V \\ \theta_{Y} \\ \end{array}$			
	Hughes type, 9-node, Heterosis isoparametric plate element									
Shell	*	*	*	*	*		$k$ $e_{y}$ $e_{y}$ $k$ $e_{y}$ $k$ $e_{y}$ $k$ $k$ $e_{y}$ $k$			
	Flat shell superimposed membrane and plate element									
Support	¥	*	¥	*	*	*	(only two components are shown)			
Spring	*	*	*	*	*	*	(only two components are shown)			
Gap	*									
Rigid										
Link	*	*	*	*	*	*	(only two components are shown for a node-to-node link)			

where:

u, v, w denote the deflections in local x, y, z directions.

 $\theta_x$ ,  $\theta_y$ ,  $\theta_z$  denote the rotations in local x, y, z directions.

* element has stiffness in the respective direction.

*Internal forces* The computed internal forces in the local coordinate system are:

Finite element	Internal forces							
Truss	Nx							
Beam	Nx	$V_y$	$V_z$	Tx	My	Mz		
Rib	Nx	$V_y$	$V_z$	Tx	My	Mz		
Membrane	nx	ny	<i>nxy</i>					
Plate				<i>m</i> _x	$m_y$	тxy	$\mathcal{U}_{XZ}$	$\mathcal{D}yz$
Shell	nx	$n_y$	<i>n</i> xy	тx	$m_y$	тxy	$v_{xz}$	$\mathcal{D}yz$
Spring	Nx	Ny	Nz	Mx	My	Mz		
Gap	Nx							
Support	Nx	Ny	Nz	Mx	My	Mz		
Rigid								
Link N-N	Nx	Ny	Nz	Mx	My	Mz		
Link L-L	nx	$n_y$	nz	mx	$m_y$	mz		

## 5.6. Main Steps of an Analysis

1.	Define the geometry of the structure, the material and cross-sectional properties of	of
	the members, the support conditions, and the loads.	

- 2. Determine the load transfer path.
- 3. Determine local discontinuities such as stiffeners, gussets, holes.
- 4. Determine the type of finite elements that will best model the behavior of the structure. With this step the properties of structural elements will be concentraded in their neutral *axis* (point, axis, or, plane).
- 5. Determine a mesh type and size for the model. The size of the mesh have to correspond to the desired accuracy of the results and with the available hardware.
- 6. Create the model:
  - a.) Equivalent geometry
  - b.) Equivalent properties
  - c.) Topology of the elements
  - d.) Equivalent support conditions
  - e.) Equivalent load (static) or masses (vibration, response-spectrum)
- 7. Check input data (accuracy, compatibility)
- 8. Run analysis
- 9. Select important results
- 10. Evaluate and check the results
  - a.) Accuracy and convergence of the solution
  - b.) Compatibility taking into account point 6.d.
  - c.) Uncommon structures shall be analyzed with other methods and/or software as well.
- 11. Restart analysis with a correspondingly updated model, if in step 10 a criteria is not satisfied.
- 12. Evaluate the results by the means of isoline/isosurface plots, animation, tables... Draw conclusions on the structure's behavior.

*g* To build a model of a structure you have to accept many assumptions so you also have to keep the effects of these assumptions in view when evaluating results.

The finite element method provides an approximative solution for surface models. To make the model match the real solution you have to use finite element meshes with an appropriate density. Making finite element meshes you have to take into account the expected stress distribution, the model geometry and the materials, supports and loads used.

The position af nodes and mesh lines (called the *topology* of the finite element mesh) depends on the geometrical discontinuities (irregular contours, line supports) and the discontinuities of loads (concentrated loads, terraced load values for line loads).

At stress concentration points (sharp corners) you have to refine the mesh. To avoid singularities due to concentrated effects you can distribute them on a small area around the point of effect.

Arc contours can be approximated as polygons. Using very small tolerance in this approximation leads to polygons with extreme small sides. The very dense mesh created on this contour may cause the model exceed the capacity of your computer.

In general if you refine the mesh you get more accurate results.

Modelling
## 5.7. Error Messages

The error messages corresponding to modeling errors are listed below:

Non-positive definite stiffness matrix

The determinant of the stiffness matrix is zero or negative due to modeling error.

Singular Jacobian matrix

Determinant of the element's Jacobian matrix is zero, due to distorted element geometry.

*Excessive element distortion during deformation* 

The element has been excessively distorted in the current increment.

*Too large rotation increment* 

The rotation increment of an element exceeds  $\pi/4$  radian (90°). You should increase the number of load increments.

Invalid conrol displacement component

The displacement control is applied about a constrained degree of freedom.

Convergence not achieved

The number of iteration is too low.

Too many eigenvalues

The rank of the mass matrix is lower than number of requested eigenvalues (frequencies or buckling modes).

No convergent eigenvalue

No eigenvalue converged.

Not the lowest eigenvalue (xx)

There are xx lower eigenvalues than the lowest the one determined

Element is too distorted

The geometry of the finite element is distorted. In order to maintain the accuracy of the results you should modify the finite element mesh to avoid too distorted element geometries.

Excessive element deformation

During a nonlinear analysis excessive deformations developed the element within an increment (load or displacement). You should increase the number of increments.

No convergence achieved within maximum number of iterations

There was no convergence within the maximum number of iterations (see... Static Analysis/Nonlinear Static Analysis/Solution Control parameters). You can increase the number of iteration. The model may not converge at the respective load level, and you should change the Solution Control parameters accordingly.

Divergence in the current iteration

A divergence was detected in the iteration process. Increments are too large or the convergence criteria are too loose.

No stiffness at node ... in direction ...

There is a singularity in the system stiffness matrix corresponding to that degree of freedom. You should check the support and degrees of freedom (DOF) settings of your model.

This page is intentionally left blank.

## 6. The Postprocessor

Static	Lets you display the results of a static analysis. (6.1)
Vibration	Lets you display the results of a vibration analysis. (6.2)
Buckling	Lets you display the results of a buckling analysis. (6.4)
R.C. Design	Lets you display the results of a reinforced concrete design analysis. (6.5)
Steel Design	Lets you display the results of a steel design analysis. (6.6)
Timber Beam Design	Lets you display the results of a timber design analysis. (6.7)

## 6.1. Static

The Static menu item allows you to display the tools for displaying and interpreting the static analysis results.



Start a linear static<br/>analysisSee... 5.1 Static AnalysisStart a nonlinear<br/>static analysisSee... 5.1 Static Analysis

Result display parameters Lets you set the options of the graphical display of the results. You can select the results of a load case/combination or critical load combination.

Display Parameters dialog shows the following options.

Display Parameters		×
$\begin{tabular}{l l l l l l l l l l l l l l l l l l l $	Component     Scale by       mxx [kNn/m]     1       Display Mode       Section Line       Undeformed       Display Shape       Undeformed       VMtke Values to       Nodes       Surfaces       Min.Max. only	Section lines  Section lines  Segment A Segment B Segment C Segment D Segment E D Correct Segment E
	Miscellanegus settings	Draw section plane contour     OK     Cancel

Analysis Type Depending on the performed analysis you can select the results of a linear or nonlinear static analysis. Each analysis type can be further defined:

#### Case

Lets you display the results of any load case/combination.

#### Envelope

Lets you display the envelope of the results from the selected load cases and/or load combinations. The program searches for the minimum and/or maximum values at each location of the selected result component.

#### Critical

Lets you generate the critical load combinations, according to the load group definitions, for each location of the selected result component.

**Display** Values If you selected envelope or critical you can choose from the following options:

### Min+Max

Displays the minimum and maximum values of the current result component.

#### Min

Displays the minimum (sign dependent) values of the current result component.

#### Max

Displays the maximum (sign dependent) values of the current result component.

Investigate all combinations resulting in the same maximum value

By default this option is off. AxisVM takes into account combinations resulting in an extreme for any result component. In certain design methods however a combination which produces no extremes can be more unfavorable.

In this case turn this option on. In design calculations AxisVM will build all possible combinations and check them according to the design code requirements. As the number of combinations can be extremely high this option is recommended only if the model size and the number of load cases are small.

Method of *Combination* 

If Critical combination formula is set to Auto AxisVM determines if ULS (ultimate limit state) or SLS (service limit state) combination is required based on the result component.

If Critical combination formula is set to Custom Min / Max / Min, Max results of all combination methods will be available in the load case combo tree regardless the current result component.

In case of Eurocode, DIN 1045-1, SIA 262 and other Eurocode based design codes the formula for creating SLS combinations can be chosen.



#### **Display Shape** Undeformed

Displays the undeformed shape (original configuration) of the model.

Deformed

Displays the deformed shape of the model.

#### Display Mode Diagram

Lets you display the current result component in a colored diagram form. The numerical values are displayed if a Show Value Labels On option is enabled.

#### Diagram+average values

This display mode is available only if line support forces are displayed. If this mode is selected line support forces diagrams are enhanced with the display and labeling of the average value. Averaging is made over continuos supports. Supports are considered to be continuous if they have the same stiffness and their angle is below a small limit.

#### Section line

Lets you display the current result component in the active section lines and/or planes in a diagram form. The numerical values are displayed if the *Show Value Labels On* option is enabled.

Isoline (contour line)

Lets you display the current result component in a line color contour plot form. The values that are represented by the isolines are specified in the Color Legend window. You can set the parameters of the Color Legend window as was described in the Information Windows paragraph. The numerical values are displayed if a *Show Value Labels On* option is enabled.

### Isosurface 2D or 3D

Lets you display the current result component in a filled color contour plot form. The ranges that are represented by the isosurfaces are specified in the Color Legend window. You can set the parameters of the Color Legend window as was described in Information Windows paragraph. The numerical values are displayed if a a *Show Value Labels On* option is enabled. **See...** 2.17.3 Color Legend Window

#### None

The current result component is not displayed.

- *Section lines* Lets you set the active section lines, planes and segments. If display mode is set to Section line result diagrams will be drawn only on active (checked) section lines. Symbol of the section planes can be displayed enabling the *Draw section plane contour* checkbox. Turning on the *Draw diagram in the plane of elements* option changes the appearance of all section diagrams. To change this parameter individually use the Section lines dialog. **See...** 2.15.12 Sections
  - *Component* Lets you select the result component to be displayed.
    - *Scale by* Lets you set the scale of a diagram drawing. The default value is 1, when the maximum ordinate is represented as 50 pixels.

#### Write Values to Nodes

•••

Writes the values of the current result component to the nodes.

#### Lines

Writes the values (intermediate values if applicable) of the current result component to the line elements.

### All surfaces

Writes the values of the current result component to the surface elements. The maximum absolute value of the nine values computed at the nodes of each surface is displayed, and the respective node is marked by a small black circle.



#### Min/max only

Writes the local min/max values only of the current result component to the nodes, lines and surfaces.



m_y moment component

R_z support force component

After clicking the *Miscellaneous Settings*... button the following options are available:

Miscellaneous settings	×
Result Smoothing Parameters	
Smoothing	Selective
Maximum Angle Allowed Between Local z Axes [°]	15,00
Maximum Angle Allowed Between Local x Axes ["]	15,00
Intensity Reference Value	
<u>Absolute maximum of the entire model in current</u>	load case or combination
C Absolute maximum of current parts in current loa	ad case or combination
C Custom Value	
	OK Cancel

Result Smoothing Parameters

#### t None

The values of the internal forces of the surface elements computed at the nodes are not averaged.

#### Selective

The values of the internal force components of the surface elements computed at the nodes are averaged in a selective way, depending on the local coordinate systems, the support conditions and the loads of the elements that are attached to a node.

### All

The values of all internal force components of the surface elements computed at the nodes are averaged.

*Intensity* Lets you display the variation of the current internal force component within the surface *Reference Value* elements in a filled color contour plot form. The numerical values are displayed if a *Show Value Labels On* option is enabled.

See... 6.1.9 Surface Elements Internal Forces



You can select a case from the drop-down list to display:

Load case, load combination The k-th increment of a nonlinear analysis Envelope display Critical combination

Available result component

eZ (mm)	•
🖃 Displacements	;
⊷eX [mm]	
−eY[mm]	
eZ (mm)	
∽ fX [rad]	
∽ fY [rad]	
fZ [rad]	
🗄 Beam Internal	Forces
🗄 Beam Stresse	8
Nodal Support	Internal Forces

	You can select a result component from the drop-down list for display:	
	Displacement (eX, eY, eZ fX, fY, fZ,eR, fR) Beam/rib internal force (Nx, Vy, Vz, Tx, My, Mz)	
	Beam/rib stress (Smin, Smax, Tymean, Tzmean)	
	Surface element internal force ( <i>nx</i> , <i>ny</i> , <i>mx</i> , <i>my</i> , <i>mxy</i> , <i>vxz</i> , <i>vyz</i> , <i>vSz</i> , <i>n</i> 1, <i>n</i> 2, <i>a nxD</i> , <i>nyD</i> , <i>mxD</i> , <i>myD</i> )	n, m1, m2, <b>a</b> m,
	Surface element stress (Sxx, Syy, Sxy, Sxz, Syz, Svm, S1, S2)	
	Nodal support force ( <i>Rx, Ry, Rz, Rxx, Ryy, Rzz</i> )	
	Line support force ( <i>Rx</i> , <i>Ry</i> , <i>Rz</i> , <i>Rxx</i> , <i>Ryy</i> , <i>Rzz</i> )	
	Surface support force ( <i>Rx</i> , <i>Ry</i> , <i>Rz</i> )	
	Spring internal force ( <i>Rx, Ry, Rz, Rxx, Ryy, Rzz</i> ) Gap internal force ( <i>Nx</i> )	
Display mode	You can select a display mode from the drop-down list:	Isosurface 2D 💌 Diagram
Ŧ	If Min,Max envelope or critical load combination is selected, the Isoline and Isosurface 2D cannot be selected.	Section Line Isoline Isosurface 2D Isosurfaces 3D None
Display scaling factor	Lets you scale the display of the diagrams.	

## 6.1.1. Minimum and Maximum Values

±max ≭min Lets you search the minimum and maximum value of the current result component. If you are working on parts, the search will be limited to the active parts. AxisVM will mark all occurrences of the minimum / maximum value.

*If parts are displayed extreme values are determined from the displayed parts only.* 

del Extreme	5	
UITACE FORCE:	8	
nx [kN/m]	nrt (kN/m)	nxD [kN/m]
ny [kN/m]	n2 [kN/m]	nyD [kN/m]
nory (kN/m)	an [°]	
mx (kNm/m)	<b>m1</b> [kNm/m]	mxD [kNm/m]
my (kNm/m)	m2 [kNm/m]	myD (kNm/m)
mxy (kNm/m)	am [°]	
vxz (kN/m)		
vyz (kN/m)		
	İ	

## 6.1.2. Animation



Lets you display the displacements, internal forces, and mode shapes in animated form (frame by frame). The animation consists of a sequence of frames that are generated by linear interpolation between initial values (frame 0) and the actual values of the current result component (frame n), according to the number of frames (n).



### Recording **Fra** Options

Lets you set the number of animationframes. You must specify a value between 3 and 99. More frames produce smoother but slower animation.

Rendered

Each frame consists of a rendered display.

## Colored

Each frame consists of an iso-line/surface display. The colors are animated according to the color legend.

*Video File* You can create a video file, *name.avi*.

Click Save button to save the parameters of the video file.

You can set the duration of displaying a frame. Lower duration will result in a bigger number of frames. A number of 30 frames/second is usual, therefore you should not normally enter less than 30 ms for the duration of a frame.

## 6.1.3. Diagram display

This dialog displays nonlinear or dynamic results as diagrams. Two diagrams can be displayed simultaneously. Each diagram has a result component on its X and Y axis. Points representing consecutive value pairs are connected. Reading coordinates can be changed by dragging the dashed lines or the black mark of the bottom trackbar. Diagram points can be displayed as a table and exported to Excel through the Clipboard.



In case of dynamic analysis the bottom trackbar displays time instead of increment numbers.





If the same X-component is chosen for the two horizontal axes their ranges can be set to the same.

*Same range on the two Y-axes* 

If the same Y-component is chosen for the two vertical axes their ranges can be set to the same.

## Fit in view in X-direction

Sets the horizontal range between minimum and maximum of X values.

Fit in view in Y-direction Sets the vertical range between minimum and maximum of Y values.

#### Interval controls

Turns on/off the green interval control rectangles of the bottom trackbar. Dragging them changes the displayed range of increments or time.

#### Point of reading

---

Turns on/off the reading point (black rectangle) of the bottom trackbar. Dragging it moves the reading point along the curve.

## 6.1.4. Pushover capacity curves



This dialog is only active if results of pushover analysis are available and it helps the user determine the capacity curve and the target displacement depending on ground motion characteristics.

A combo box on the top of the dialog lets the user select the pushover load case to be analyzed. Results are based on an acceleration-displacement response spectrum with properties specified on the left side of the dialog. These are identical to the properties of response spectra used for Seismic loads (**See...** 4.10.20 Seismic Loads). Main results of the calculations are shown both on the bottom left side of the dialog and under the diagrams themselves.

The default dialog displays a capacity curve for both the Multi Degree of Freedom System (MDOF) and the equivalent Single Degree of Freedom System (SDOF).

The sky blue curve is the capacity curve of the equivalent single degree of freedom system (SDOF). It has the same shape as the deeper blue curve for the multi degree of freedom system (MDOF). Its points are a result of dividing the corresponding force and displacement values of the MDOF curve by  $\Gamma$ .

Generally the end point of both capacity curves is the point corresponding to the maximum displacement (divided by  $\Gamma$  for the SDOF curve) set by the user at the beginning of the non-linear static analysis.

The resulting curve on the figure below shows that the structure is capable of even more displacement, since the base shear force (vertical axis) is increasing as the displacements are increasing. The maximum value for the shear force can only be determined by running another analysis limited by a larger displacement and checking if the curve reached a maximum after which the base shear started to decrease. If so, then the maximum value is at the maximum of the curve. If no maximum has been reached, the displacement has to be increased even further if necessary.



## 6.1.4.1. Capacity curves according to eurocode 8

All of the results are based on the N2 method (see 11.32) recommended in Appendix B of Eurocode 8. The bilinear force-displacement relationship for the SDOF system (green curve) is calculated by taking the force at the target displacement ( $d_t^*$ ) as the force that corresponds to yielding ( $F_y^*$ ) and defining yield displacement ( $d_y^*$ ) using the equivalent deformation energy principle.

A vertical red line marks 150% of the target displacement ( $d_t$ ) according to Eurocode 8 (4.3.3.4.2.3). Generally if the deformation capacity of the structure is above this level (the line style is dash-dot) it fulfills the deformation capability requirements, otherwise (the lines style is continuous) it fails these requirements.

## 6.1.4.2. Acceleration-Displacement Response Spectrum (ADRS)

The Acceleration-Displacement Response Spectrum (ADRS) is shown by switching to the ADRS tab on the dialog. Both elastic and inelastic ADRS spectra, SDOF and equivalent bilinear capacity curves are shown here.

A separate line highlights the natural period corresponding to the elastic behavior of the structure. The intersection of capacity and demand corresponding to the target displacement is marked by a red circle.



*Results* The variables marked by an asterisk (*) represent the SDOF system's behavior, while the others correspond to the MDOF system.

- $\Gamma$  transformation factor for computing SDOF characteristics
- *m** mass of equivalent SDOF system
- $F_y^*$  base shear force at  $d_m^*$  displacement of the equivalent SDOF system and yield force of the elasto-perfectly plastic force-displacement relationship
- $d_m^*$  ultimate displacement of the idealized bilinear force-displacement relationship (not necessarily the ultimate displacement of the SDOF system due to the iterating procedure of the N2 method)
- $d_y^*$  yield displacement of the idealized bilinear force-displacement relationship
- *T** natural period of the equivalent SDOF system
- $d_{et}^*$  target displacement of the equivalent SDOF system with period  $T^*$  and unlimited elastic behavior
- $d_t^*$  target displacement of the equivalent SDOF system considering inelastic behavior It represents the end of the green bilinear capacity curve.
- $d_t$  target displacement of the MDOF system considering inelastic behavior

#### Toolbar



Print drawing

Prints the current diagram



*Copy to Clipboard* Copies the current diagram to the Clipboard.

다리

Add drawing to Gallery

Saves the current diagram to the Gallery to make it available for reports.

## Table

Turns the table displaying numerical values on/off.

Add to Drawings Library

Saves the current diagram to the Drawings Library to make it available for reports.

## 6.1.5. Result Tables

Table Browser lets you display the numerical values of the results in a table in customizable form. If you switched on parts, the table will list the values corresponding to the active parts. If you selected elements the table will list the selected elements only by default. You can change the range of listed elements by clicking the property filter button on the Table Browser toolbar.

You can transfer data to other applications via Clipboard. See... 2.9 Table Browser.

Displaying results [ <b>Ctrl]+[R]</b>	Display Options     X       Nodal Displacements       Image: Extremes	After callir table and/o you need <i>Format / Re</i>	ng the Table Browser you can set if you need a detailed for the extremes and you can select which components the extremes from. This dialog can be called later from <i>sult Display Options</i> .
	Extremes to Find ♥ eX (mm) ♥ eY (mm) ♥ eZ (mm) ♥ eR (mm) ♥ fX (rad) ♥ fX (rad) ♥ fZ (rad) ♥ fR (rad)	Results	Unchecking this option removes the detailed results leaving the extremes as the only content of the table.
	OK Cancel	Extremes	Unchecking this option removes the summary of extremes from the end of the table.

*Extremes to find* You can set the components for which you want to find the *extreme (maximum and minimum)* values. Among the minimum and maximum values the concomitant values of the different result components are displayed if the minimum/maximum values occur in a single location or otherwise. If there are multiple locations the symbol * will appear, and in the Loc (location) column the first occurrence of the extreme value will be displayed.

When you display the *results of critical combinations* in addition to the minimum and maximum values, the load cases that lead to the critical values are included with the following notations:

- [...] represents the results of a permanent load case.
- { ... } represents the results of an incidental load case.
- (...) represents the results of an exceptional load case.

Eile	Table Browser Edit F <u>o</u> rmat <u>R</u> eport <u>H</u> el	p									<u>_ 0 ×</u>
	- Load Combinations (3 🔺	$\square$		$\times$	Ba (			52		1	
	Incidental combination		*							티	
	H Weight Report	Beam	For	ces [Lir	near, Co	o #2]					
1	RESULIS			1.0.0	Nha			Ter		14-	
	+ Displacements		Sh.	LOC.	INX ILAU	UVY DAN	V Z ILAD	IX ficklum1	WIY ficture1	IVIZ fickioni	
	- Internal Forces			[m]	[KN]	[KIN]	[KN]	[KINITI]	[KINITI]	[KINM]	
	+ Truss Intern	6	3	0	*	1,23	*	*	*	*	
	🖃 Beam Interna	20	3	0	-41,46	-0,43	-63,51	0,07	114,66	-0,01	
	- Self Weight	23	3	3,048	-41,45	0,43	61,98	-0,07	105,32	-0,01	
	Snow	3	3	0	*	*	*	-0,68	*	*	
	Wind	6	3	0	*	*	*	0,68	*	*	
	Co.#1	21	3	3,048	-39,91	-0,04	-20,31	0	-140,50	1,43	
	Co.#2	22	3	0	-39,91	0,04	18,78	0	-140,50	1,43	
	Co.#3	20	3	0	-41,46	-0,43	-63,51	0,07	114,66	-0,01	
	- Envelope M	1	2	3,658	-34,80	0,41	-18,86	0	-50,78	-1,51	
	Critical Min	4	3	3,048	-18,92	-0,09	-8,55	-0,28	-59,03	4,05	
	Beam Fnd Int	5	3	0	-18,92	0,09	7,02	0,28	-59,03	4,05	-
		,							ок	c	ancel

Property Filtering

See in detail... 2.9 Table Browser.

Print [Ctrl]+[P] Clicking the Print tool button or choosing the *File / Print* menu item the print dialog appears. **See...** 3.1.10 Print.

## 6.1.6. Displacements

Node

At each node, six nodal displacement components (three translations and three rotations) are obtained in the global coordinate system.

The resultant values of translations (eR) and of rotations ( $\theta$ R) are also determined.



### **Displaying the displacements of a cantilever** (membrane model):



Beam

For each beam element the intermediate displacements are obtained in the local and global coordinate systems. When displaying the displacements of the structure the beam displacements are related to the **global** coordinate system. If you pick the cursor on a beam element the six beam displacement components related to the element **local** coordinate system are displayed in a diagram form.

You can display displacements of more than one beam element if:

- a) The local coordinate system of the elements are almost or entirely identical.
- See... 2.15.15.3 Drawing/ Contour line angle
- b) The local x orientation is the same.
- c) The elements have the same material



You can display the diagrams corresponding to any load case or combination, as well as envelopes. You can turn on and off the display of envelope functions and set the position along the member where you want the results displayed.

Save diagramsAssociative diagrams can be saved to the Drawings Library. Drawings from this library canto the Drawingsbe inserted into reports. After changing and recalculating the model diagrams in the libraryLibraryand reports change accordingly.

6

Result Tables See... 6.1.5 Result Tables

## 6.1.7. Truss/Beam Element Internal Forces

Truss

Axial internal forces  $(N_x)$  are calculated for each truss element.

A positive axial force corresponds to tension, a negative axial force corresponds to compression.  $Nx \rightarrow Nx$ 

When displaying the *Envelope* and *Critical Combination* results the minimum and maximum values can concomitantly be displayed.

#### Displaying the internal forces of a truss girder:





Beam

Three orthogonal internal forces, one axial and two shear forces  $(N_x, V_y, V_z)$  and three internal moments, one torsional and two flexural  $(T_x, M_y, M_z)$  are calculated at the intermediate cross-sections of each element.



The internal forces are related to the element local coordinate system, and the positive sign conventions apply as in the figure above. The moment diagrams are drawn on the tension side of the beam elements.

## Displaying the internal forces of a frame:







If you click a beam element all six beam internal force components are displayed in a diagram form.

You can display internal forces of more than one beam element if:

- a) The local coordinate system of the elements are almost or entirely identical. **See...** 2.15.15.3 Drawing/ Contour line angle
- b) The local x orientation is the same.
- c) The elements have the same material.



On selecting envelope or critical load combination, the selected beam internal force minimum and maximum values of the intermediate cross sections will be displayed.

You can display the diagrams corresponding to any load case or combination, as well as envelopes. You can turn on and off the display of envelope functions and set the position along the member where you want the results displayed.

Associative diagrams can be saved to the Drawings Library. Drawings from this library can

be inserted into reports. After changing and recalculating the model diagrams in the library

Save diagrams to the Drawings Library

*Library* and reports change accordingly.

Result Tables

If the min/max values occur in a single location the concomitant values of the afferent internal force components are displayed, or the symbol * (if there are multiple locations). An occurrence of such a location is displayed.

See... 6.1.5 Result Tables

## 6.1.8. Rib Element Internal Forces

Three orthogonal internal forces, one axial and two shear forces  $(N_x, V_y, V_z)$  and three internal moments, one torsional and two flexural  $(T_x, M_y, M_z)$  are calculated at the nodes of each element. The rib can be used independently (not connected to a surface element), or connected to a surface element.



The internal forces are related to the element local coordinate system positioned in the center of gravity of the cross-section, and the positive sign conventions apply as in the figure below. The moment diagrams are drawn on the tension side of the beam elements. If the rib is connected eccentrically to a shell element, axial forces will appear in the rib and in the shell.





Result Tables See... 6.1.5 Result Tables

## 6.1.9. Surface Elements Internal Forces

Internal forces

The internal forces and the positive sign conventions of each surface element type are summarized in the table below.





Displaying the internal forces of a ribbed plate:

The x and y index of the plate moments indicates the direction of the normal stresses that occur due to the corresponding moment, and not the rotation axis.

So, the  $m_x$  moment rotates about the *y* local axis, while the  $m_y$  about the *x* local axis.

The moment diagrams of plate and shell elements are drawn on the tension side. On the top surface (determined by the local z direction) the sign is always positive, on the bottom surface it is always negative.

Intensity variation The finite element method is an approximate method. Under *normal circumstances* the results converge to the exact values as the mesh is refined. The refinement of the mesh (the number of the elements used in the mesh), the geometry of the elements, the loading and the support conditions, and many other parameters influence the results. Therefore some results will be relatively accurate whereas other results require

the user to determine if they meet the conditions of accuracy that he expects.

The intensity variation values are intended to give you help in identifying the regions in your model (mesh) where it is possible that the accuracy of the results is not satisfactory, without performing an additional analysis. This method does not show that the results are good, but will highlight intensity variations with high magnitudes, where you may want to check and/or refine your mesh.

The allowable values of the intensity variation can be determined based on practice.

Result Tables See... 6.1.5 Result Tables

Principal forces

The  $n_1$ ,  $n_2$ ,  $\alpha_n$ ,  $m_1$ ,  $m_2$ ,  $\alpha_m$  principal internal forces and the vSz resultant shear forces are computed. The sign conventions are as follows:

 $m_1 \geq m_2 \;, \quad n_1 \geq n_2$ 

 $-90^{\circ} < \alpha \le +90^{\circ}$  (relative to the local x axis)



	Sh	ell
	Membrane	Plate
$n_1$	$n_1 = \frac{n_x + n_y}{2} + \sqrt{\left(\frac{n_x - n_y}{2}\right)^2 + n_{xy}^2}$	-
<i>n</i> ₂	$n_2 = \frac{n_x + n_y}{2} - \sqrt{\left(\frac{n_x - n_y}{2}\right)^2 + n_{xy}^2}$	-
$\alpha_n$	$tg(2\alpha_n) = \frac{2n_{xy}}{n_x - n_y}$	-
$m_1$	-	$m_1 = \frac{m_x + m_y}{2} + \sqrt{\left(\frac{m_x - m_y}{2}\right)^2 + m_{xy}^2}$
<i>m</i> ₂	-	$m_2 = \frac{m_x + m_y}{2} - \sqrt{\left(\frac{m_x - m_y}{2}\right)^2 + m_{xy}^2}$
$lpha_m$	-	$tg(2\alpha_m) = \frac{2m_{xy}}{m_x - m_y}$
vSz	-	$vSz = \sqrt{v_{xz}^2 + v_{yz}^2}$

In the case of plane strain membrane elements,  $n_z \neq 0$  and is not determined.

Gerror The internal forces can be displayed in diagram, section line, isoline or isosurface forms.

The principal directions ( $\alpha_n$ ,  $\alpha_m$ ) can be displayed only in diagram form.

The direction vector color and size are determined based on the value of the respective principal internal forces.

If the principal internal force is negative the corresponding direction vector is bounded by two segments perpendicular to it.



Result Tables See... 6.1.5 Result Tables

Reinforcement forces For surface elements nxv, nyv, mxv, myv reinforcement (design) forces and moments are also calculated according to the following rules:

$$n_{xv} = n_x \pm |n_{xy}| \quad , \quad n_{yv} = n_y \pm |n_{xy}|$$
$$m_{xv} = m_x \pm |m_{xy}| \quad , \quad m_{yv} = m_y \pm |m_{xy}|$$

 $\mathop{{ \mbox{\tiny CV}}}$  The reinforcement design forces can be displayed in diagram, section line and iso-line / surface colored form.

## 6.1.10. Support Element Internal Forces

↔ The internal forces can be displayed in diagram or colored form. In the case of nodal supports, when displaying in diagram form, the internal force components are represented as vectors.

The resultant internal forces  $R_{eR}$ ,  $R_{\theta R}$  are computed as follows:

$$R_{eR} = \sqrt{R_{ex}^2 + R_{ey}^2 + R_{ez}^2} \qquad R_{\theta R} = \sqrt{R_{\theta x}^2 + R_{\theta y}^2 + R_{\theta z}^2}$$

## Displaying the internal forces of supports in a frame and a shell structure:





ReR resultant forces

Ry edge forces





Result Tables

See... 6.1.5 Result Tables

Diagram +average values When displaying line support forces a special display mode (Diagram + average values) is available. If this mode is selected line support forces diagrams are enhanced with the display and labeling of the average value. Averaging is made over continuos supports. Supports are considered to be continuous if they have the same stiffness and their angle is below a small limit. Labels also show the length of the averaging segment.





## 6.1.11. Internal forces of line to line link elements and edge hinges

Internal forces AxisVM determines the *nx*, *ny*, *nz* forces and *mx*, *my*, *mz* moments for line to line link elements and edge hinges. If any stifness component is set to zero the related result component is zero and not displayed neither in the component combo nor in result tables.

## 6.1.12. Truss/Beam/Rib Element Stresses

The display modes for stress results are the same as for the internal forces. The table of the stress results are similar to those of internal forces.

Truss The  $S_x = N_x/A_x$  stress value is calculated for each truss element. A positive value means tension.

Beams / Ribs The following stress values are calculated in each stress point of each cross-section of the beam/rib element:

Normal stress from tension/compression and bending is calculated disregarding warping stress:

$$S_{x,i} = \frac{N_x}{A_x} + \frac{M_y I_z + M_z I_{yz}}{I_y I_z - I_{yz}^2} z_i - \frac{M_z I_y + M_y I_{yz}}{I_y I_z - I_{yz}^2} y_i$$

where  $y_i$ ,  $z_i$  are the stress point coordinates. Positive stress value means tension in the cross-section.

Resultant shear stress is calculated from shear and twisting (Saint-Venant) disregarding warping shear stress.

For thick-walled cross-sections  $V_i = \sqrt{V_{y,i}^2 + V_{z,i}^2}$ ,

where shear stress components are:

$$\begin{split} V_{y,i} &= \frac{V_y}{A_x} \left( \frac{\partial \boldsymbol{\varPhi}_y}{\partial y} \right)_i + \frac{V_z}{A_x} \left( \frac{\partial \boldsymbol{\varPhi}_z}{\partial y} \right)_i + \frac{M_x}{I_x} \left( \left( \frac{\partial \boldsymbol{\omega}}{\partial y} \right)_i - z_i \right) \\ V_{z,i} &= \frac{V_y}{A_x} \left( \frac{\partial \boldsymbol{\varPhi}_y}{\partial z} \right)_i + \frac{V_z}{A_x} \left( \frac{\partial \boldsymbol{\varPhi}_z}{\partial z} \right)_i + \frac{M_x}{I_x} \left( \left( \frac{\partial \boldsymbol{\omega}}{\partial z} \right)_i + y_i \right) \end{split}$$

 $\Phi_y$  and  $\Phi_z$  are the shear stress functions for shear in y and z direction,  $\omega$  is the warping function.

For thin-walled cross-sections:

$$V_{i} = \left| \frac{V_{y}}{A_{x}} \left( \frac{\partial \boldsymbol{\Phi}_{y}}{\partial s} \right)_{i} + \frac{V_{z}}{A_{x}} \left( \frac{\partial \boldsymbol{\Phi}_{z}}{\partial s} \right)_{i} + \frac{M_{x}}{I_{x}} \left( \left( \frac{\partial \boldsymbol{\omega}}{\partial s} \right)_{i} + m_{i} \right) \right| + \frac{M_{x}}{I_{x}} t_{i} ,$$

where the last two terms are the shear stress from twisting derived from shear flow in closed and open subsections.  $m_i$  is the distance of the centre of gravity from the segment,  $t_i$  is the wall thickness of the segment.  $\omega$ ,  $\Phi_v$  and  $\Phi_z$  are centerline values.

Von Mises stress is defined as  $S_{o,i} = \sqrt{S_{x,i}^2 + 3V_i^2}$ 

If a cross-section contains two or more separate parts  $V_i$  and  $S_{o,i}$  is not calculated.

Mean shear stresses:  $V_{y,mean} = V_y / A_y$ ,  $V_{z,mean} = V_z / A_z$ , if Ay, Az = 0 then Ay = Az = Ax.

Beam stresses *Sminmax, Vminmax, Sominmax* are minimum / maximum values within the cross-section and displayed like internal forces.

You can click a beam/rib element to display stress diagrams. On the left the minimum/maximum values along the line are displayed. Dragging the blue line with the mouse the evaluation position can be changed. The axonometric diagrams in the middle and the tables on the right show the stress distribution within the section at the evaluation point. Select more elements before clicking to display them in one diagram. Continuous beams/ribs can be displayed in one diagram if conditions described in section 6.1.7 Truss/Beam Element Internal Forces are satisfied.



You can display the diagrams corresponding to any load case or combination, as well as envelopes. You can turn on and off the display of envelope functions and set the position along the member where you want the results displayed.

Save diagrams to the Drawings Library Associative diagrams can be saved to the Drawings Library. Drawings from this library can be inserted into reports. After changing and recalculating the model diagrams in the library and reports change accordingly.

6

œ

Selecting envelope or critical combinations only one of the min and max components will appear depending on the component. If extreme values are located in one cross-section only you will see values of the other components as well. Otherwise a * will appear and the cross-section location will be the first one.

Result Tables See... 6.1.5 Result Tables

## 6.1.13. Surface Element Stresses

Component	Membrane	Plate	Shell
S _{xx}	$s_{xx} = \frac{n_x}{t}$	$s_{xx} = \pm \frac{6}{t^2} \cdot m_x$	$s_{xx} = \frac{n_x}{t} \pm \frac{6}{t^2} \cdot m_x$
$S_{yy}$	$s_{yy} = \frac{n_y}{t}$	$s_{yy} = \pm \frac{6}{t^2} \cdot m_y$	$s_{yy} = \frac{n_y}{t} \pm \frac{6}{t^2} \cdot m_y$
S _{xy}	$s_{xy} = \frac{n_{xy}}{t}$	$s_{xy} = \pm \frac{6}{t^2} \cdot m_{xy}$	$s_{xy} = \frac{n_{xy}}{t} \pm \frac{6}{t^2} \cdot m_{xy}$
S _{XZ}		$s_{xz} = \frac{3v_{xz}}{2t}$	$s_{xz} = \frac{3v_{xz}}{2t}$
S _{yz}		$s_{yz} = \frac{3v_{yz}}{2t}$	$s_{yz} = \frac{3v_{yz}}{2t}$

The following stress components are calculated at each node of the element in the top, center, and bottom fiber:

In the case of plane strain membrane elements  $s_{zz} \neq 0$ , and is determined as  $s_{zz} = v \cdot (s_{xx} + s_{yy})$ 

In case of moments the x or y suffix refers to the direction of the section, therefore mx moment will make the plate rotate around the local y direction and my around the local x direction.

Von Mises stresses

es The Von Mises stress is computed:

$$s_o = \sqrt{0.5 \left[ (s_{xx} - s_{yy})^2 + (s_{yy} - s_{zz})^2 + (s_{zz} - s_{xx})^2 \right] + 3 \left( s_{xy}^2 + s_{yz}^2 + s_{zx}^2 \right)}$$

Ger Stress values can be displayed as a diagram, section diagram, as isolines or isosurfaces.

Result Tables See... 6.1.5 Result Tables

## 6.1.14. Influence Lines

Displays the internal force influence lines corresponding to the unit applied forces Px, Py, Pz that act in the positive direction of the global coordinate axes. An ordinate of the influence line represents the value of the respective internal force that occurs in the respective cross-section caused by an applied unit force at the position of the ordinate.

Truss

Clicking a truss shows the elements' absolute maximum ordinate value.

## Displaying the axial force influence line diagrams of a truss girder:



Beam

Clicking a beam shows the elements' absolute maximum ordinate value and its location.

## Displaying the internal force influence line diagrams of a frame:





## 6.1.15. Unbalanced Loads

Edit Format Report He -Load Cases (3) -Load Groups (2) -Load Combinations (3) Weight Report	p	Ê	■ 8		d 🛐			
Results	Name	Forces	F _X [kN]	F _γ [kN]	F _Z [kN]	M _X [kNm]	M _Y [kNm]	M _Z (kNm)
🖃 Linear Analysis	1 Self Weight	E	0	0	-76.97	-731.30	469.20	0
Displacements		UNB	0	0	0	0	0	0
- Internal Forces	2 Snow	E	0	0	-292.61	-2675.62	1783.75	0
Beam Internal		UNB	0	0	0	0	0	0
+ Beam End Inte	3 Wind	E	58.52	25.00	0	-89.19	104.40	-382.72
+ Nodal Support		UNB	0	0	0	0	0	0
Stresses     Unbalanced Loads     Libraries     Material Library     Cross-Section Libr								

The resultant of all external loads with respect to the origin of the global coordinate system is calculated (in the direction X, Y, Z, XX, YY, ZZ) for each load case.

The unbalanced loads for each load case is also displayed (UNB) by its components (in the direction X, Y, Z, XX, YY, ZZ). The unbalanced loads are not appearing in the supports, therefore, if there are non-zero unbalanced load components, it usually means that a part of the external loads are supported by constrained degrees of freedom and not the supports.

*The second of the unbalanced loads after each analysis run.* 

## 6.2. Vibration

Geometry	Elements	Loads	Static	Vibration	Buckling	R.C. Design	Steel Design		
" <del>hor</del> t	Mode 1	1 (1,39 H	z) 🔽 e	Z	• Isos	urface	▼1 ♣	±max ≭min	

Displays the results of a vibration analysis (mode shapes and frequencies). You must specify the mode shape number.

The mode shapes are normalized with respect to the mass.

## **Displaying mode shapes:**





Plate, second mode



In the Info Window the following will appear:

f	the frequency
ω	the circular frequency
Т	the period
Ev	the eigenvalue
Error	the relative Error of the eigenvalue
Iteration	the number of iteration performed until convergence was achieved
AxisVM store	s the vibration analysis results corresponding to each case.

Result table See... 6.1.5 Result Tables

¢,

## 6.3. Dynamic



Displays the results of a dynamic analysis.

Available settings and display modes are the same as for static results. See... 6.1 Static.

## 6.4. Buckling

C)

Geometry	Elements	Loads	Static Vi	ibration <b>Bu</b>	uckling	R.C. Design	Steel De	esign		
^{††} Per ₿	Mode 1	(6,644)	▼ eZ		▼ Isosu	rface	▼ 1,5	•	±max ≭min	

Displays the results of a buckling analysis (buckling mode shapes and critical load parameters).

In the Info Window the following will appear:

Buckling of a frame:





AxisVM stores the buckling analysis results corresponding to each case.

## 6.5. R.C. Design

## 6.5.1. Surface Reinforcement

	R.C. Design	
Sosurface 2D	▼ 1 ★ ≭max ≭min	፲॰ 📮 ᆍ 占 cobiax

Design Codes

Result components

Eurocode 2: EN 1992-1-1:2004 DIN: DIN 1045-1:2001-07 SIA: SIA 262:2003

Surface reinforcement can be calculated based on Eurocode 2. The calculation of the reinforcement of membrane, plate, and shell elements is based on the 3rd stress condition. Reinforcement directions are the same as the local x and y directions. The nominal moment and corresponding axial strengths are determined based on the restricted direction optimal design.

The minimum reinforcement is not calculated. If the amount of reinforcement that is calculated is less than the minimum reinforcement, the calculated values are informative only, and are not based on the assumptions of an under reinforced design.



	*
mxD, myD,	
nxD, nyD:	design forces
axb:	calculated reinforcement area at the bottom in x direction
ayb:	calculated reinforcement area at the bottom in y direction
axt:	calculated reinforcement area at the top in x direction
ayt:	calculated reinforcement area at the top in y direction
xb:	actual (applied) reinforcement at the bottom in x direction
yb:	actual (applied) reinforcement at the bottom in y direction
xt:	actual (applied) reinforcement at the top in x direction
yt:	actual (applied) reinforcement at the top in y direction
xb–axb:	reinforcement difference at the bottom in x direction
yb–ayb:	reinforcement difference at the bottom in y direction
xt–axt:	reinforcement difference at the top in x direction
yt–ayt:	reinforcement difference at the top in y direction
vRd,c:	shear resistance
vSz–vRd,c:	difference between the resultant shear force perpendicular
	to the surface and the shear resistance
wk(b)	crack opening in the axis of bottom reinforcement
wk(t)	crack opening in the axis of top reinforcement
wk2(b)	crack opening at the bottom of the plate
wk2(t)	crack opening at the top of the plate
wR(b)	crack direction at the bottom of the plate
wR(t)	crack direction at the top of the plate

Reinforcement parameters

In the surface reinforcement design, the following parameters must be assigned to the finite elements:

		Jertiles
Surface Reinforcement Parameters (Eurocode)	×	Steel rebar properties
Materials Concrete	2/15	
Rebar steel B5	00A	
Thickness (h) [cm] = Unfavorable eccentricity (N > 0) =	29,0 • 0 • *h	
Unfavorable eccentricity (N < 0) =	0 🔹 *h	
Position	n] = 3,0 🔽	
× _{bottom} [cm] = 3,0 Y _{bottom} [cr	n] = 3,0 •	Reinforcement cover position
☐ Use this rebar steel and concrete cover by default		
Pick Up >>	Cancel	

Materials Concrete material, rebar material

*Thickness* **h** is the total thickness used in the calculation

Unfavorable It has to be added in case of Eurocode2.

*eccentricity* Extra eccentricities will always be added to the actual value (calculated from normal forces and moments) to increase the absolute value of the excentricity.

*Position* **xbottom**, **ybottom**, **xtop**, **ytop** position (< h/2)

The position of the reinforcement is defined as the distance between the edge of the concrete and the axis of the rebar.



## 6.5.1.1. Calculation based on Eurocode 2

Plate

If  $m_x$ ,  $m_y$ ,  $m_{xy}$  are the internal forces at a point, then the nominal moment strengths are as follows:

the moment optimum is:	$\Delta m_2 = 0$	m >m
- the moment optimum is.	$\Delta m_1 = \min!$	$m_{\chi} \ge m_{y}$



# *Results* AxisVM calculates the tension and/or compression reinforcements (for doubly reinforced sections).

Membrane

Only plane stress membranes can be reinforced.

If  $n_x$ ,  $n_y$ ,  $n_{xy}$  are the internal forces at a point, then the nominal axial strengths are as follows: - the axial force optimum is:  $\Delta n_2 = 0$ 

 $\Delta n_1 = \min!$ 

 $n_y \ge n_x$ 



- *Results* AxisVM calculates the tension or compression reinforcements. Compression reinforcement is calculated only in the points at which the axial compression resistance of the section without reinforcement is lower than the compressive design axial force.
- Shell If  $n_x$ ,  $n_y$ ,  $n_{xy}$ ,  $m_x$ ,  $m_y$ ,  $m_{xy}$  are the internal forces in a point, than the design axial forces and moments are established based on the reserve axial force optimum and reserve moment optimum criterias that were emphasized, at the membrane reinforcement and plate reinforcement description.

The program calculates the necessary tensile and compressive reinforcement.

- ResultsThe following values are provided as results: axb, axt, ayb, aytTotal reinforcement in x direction: Ax = axb+axtTotal reinforcement in y direction: Ay = ayb+ayt
  - The total amount of reinforcement necessary is Ax + Ay.
  - The error message The section cannot be reinforced appears if:  $Ax > 0,04A_c$ , or  $Ay > 0,04A_c$ where  $A_c$  is the concrete cross-section area.
  - *Tables* The following symbols are used in tables:(-) compression reinforcement bar??? the section cannot be reinforced in the corresponding direction

#### 6.5.1.2. Calculating based on DIN 1045-1 and SIA 262

 Plate, Membrane,
 Reinforcement of membranes, plates and shells are calculated according to the three-layer method.

The internal forces  $(m_{x'}, m_{y'}, m_{xy'}, n_{x'}, n_{y'}, n_{xy})$  are calculated in the perpendicular directions of the reinforcement.

The surface is divided into three layers. Membrane forces for the top and bottom layers are calculated then design forces and the required amount of reinforcement is determined.



Besides calculating the required reinforcement zones of concrete are checked for shear and compression according to *A*, *B* and *C* cases.



Error message

*The error message The section cannot be reinforced appears.* 

If the compressed zone of the concrete fails due shear forces.

If the compression principal stress is higher than  $f_{cd}$ .

 $Ax > 0,04A_c$ , or  $Ay > 0,04A_c$ , where  $A_c$  is the concrete cross-section area.

*Tables* The following symbols are used in tables:

(-) compression reinforcement bar

??? the section cannot be reinforced in the corresponding direction

No symbol appears when tension reinforcement is required.

## 6.5.2. Actual Reinforcement

Actual Reinforcement Lets you apply an actual reinforcement to the surface elements depending on the calculated reinforcements.

Using the actual reinforcement you can perform a non-linear plate deflection analysis.

There are two ways to define actual reinforcement:

- 1.) select surface elements or domains then click the button on the toolbar to specify reinforcement
- 2.) click the button with no selection, specify reinforcement then draw meshindependent reinforcement domains.

Rebar statistics

The actual reinforcement within the model can be checked by displaying *Rebar statistics* in the *Weight Report* section of the *Table Browser*. This table lists total length and mass of rebars and the total reinforced concrete surface and volume per rebar diameter.



### 6.5.2.1. Reinforcement for surface elements and domains

#### Reinforcement



The actual reinforcement of the selected surfaces is shown in the tree on the left. Selecting a reinforcement makes its parameters editable on the right. Changing the values updates the tree.

- *Min. Thickness* Min. Thickness displays the minimum thickness entered as surface reinforcement parameter for the selected elements, and not the minimum thickness of the elements.
  - *The position of the rebar is defined as the distance between the side of the concrete and the axis of the rebar.*



*Add and Delete* The applied reinforcement is shown in a tree view on the left. By selecting a reinforcement you can change its parameters in the right side. By selecting a location (e.g. x Direction / Top Reinforcement) you can set a new reinforcement on the right side and add it.

Use the Delete button (or **[Del]** key) to delete reinforcement or the Add button (or **[INS]** key) to add reinforcement to a group. If you select a node of the tree view the Delete button (or **[Del]** key) will delete all the reinforcements under that node. The Add button (or **[INS]** key) will add reinforcement to the corresponding group.

Max. Reinforcement in Selection

Parameters according to design codes In the Max. Reinforcement in Selection group box the maximum calculated reinforcement values are displayed corresponding to different directions of the selected elements.

Load duration C Short term (kt = 0.6) C Long term (kt = 0.4)	Eurocode, Swiss code (SIA), Italian code
Min. Thickness (h) [cm] = 29,0	
Primary direction of reinforcement Top Surface $c_T [cm] = 1.6$ $\ge 1.6$ $\bigcirc \times \bigcirc y$ $h$ $\longrightarrow \times$ $\bigcirc \times \bigcirc y$ $h$ $\longrightarrow \times$ Bottom Surface $c_B [cm] = 1.6$ $\checkmark \ge 1.6$ $\checkmark$ Apply minimum cover	Dutch code (NEN)
Rebar steel	
No additional parameters.	German code (DIN 1045-1)

#### 6.5.2.2. Mesh-independent reinforcement

To define mesh-independent reinforcement set reinforcement first then draw rectangular or polygonal reinforcement domains.

If no surfaces or domains are selected clicking the button on the toolbar displays this dialog.

Actual Reinforcement	د	<
Parameters (Eurocode) Reinforcement		•
<ul> <li>x Direction         <ul> <li>Top Reinforcement = 914</li> <li>16 mm / 220 mm (3.0 cm) [R]</li> <li>Bottom Reinforcement = 1340</li> <li>16 mm / 150 mm (3.0 cm) [R]</li> </ul> </li> <li>y Direction         <ul> <li>Top Reinforcement = 914</li> <li>16 mm / 220 mm (4.6 cm) [R]</li> <li>Bottom Reinforcement = 1340</li> <li>16 mm / 150 mm (4.6 cm) [R]</li> </ul> </li> </ul>	Rebars         Type       Ribbed         Ø [mm] =       16         Ø [mm] =       220         Spacing [mm] =       220         Rebar position [cm] =       4.6         A _s [mm ² /m] =       914	
<u>Pick Up &gt;&gt; 🔓 👘 🏠 🗐 🦓 💮 🚺 </u>	★ 약 914 전 1340Close	

Reinforcement can be added or deleted the same way as above.

The dialog can be reduced to a toolbar. Clicking the triangle icon at the top right corner shrinks or opens up the dialog. The reinforcement amounts specified are displayed as symbols. The amounts of top and bottom y reinforcement are written along the vertical line. The amounts of top and bottom x reinforcement are written along the horizontal line.

	Actual Reinforcement
	Pick Up >> $k_2$ $m$ $m$ $m$ $m$ $m$ $m$ Pick Up >> $k_3$ $m$ $m$ $m$ $m$ $m$
	Toolbar icons:
$\searrow$	Displays the selection toolbar to select existing domains. The current reinforcement is applied when the selection is completed.
	Option to draw rectangular reinforcement domains.
	Option to draw skewed rectangular reinforcement domains.
$\bigcirc$	Option to draw polygonal reinforcement domains.
	Option to apply reinforcement to domains just by clicking them.
	Reinforcement is applied only where reinforcement domains fall on surface elements or domains.
Ħ	Contours of reinforcement domains are identified by the cursor. Clicking reinforcement domains allow making changes in the reinforcement. <b>[SHIFT]</b> + clicking selects multiple reinforcement domains. Clicking on one of the selected domains allow making changes in multiple reinforcement domains. This is the same method used for elements or mesh-independent loads.
	Mesh-independent reinforcement domains are displayed as contours made of dashed brown lines. A symbol showing top and bottom reinforcement amounts in <i>x</i> and <i>y</i> directions appear at the center. Centerpoint is connected to two vertices of the domain polygon by continuous brown lines.
	When modifying an existing reinforcement domain two methods are available:
<b>Overwrite</b>	New reinforcfement overwrites the existing one.
*0	New reinforcement is added to the existing one.

## 6.5.3. Crack Opening Calculation

Add

Design Codes

Eurocode 2: EN 1992-1-1:2004 DIN: DIN 1045-1:2001-07

After the assignment of the actual reinforcement the program calculates the crack openings and crack directions in the membrane, plate and shell elements.

The direction of the reinforcement is relative to the surface element's local x and y axes. The program displays the crack openings in a color coded mode, can draw the crack map and the crack angles.

The set of the parameters can be seen in the previous section.

*Results* In the table of results the following information can be found:

Aax, Aay actual reinforcement in x and y direction

- *wk* crack opening at the axis of the rebar
- *wk*2 crack opening at the edge of the slab
- $x_{s2}$  position of the neutral axis relative to the edge on the compressed side
- $\sigma_{s2}$  rebar stress
- *wR* angle of cracking relative to the local x direction

nx, ny, nxy, mx, my, mxy surface forces and moments





A warning message will appear if the calculated rebar stress is higher than the characteristic yield strength.

The calculation of crack opening is based on the actual reinforcement assigned to the surfaces.

### 6.5.3.1. Calculation based on Eurocode 2

 $w_k = s_{r,\max} \cdot (\varepsilon_{sm} - \varepsilon_{cm})$ , where  $s_{r,\max}$  is the maximum cracking,  $\varepsilon_{sm}$  is the strain of the rebar,  $\varepsilon_{cm}$  is the strain of the concrete between cracks.

Ŧ

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_{s2} - k_t \frac{f_{ctm}}{\rho_{\rho,eff}} (1 + \frac{E_s}{E_{cm}} \rho_{\rho,eff})}{E_s} \ge 0.6 \cdot \frac{\sigma_{s2}}{E_s}$$
$$s_{r,\max} = 3.4 \cdot c + 0.425 \cdot k_1 \cdot k_2 \frac{\overline{\phi}}{\rho_{\rho,eff}}, \text{ where }$$

- $\overline{\phi}$  is the average rebar diameter,
- c is the concrete cover,

k₁ is a factor depending on rebar surface (ribbed or plain),

- k₂ is a factor depending on the character of the eccentric tension,
- $k_t$  is a load duration factor

for short term loads	$k_t = 0,6$
for long term (permanent) loads	$k_t = 0,4$

 $\rho_{\rho,eff} = \frac{A_s}{A_{c,eff}}$  is the effective reinforcement ratio.

If plain rebars are used or the spacing of ribbed rebars exceeds  $5 \cdot (c + \overline{\phi}/2)$ , then  $s_{r,\max} = 1, 3 \cdot (h - x_2)$ .

The program takes account of the fact that cracking is not perpendicular to any of the reinforcement directions and calculates its angle relative to the x axis.

## 6.5.3.2. Calculation based on DIN 1045-1

 $w_k = s_{r,\max} \cdot (\varepsilon_{sm} - \varepsilon_{cm})$ , where  $s_{r,\max}$  is the maximum cracking,  $\varepsilon_{sm}$  is the strain of the rebar,  $\varepsilon_{cm}$  is the strain of the concrete between cracks.

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_{s2} - 0.4 \cdot \frac{f_{ctm}}{\rho_{eff}} (1 + \frac{E_s}{E_{cm}} \rho_{eff})}{E_s} \ge 0.6 \cdot \frac{\sigma_{s2}}{E_s}$$
$$s_{r,max} = \frac{\overline{d}}{3.6 \cdot \rho_{eff}} \le \frac{\sigma_{s2} \cdot \overline{d}}{3.6 \cdot f_{ctm}}, \text{ where}$$

*d* is the average rebar diameter

 $\rho_{eff} = \frac{A_s}{A_{c,eff}}$  is the effective reinforcement ratio.

The program takes account of the fact that cracking is not perpendicular to any of the reinforcement directions and calculates its angle relative to the x axis.

## 6.5.4. Non-linear deflection of RC plates

In case of the linear static analysis the plate deflection is calculated according to the elastic theory. In fact the behaviour of RC plates is non-linear due to two opposite effects. The actual reinforcement increases the bending strength but cracking decreases it.

The non-linear RC plate deflection analysis follows up these two effects with the actual reinforcement.

The program performs a non-linear analysis in an iterative way using the momentcurvature diagrams of RC cross-sections. The strength effect of the tensile concrete is also taken into account.

This non-linear analysis is available based on Eurocode, DIN 1045-1 (German), SIA-262 (Swiss), NEN (Dutch), MSz (Hungarian) and STAS (Romanian) design codes.

## The main **steps of a plate deflection calculation** are

- 1.) performing a linear analysis of the plate
- 2.) calculating the required reinforcement
- 3.) applying the actual reinforcement
- 4.) performing a non-linear analysis of the plate
- When you start the non-linear analysis, check the Use actual reinforcement in the calculation checkbox.

Plate deflection:





Linear (elastic) analysis

Non-linear analysis

## 6.5.5. Shear resistance calculation for plates and shells

Design Codes

Eurocode 2: EN 1992-1-1:2004 DIN: DIN 1045-1:2001-07 SIA: SIA 262:2003

AxisVM calculates the shear resistance of the reinforced plate or shell without shear reinforcement, the normal shear force and the difference between them.

 $v_{Sz} = \sqrt{v_{xz}^2 + v_{yz}^2}$  is the resultant shear force, where  $v_{xz}$ , and  $v_{yz}$  are the shear force components in planes with normals in the local x and y direction.

 $\phi = \arctan(v_{yz} / v_{xz})$  is the angle of the normal of the plane, in which resultant shear force of  $q_{Rz}$  acts.

 $d = (d_x + d_y)/2$  is the average effective height.

 $\rho_1 = \sqrt{\rho_x \cdot \rho_y}$  is the reinforcement ratio of the longitudinal reinforcement.

 $\rho_x$  and  $\rho_y$  are rebar ratios calculated from tension reinforcement in *x* and *y* directions of the reinforcement.

The calculation of the shear resistance is based on the actual reinforecement assigned to the surfaces.
#### 6.5.5.1. Calculation based on Eurocode 2

Shear resistance is

$$\begin{split} V_{Rd,c} &= \left[ C_{Rd,c} \cdot k \cdot (100 \cdot \rho_l \cdot f_{ck})^{1/3} + k_1 \cdot \sigma_{cp} \right] \cdot d \ge (v_{\min} + k_1 \cdot \sigma_{cp}) \cdot d \text{ , where} \\ C_{Rd,c} &= 0.18 / \gamma_c \text{ , } k = 1 + \sqrt{(200/d)} \le 2.0 \text{ , } k_1 = 0.15 \\ \sigma_{cp} &= \frac{N_{Ed}}{A_c} \le 0.2 \cdot f_{cd} \text{ , } v_{\min} = 0.035 \cdot k^{3/2} \cdot f_{ck}^{1/2} \end{split}$$

 $N_{Ed}$  is the normal force in the shell perpendicular to the plane of  $q_{Rz}$ .  $N_{Ed}$  is positive in compression.

The reinforcement ratio is  $\rho_l \leq 0.02$ .

Gev The  $V_{Rdc}$  shear resistance and the difference between actual shear force and the shear resistance ( $v_{Sz}$ - $V_{Rdc}$ ) can also be displayed with isolines and isosurfaces.

### 6.5.6. Column Reinforcement

Design Codes

The reinforced column check can be performed based on the following design codes:

Eurocode 2:	EN	1992-1-1:2004
DIN:	DIN	1045-1:2001-07
SIA:	SIA	262:2003

Commands for editing are the same as in the main window. **See...** 2.5 Using the Cursor, the Keyboard, the Mouse.

On **Reinforcement bars** tab the cross-section can be choosen, material parameters of the concrete column and the rebars, buckling lengths of the column can be set and rebars can be placed.

After clicking the Column Check tab N-M strength interaction diagrams are calculated.





Opens a new cross-section or reinforcement.



Only cross-sections with graphics data can be opened.

*ve* Saves the reinforcement under a name for further use.



Save diagram to the Drawings Library.

#### Define Reinforcement

Parameters

The following icons are available on the Define Reinforcement menu:

Lets you specify the parameters for calculation of the load-moment strength interaction diagram.

The unfavorable eccentricity increments determined based on the buckling parameters are displayed in the internal force check table.

Parameters	×
Buckling parameters	Materials
×	Concrete
	C25/30
L .	$\varphi_{eff} = 2.000$
*	<u>R</u> ebar steel
Calculate eccentricity increment in z direction	8500A
β _{yy} = 1.000	
Calculate eccentricity increment in y direction	
β ₂₂ = 1.000	
L (m) = [4,000	Stirrup distance
L [m] =  4.000	s _w [mm] = 200
└── Use this rebar steel by default	
	OK Cancel

### **Reinforcement Bars**

To a point Covering

Generates a reinforcement bar with a specified diameter to the location of the cursor. If the cursor is on a corner or on the contour line the reinforcement will be generated taking into account the concrete cover.



×

Inserts evenly N+1 new rebars between two selected points.



Inserts evenly N+1 new rebars between a selected starting point and an end-point of a circular arch.

*Diameter* Lets you define or modify the diameter of a rebar.

Lets you define or modify the concrete covering.

To modify, select the rebars and than the enter the diameter or select a value from the list.

Covering

Concr. C. [mm] = 35,0 💌

Ø [mm] = 25 💌

*The second transform of the concrete cover is the distance from the extreme fiber to the rebar!* 

Modifying the geometry of the rebars:

- 1. Move the cursor over the centroid of the rebar.
- 2. Use the left  ${}^{\circ}$  button (keep depressed) to move the rebar to its new location, or, enter its new coordinates numerically in the coordinate window.



The division number which defines the number of rebars as N+1.

Translate



Creates new rebars by copying existing ones by translation.



Creates new rebars by copying existing ones by rotation.



*Mirror* Creates new rebars by mirroring existing ones.



Modifying the geometry of the rebars:

- 1. Move the cursor over the centroid of the rebar.
- 2. Use the left button (keep depressed) to move the rebar to its new location, or, enter its new coordinates numerically in the coordinate window.

Column Check

Calculates the interaction diagram based on the cross-section properties and reinforcement parameters and determines the eccentricity increments for the forces in the selected columns (or any  $N_{xr} M_{yar} M_{zar} M_{yfr} M_{zf}$  values) based on the given buckling parameters and according to the requirements of the current design code.

Calculates  $N_{xd}$ ,  $M_{yd}$ ,  $M_{zd}$  design forces using the eccentricity increments and checks if these points are within the interaction diagram.

The display of the diagram can be set in the Display Parameters window.

Allows setting the display modes for the interaction diagram.

Display Parameters	×
Display Mode	Axial Forces
C Cross-section	✓ -5963.495
• N-M Surface	-5900.000
C N Mu Diaman	-5800.000
C N- <u>W</u> y Diagram	-5700.000
C N-Mz Diagram	□ -5600.000 □ -5600.000
C My-Mz Diagram (N=const.)	□ -5500.000 □ 5400.000
	-5400.000
N [KN] = 0	I -5200.000
C Critical Eco. Curves	-5100.000
Cilica Ecc. carves	-5000.000
d abels	-4900.000
	-4800.000
Critical Ecc. Curves	-4700.000
Axial Forces	-4600.000
Internal Forces	-4500.000
Graphic symbols	
	4300.000
	-4200.000
••• «1»	-4000.000
Axial Forces	-3900.000
Cross-section	-3800.000
Dispetave Conveligators	-3700.000
	✓ -3600.000
Corner Coordinates	-3500.000
Reinforcement	
	OK Cancel
I Netresh All	

 $\mathcal{G}$  Blue color shows that the  $N_{xd}$ - $M_{yd}$ - $M_{zd}$  values are within the interaction diagram. Red color shows that  $N_{xd}$ - $M_{yd}$ - $M_{zd}$  values are ouside the interaction diagram.

The normal forces for these points are always displayed.

Select display mode by clicking a radio button in the Display Mode group box. It has the same effect as selecting it from the dropdown list.

Select axial force values to use when drawing the 3D interaction diagram (N-M Surface) from the check list.

In the Labels group you can turn on and off axial force labeling, the display of graphic symbols for internal forces of selected columns in the N-My-Mz space and display options for the cross-section display mode.





*N-M surface* Displays the  $N_x$ - $M_y$ - $M_z$  strength interaction 3D diagram.

*N-M diagram* Displays the  $N_x$ - $M_y$ , or  $N_x$ - $M_z$  load-moment strength interaction diagram.



This display mode can be used with cross-sections that are symmetric. You can display the design values of the internal forces, by enabling the *Write Values to* check-box.

The design values of the internal forces are displayed as follows:

Gev Blue rectangle: the design value  $N_{xd}$ - $M_{yd}$ - $M_{zd}$  is under the interaction surface. x red cross: the design value  $N_{xd}$ - $M_{yd}$ - $M_{zd}$  is above the interaction surface.



*N-Mz diagram* Displays the  $M_x$ - $M_y$  interaction diagram at a given N value.

Displays the load eccentricity limit curves based on the  $\frac{M_{yRi}}{N_i}$  or  $\frac{M_{zRi}}{N_i}$ .

Blue rectangle: the design value  $N_{xd}$ - $M_{yd}$ - $M_{zd}$  is inside the load eccentricity limit curve. x red cross: the design value  $N_{xd}$ - $M_{yd}$ - $M_{zd}$  is outside the load eccentricity limit curve. Internal forces

The Column Internal Force Check table contains the maximum normal forces and moments at the top and bottom end of the selected columns and different eccentricity values.

Additional columns displaying  $M_{yHmin}$ ,  $M_{yHmax}$ ,  $M_{zHmin}$ ,  $M_{zHmax}$  moment resistance maximums at the given  $N_x$  are also available.

🔏 Colu	mn Internal I	Force Check	(														- 🗆 ×
<u>File E</u> d	it F <u>o</u> rmat <u>H</u>	elp															
+ × 🖻 🛍 🗐 🖾 📲																	
Colur	nn Interna	l Force C	heck [10	fi25, Lin	ear, ST1	]											
	Buckling parameters	N× [kN]	My _b [kNm]	Mz _b [kNm]	My _t [kNm]	Mz _t [kNm]	e ₀ b _v [mm]	e ₀ b _z [mm]	e ₀ t _v [mm]	e ₀ t _z [mm]	ee _v [mm]	ee _z [mm]	ei _v [mm]	ei _z [mm]	e _{2v} [m]	e _{2z} [m]	Passed
	β _{vv} = 1																
	β ₂₂ = 1																
	L = 4.000 m																
1		-3600.000	-72.798	-83.549	145.600	167.100	-23.2	20.2	46.4	-40.4	18.6	-16.2	10.0	10.0	20.5	14.2	yes
-		-3600.000	54.300	75.600	121.500	-96.600	21.0	-15.1	-26.8	-33.8	-10.7	-26.3	10.0	10.0	20.5	14.2	yes
-		-3600.000	-81.400	-16.300	167.200	49.600	-4.5	22.6	13.8	-46.4	6.5	-18.8	10.0	10.0	20.5	14.2	yes
Editina D	Design axial for	ce															
2	-																ок



On  $N-M_R$  strength interaction diagrams and on load eccentricity limit curves points represent these design loads. Custom force and moment values can also be entered into the table. These points will be displayed in the  $N-M_R$  strength interaction diagrams and in the load eccentricity limit curves. Signs of the foces and moments are determined according to the picture.

Rebars thinner than 1/12 of the stirrup distance will be ignored for compression.



#### 6.5.6.1. Check of reinforced columns based on Eurocode 2

The design moments in bending directions are  $M_d = N_d \cdot e_d$ 

where  $N_d$  is the normal force in the column and  $e_d = e_e + e_i + e_2$  is the standard eccentricity in the given bending direction.

 $e_0 = M_1/N_d$  initial eccentricity calculated from the first order force and moment. If moments at the top and bottom end of the column are different, a substitute initial

eccentricity will be determined:

$$e_e = \max \left\{ \begin{array}{c} 0.6e_a + 0.4e_b \\ 0.4e_a \end{array} \right\} \text{ and } |e_a| \ge |e_b|,$$

where  $e_a$  and  $e_b$  are the initial eccentricities at the ends of the column.

*e_i* : increment due to inaccuracies (imperfection)

$$e_i = \alpha_h \Theta_0 \frac{l_0}{2}$$
, where  $l_0$  is the buckling length.  
 $\alpha_h = \frac{2}{\sqrt{l}}$  and  $2/3 \le \alpha_h \le 1$ , where *l* is the mesh length

 $e_2$ : second order increment of the eccentricity.

$$e_{2} = \frac{1}{r} \frac{l_{0}^{2}}{\pi^{2}}, \text{ where } \frac{1}{r} = K_{r} K_{\varphi} \frac{f_{yd}}{E_{s} \cdot 0.45 \cdot d'}$$
$$K_{r} = \min \left\{ \frac{N'_{u} - N_{Ed}}{N'_{u} - N_{bal}}; \quad 1.0 \right\},$$
$$K_{\varphi} = \max \left\{ 1 + \beta \varphi_{ef}; \quad 1.0 \right\}$$
$$\beta = 0.35 + \frac{f_{ck}}{200} - \frac{\lambda}{150} \quad (f_{ck} \text{ in N/mm}^{2})$$

 $d' = (h/2) + i_s$ , where i_s is the radius of inertia of the rebars

Increments of eccentricities are determined in both bending planes and checks the following design situations:

$$M_{dy} = N_d * (ee_z \pm (e_{iz} + e_{2z}))$$
  
$$M_{dz} = -N_d * (ee_y \pm (e_{iy} + e_{2y}))$$

At the bottom and top end of the column:

$$\begin{split} M_{dy} &= N_d * (e_{0az} \pm e_{iz}) \\ M_{dz} &= -N_d * (e_{0ay} \pm e_{iy}) \\ M_{dy} &= N_d * (e_{0bz} \pm e_{iz}) \\ M_{dz} &= -N_d * (e_{0by} \pm e_{iy}) \end{split}$$

AxisVM checks whether the calculated design loads ( $M_{dy}$ ,  $M_{dz}$ ,  $N_d$ ) are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the column with the given cross-section and reinforcement fails.

 $e_{0ay}$ ,  $e_{0az}$ , and  $e_{0by}$ ,  $e_{0bz}$  are the initial eccentricities at the bottom and top end of the column.

The calculation takes the following assumptions:  $\sigma_{\epsilon}$  diagrams:



### 6.5.6.2. Check of reinforced columns based on DIN1045-1

Design moments in bending directions are  $M_d = N_d \cdot e_d$ 

where  $N_d$  is the normal force in the column and  $e_d = e_0 + e_a + e_2$  is the critical eccentricity in the given bending direction.

 $e_0 = M_{dl}/N_d$  initial eccentricity calculated from the first order force and moment.

If moments at the top and bottom end of the column are different, a substitute initial eccentricity will be determined:

$$e_e = \max \left\{ \begin{array}{c} 0.6e_a + 0.4e_b \\ 0.4e_a \end{array} \right\} \text{ and } |e_a| \ge |e_b|,$$

where  $e_a$  and  $e_b$  are the initial eccentricities at the ends of the column.

 $e_a$ : increment due to inaccuracies (imperfection)

$$e_{a} = \alpha_{a1} \frac{l_{0}}{2} \text{, where } l_{0} \text{ is the buckling length.}$$
$$\alpha_{a1} = \frac{1}{100\sqrt{l}} \leq \frac{1}{200} \text{, where } l \text{ is the mesh length.}$$
$$\lambda_{\max} = \max\left\{25; \frac{16}{\sqrt{N_{d} / A_{c} f_{cd}}}\right\}$$

If  $\lambda_{max} \ge \lambda$  second order increment of eccentricity has to be taken into account, where  $\lambda$  is the column slimness calculated from the concrete cross-section.

 $e_2$ : second order increment of the eccentricity.

$$\begin{split} e_2 &= K_1 \cdot \frac{1}{r} \cdot \frac{l_0^2}{10} , \text{ where } \frac{1}{r} = K_2 \frac{2 \cdot f_{yd}}{E_s \cdot 0.9 \cdot d} , \\ K_1 &= \min \left\{ \frac{\lambda}{10} - 2.5; \quad 1.0 \right\} , \qquad \qquad K_2 = \frac{N_{ud} - N_d}{N_{ud} - N_{bal}} \leq 1.0 , \end{split}$$

*d* is the effective height of the cross-section

Increments of eccentricities are determined in both bending planes and checks the following design situations:

$$M_{dy} = N_d * (e_{0z} \pm (e_{az} + e_{2z}))$$
  
$$M_{dz} = -N_d * (e_{0y} \pm (e_{ay} + e_{2y}))$$

At the bottom and top end of the column:

$$\begin{split} M_{dy} &= N_d * (e_{0az} \pm e_{az}) \\ M_{dz} &= -N_d * (e_{0ay} \pm e_{ay}) \\ M_{dy} &= N_d * (e_{0bz} \pm e_{az}) \\ M_{dz} &= -N_d * (e_{0by} \pm e_{ay}) \end{split}$$

AxisVM checks whether the calculated design loads ( $M_{dy}$ ,  $M_{dz}$ ,  $N_d$ ) are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the column with the given cross-section and reinforcement fails.

 $e_{0ay}$ ,  $e_{0az}$ , and  $e_{0by}$ ,  $e_{0bz}$  are the initial eccentricities at the bottom and top end of the column.

The calculation takes the following assumptions:  $\sigma_{\epsilon}$  diagrams:



### 6.5.6.3. Check of reinforced columns based on SIA 262

Design moments in bending directions are  $M_d = N_d \cdot e_d$ 

where  $N_d$  is the normal force in the column and  $e_d = e_{0d} + e_{1d} + e_{2d}$  is the critical eccentricity in the given bending direction.

 $e_{0d}$ : increment due to inaccuracies (imperfection)

$$e_{0d} = \max\left\{\alpha_i \frac{l_{cr}}{2}; \frac{d}{30}\right\}, \text{ where } \frac{1}{200} \ge \alpha_i = \frac{0.01}{\sqrt{l}} \ge \frac{1}{300},$$

 $l_{cr}$  is the buckling length, *l* is the actual length, *d* is the effective height of the cross-section.

 $e_{1d} = M_{dl}/N_d$  initial eccentricity calculated from the first order force and moment. If moments at the top and bottom end of the column are different, a substitute initial eccentricity will be determined:

$$e_e = \max \begin{cases} 0.6e_a + 0.4e_b \\ 0.4e_a \end{cases} \text{ and } |e_a| \ge |e_b|,$$

where  $e_a$  and  $e_b$  are the initial eccentricities at the ends of the column.

 $e_2$ : second order increment of the eccentricity.

$$e_{2d} = \chi_d \frac{l_{cr}^2}{\pi^2}$$
, where  $\chi_d = \frac{2f_{sd}}{E_s(d-d')}$ 

Increments of eccentricities are determined in both bending planes and checks the following design situations:

$$M_{dy} = N_d^* (e_{1z} \pm (e_{0z} + e_{2z}))$$
  
$$M_{dz} = -N_d^* (e_{1y} \pm (e_{0y} + e_{2y}))$$

At the bottom and top end of the column:

 $M_{dy} = N_d * (e_{az} \pm e_{0z})$   $M_{dz} = -N_d * (e_{ay} \pm e_{0y})$   $M_{dy} = N_d * (e_{bz} \pm e_{0z})$  $M_{dz} = -N_d * (e_{by} \pm e_{0y})$ 

AxisVM checks whether the calculated design loads ( $M_{dy}$ ,  $M_{dz}$ ,  $N_d$ ) are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the column with the given cross-section and reinforcement fails.

 $e_{ay}$ ,  $e_{az}$  and  $e_{by}$ ,  $e_{bz}$  are the initial eccentricities at the bottom and top end of the column.

The calculation takes the following assumptions:  $\sigma_{,\epsilon}$  diagrams:



Longitudinal rebars will not be taken into account for compression if any of the following criteria is met (*s* is the stirrup distance):

 $\emptyset < 8$   $s > 15 \emptyset$   $s > a_{min}$ s > 300 mm

### 6.5.7. Beam reinforcement design

Design Codes

 Eurocode 2:
 EN
 1992-1-1:2004

 DIN:
 DIN 1045-1:2001-07

 SIA:
 SIA 262:2003

The beams are structural elements, with one dimension (the length) significantly greater than the dimensions of the cross section, loaded in bending and shear, and axial force is zero or of a small, negligible value.

The beam reinforcement design module can be applied to beam structural elements modeled by beam or rib finite elements, that have the same material and constant or variable rectangular or T cross sections, assuming that the load is applied in the symmetry plane of the cross section.

The computed longitudinal top and bottom reinforcement are of the same steel grade, while the stirrups could have steel grade different from the longitudinal ones.

Variable cross-section



The change in shear force due to variable crosssection is taken into account.

Where sign of the moment does not change a simple rule can be applied: if section height changes the same way as the moment along the line shear capacity increases otherwise it decreases.

Shear force is modified by  $\Delta V = 2A_s f_{yd} \sin \alpha$ , where  $A_s$  is the longitudinal tension reinforcement area,  $\alpha$  is the angle between the extreme fiber and the centerline. Longitudinal reinforcement is assumed to be parallel with the extreme fiber.

Steps of design

### The design is performed in two steps:

- 1. Design of longitudinal reinforcement for moments about y, or z axis (M_y, or M_z).
- 2. Determination of spacing of vertical stirrups considering shear forces about y or z axis (Vy or Vz) and the twisting moment (Tx).

The axial force is not taken into account. If the axial force cannot be neglected, the use of the Column Design module is recommended.

Bending and shear/twisting is analyzed separately, however the longitudinal tensile reinforcement is taken into account in the determination of the shear capacity.

The increase in the tension in the longitudinal rebars due to the shear cracks are accounted by shifting the moment.

*AxisVM performs only design procedures listed in this section.* 

Any other requirement shall be fulfilled by the user, following the requirements of the design codes, and corresponding other regulations.

The Beam Design module does not check the effect of biaxial bending, lateral torsional buckling transversal stresses due to direct application of point loads, or any interaction involving these.

The module cannot be applied to the design of short cantilevers.

#### User's Manual

Define of size of support



Clicking on the support the following dialog window is displayed:



Lets you specify the segments each side of the support that will be not included in the calculations. The internal forces are linearly interpolated within the segments.

The diagram below shows the moment/shear force reduction above supports:







Selection of the z-x or y-x plane of the internal forces used for design.



Stirrup Stirrup legs: lets you set the number of stirrup legs subject to shear.

¢, Rebar positions (ub, ut): distance between the centroid of rebar and the corresponding extreme fiber of the concrete.



- $u_b$ : the distance of the center of the bottom rebar from the edge of the cross section.
- $u_t$ : the distance of the center of the top rebar from the edge of the cross section.



splay			×	
Diagrams			_	
	Display	Labeling Extremes		Diagrams off/c
Model	V			
M _v	V	-	-	Labeling off/o
A _{sb} /A _{st}	Г	Γ		
V _z				
Stirrup distance	Г	Г		
M _x				
A _{scs}	Г	Γ		
	ОК	Cancel		

As results provided are the longitudinal reinforcement from bending, maximum stirrup spacing and the longitudinal reinforcement from torsion diagrams.

300

€

Longitudinal reinforcement from bending On the longitudinal reinforcement diagram the tension reinforcement is displayed in blue, the compression reinforcement in red, and the minimal reinforcement according to the design code in gray.



# Longitudinal reinforcement from torsion

The longitudinal reinforcement diagram is displayed in purple.

The longitudinal reinforcement from torsion should be placed uniformly around the crosssection contour.



*Stirrup spacing* The allowable maximum stirrup spacing is displayed in black, the calculated spacing in blue, and the minimal spacing according to the design code in gray.



### 6.5.7.1. Beam Reinforcement Design based on Eurocode2

Syml	pols, material properties, partial factors
f _{cd}	design value of the compressive strength of the concrete
f _{ctd}	design value of the yield strength of the concrete
α	= 0.85; a coefficient, that takes the sustained load and other unfavorable effects into
	account
$\gamma_{\rm c}$	= 1.5; partial factor of the concrete
f _{yd}	design value of flow limit of rebar steel
$\epsilon_{su}$	limiting strain of rebar steel
Es	(=200 kN/mm ² ); Young modulus of rebar steel
$\gamma_{\rm s}$	= 1.15; partial factor of the steel

### Shear & torsion reinforcement design of stirrups

The design is based on the following values of design shear resistance:

 $V_{Rd,c}$  Design shear resistance of the cross-section without shear reinforcement.

- $V_{Rd,max}$  Maximum shear force that can be transmitted without the failure of the inclined compression bars.
- V_{Rd,s} Design shear resistance of the cross-section with shear reinforcement.
- T_{Rdc} Design torsional resistance of the cross-section without shear reinforcement.
- $T_{Rd,max}$  Maximum torsional moment that can be transmitted without the failure of the inclined compression bars.

AxisVM calculates the shear & torsion reinforcement assuming that shear crack inclination angle is 45°. The relation between the capacity of inclined compression concrete bars and the design values is checked.

$$\frac{V_{Ed}}{V_{Rd,\max}} + \frac{T_{Ed}}{T_{Rd,\max}} \le 1 \text{, where}$$

$$V_{Rd,\max} = \frac{\alpha_{cw} b_w z v_1 f_{cd}}{\cot \Theta + \tan \Theta} \text{ and } T_{Rd,\max} = 2v\alpha_{cw} f_{cd} A_k t_{ef,i} \sin \Theta \cos \Theta$$

If the cross-section does not fail it is checked if shear & torsion reinforcement is required according to the formula

$$\frac{V_{Ed}}{V_{Rd,c}} + \frac{T_{Ed}}{T_{Rd,c}} \le 1 \text{ , where}$$

$$V_{Rd,c} = \left[ C_{Rd,c} k \left( 100 \rho_l f_{ck} \right)^{\frac{1}{3}} + k_1 \sigma_{cp} \right] b_w d \text{ and } T_{Rd,c} = 2 f_{ctd} t_{ef,i} A_k$$

If sheari & torsion reinforcement is required,

$$\frac{\sum A_{sl} f_{yd}}{u_k} = \frac{T_{Ed}}{2A_k} \cot \Theta \text{, therefore } A_{sl} = \frac{T_{Ed} u_k}{2A_k f_{yd}} \tan \Theta$$

Spacing of shear & torsion stirrups is calculated from these formulas:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot \Theta \text{ and } V_{Rd,s} \ge V_{Ed} + V_{Ed,i}.$$

$$s = \frac{A_{sw}}{V_{Ed} + V_{Ed,i}} z f_{ywd} \cot \Theta$$

Using the variable angle truss method, significant saving of shear reinforcement can be achieved if the compressed concrete beams have extra resistance, i.e:

$$\frac{V_{Ed}}{V_{Rd,\max}} + \frac{T_{Ed}}{T_{Rd,\max}} << 1$$

By changing the shear crack inclination angle the compressed concrete beams gets more load while shear reinforcement gets less. The actual saving depends on the design rules.

If the user chooses the variable angle truss method, AxisVM determines the direction of the shear crack between 21,8° (ctg $\Theta$ =2,5) and 45° (ctg $\Theta$ =1) before the calculation of the reinforcement so that the exploitation of the inclined concrete compression beams reach its maximum (at most 100%). The shear crack inclination angle is increased in small steps to meet the requirement

$$\frac{\overline{V_{Ed}}}{V_{Rd,\max}} + \frac{T_{Ed}}{T_{Rd,\max}} \le 1$$

The cross-section fails if critical shear force is higher than the shear resistance of the compressed concrete beams, i.e.:

$$\frac{V_{Ed}}{V_{Rd,\max}} + \frac{T_{Ed}}{T_{Rd,\max}} > 1$$

Design rules applied in calculation:

On the basis of equation 9.2.2 (9.5N)  $\rho_{w,\min} = 0.08 \sqrt{f_{ck}} / f_{yk}$  and of equation 9.2.2 (9.4)  $\rho_w = A_{sw} / sb_w$  so the ratio of shear reinforcement is  $s_{\max 1} = A_{sw} / \rho_{w,\min} b_w$ .

9.2.2 (9.6N) states that:  $s_{max2} = 0.75d$ .

### Longitudinal Beam Reinforcement

AxisVM calculates longitudinal reinforcement according to this figure:



Limit stress is assumed in the rebars. The depth of the compressed zone will be less than

 $x_0 = d \cdot \frac{\varepsilon_{cu} - \varepsilon_{c1}}{\varepsilon_{s1} - \varepsilon_{cu}} \,.$ 

If calculation results in a greater depth than  $x_0$ , a compression reinforcement is applied, but the sum of the area of reinforcement on the compression and on the tension side cannot exceed 4% of the concrete cross-section area.

The required top and bottom reinforcement along the beam and the moment diagram shift is calculated for each load case.

Due to inclined cracks tension reinforcement is designed for a force greater than calculated from M/z.

This is taken into account by different design codes by shifting the moment diagram.

Minimum ( $M_{min} \le 0$ ) and maximum ( $M_{max} \ge 0$ ) values of the moment diagram and the corresponding reinforcement on tension and compression side is determined. Tension reinforcement is displayed in blue, compression reinforcement in red, the minimal tension reinforcement required by the design code appears in grey.

Compression reinforcement has to be considered even if tension reinforcement is the critical one, as longitudinal rebars thinner than 1/12 of the stirrup distance has to be ignored when determining the compression rebar diameter or the stirrup spacing.

### 6.5.7.2. Beam Reinforcement Design based on DIN 1045-1

#### DIN 1045-1

Sym	bols, material properties, partial factors
$f_{cd}$	design value of the compressive strength of the concrete
f _{ctm}	mean value of the tensile strength of the concrete
α	= 0.85; a coefficient, that takes the sustained load and other unfavorable effects into
	account
$\gamma_{\rm c}$	= 1.5; partial factor of the concrete
f _{yd}	design value of flow limit of rebar steel
$\epsilon_{su}$	limiting strain of rebar steel
Es	(=200 kN/mm ² ); Young modulus of rebar steel
$\gamma_{\rm s}$	= 1.15; partial factor of the steel

#### Shear & torsion reinforcement design of stirrups

The design is based on the following three values of design shear resistance:

- $V_{Rd,ct}$  Design shear resistance of the cross-section without shear reinforcement.
- V_{Rd,max} Maximum shear force that can be transmitted without the failure of the inclined compression bars.
- V_{Rd,sy} Design shear resistance of the cross-section with shear reinforcement.

No shear reinforcement is required if:  $V_{Ed} \le V_{Rd,ct}$  DIN 1045-1 10.3.1 (2)

The cross-section does not fail if: 
$$V_{Ed}$$

If  $V_{Ed} > V_{Rd,ctr}$  shear reinforcement should be applied DIN 1045-1 10.3.1 (3)

Stirrup spacing is determined to meet the requirement  $V_{Ed} \leq V_{Rd,sy}$ .

For cross sections with shear reinforcement we can choose between the *regular* method (45th cracking) and *Variable Angle Truss* (VAT) method.

 $\leq V_{Rd,max}$ .

If the assumed compression trusses have reserve ( $V_{Rd,max} > V_{Ed}$ ) according to the regular method, the VAT method will lead to considerable savings in shear reinforcement.

By changing the shear crack inclination angle the compressed concrete beams gets more load while shear reinforcement gets less.

The program is calculating the value of  $\operatorname{ctg} \phi$ :

In case of regular concrete: In case of light concrete:

$$\cot \Theta = \frac{1,2-1,4 \cdot \frac{\sigma_{cd}}{f_{cd}} A_{sw}}{1 - \frac{V_{Rd,c}}{V_{Ed}}}$$
$$0.58 \le \cot \Theta \le 3.0$$
$$0.58 \le \cot \Theta \le 2.0$$

DIN 1045-1 10.3.4 (3)

The regular method assumes the angle of shear cracks to be  $45^\circ$ , so  $\cot \Theta = 1$ .

$$V_{Rd,sy} = \frac{A_{sw}}{s_w} \cdot f_{yd} \cdot z \cdot \cot \Theta \qquad \qquad \text{DIN 1045-1} \quad 10.3.4 \text{ (7)}$$

is the shear resistance due to the shear reinforcement.

If torsion is considerable, AxisVM also checks the following condition:

$$\left[\frac{T_{Ed}}{T_{Rd,\max}}\right]^2 + \left[\frac{V_{Ed}}{V_{Rd,\max}}\right]^2 \le 1$$
 DIN 1045-1 10.4.2 (5)

No calculated shear & torsion reinforcement has to be applied if

$$T_{Ed} \leq \frac{V_{Ed} b_w}{4.5}$$
 and  $V_{Ed} \left[ 1 + \frac{4.5 T_{Ed}}{V_{Ed} b_w} \right] \leq V_{Rd,ct}$ 

DIN 1045-1 10.4.1 (6)

### Stirrup reinforcement from twisting moment

Resistant twisting moment on the basis of the failure of the compressed concrete bar:

$$T_{Rd,sy} = 2\frac{A_{sw}}{s_w} f_{yd} A_k \cot \Theta$$

The stirrup distance:

$$s_w = 2 \frac{A_{sw}}{T_{Ed}} f_{yd} A_k \cot \Theta$$

Longitudinal reinforcement is calculated from twisting moment

$$T_{Rd,sy} = 2 \frac{A_{sl}}{u_k} f_{yd} A_k \tan \Theta$$
, so  $A_{sl} = \frac{T_{Ed} u_k}{2f_{yd} A_k \tan \Theta}$ , which should be placed evenly along

the cross-section contour.

The actual stirrup distance is taken into account form the summary of the torsion stirrup distance and the shear stirrup distance:

$$s_w = \frac{1}{\frac{1}{\frac{1}{s_{w,V}} + \frac{1}{s_{w,T}}}}$$

### Longitudinal Beam Reinforcement based on DIN1045-1



The limit stress is developing in the reinforcement. The depth of the compressive concrete  $\varepsilon_{c2\mu} - \varepsilon_{c2}$  where  $\rho_{c2\mu} = f_{c2\mu} - \varepsilon_{c2}$ 

zone will exceed  $x_0 = d \cdot \frac{\varepsilon_{c2u} - \varepsilon_{c2}}{\varepsilon_{s1} - \varepsilon_{c2u}}$ , where  $\varepsilon_{s1} = f_{yd} / E_s$ .

If from the calculation a greater height than x0 is obtained, compressive steel cross section is applied, but the sum of the compressive and tensile steel cross section cannot exceed 8% of the concrete cross section.

The software calculates for each load case and cross section the lower and upper reinforcement, and the value of the moment shifting.

Due to oblique cracks the tension reinforcement is designed for a tension force greater than calculated from M/z.

This is taken into account by design codes by shifting the moment diagram (DIN 1045-1 13.2.2)

Minimum ( $M_{min} \le 0$ ) and maximum ( $M_{max} \ge 0$ ) values of the moment diagram, and the corresponding tension and compression reinforcements are determined. On the reinforcement diagram the tension reinforcement is displayed in blue, the compressive in red, and the minimal tension reinforcement according to the design code in grey.

The compression reinforcement is necessary even if the tension reinforcement is the critical, because at the determination of the compression reinforcement diameters and stirrup spacing is taken into account that only the 1/12 of the stirrup spacing or longitudinal rebars with greater diameter are included.

### Construction rules considered in the program

Ratio of stirrup reinforcement:  $\rho_w = \frac{A_{SW}}{c} b_w$ 

From the above expression:  $s_{\max 1} = \frac{A_{sw}}{\rho_w b_w}$  where,  $\rho_w = 0.16 \frac{f_{ctm}}{f_{yk}}$ 

Minimal value of  $\rho_w$  is may calculated from Table 29. in DIN 1045-1 13.1.3 The  $s_{max}$  stirrup distance is taking into account Table 31. in DIN 1045-1 13.2.1 The maximum stirrup distance from twisting moments is  $u_k/8$ .

#### Warnings, error messages

The software sends warning message and does not draw any reinforcement diagram in the following cases:

Message

The cross section is not acceptable for shear/torsion

Event

Any of the following conditions is not satisfied:

$$\mathbf{V}_{\text{Rd,max}} \ge \mathbf{V}_{\text{Ed}} \text{ or } \left[ \frac{T_{Ed}}{T_{Rd,\text{max}}} \right]^2 + \left[ \frac{V_{Ed}}{V_{Rd,\text{max}}} \right]^2 \le 1$$

- 2

-2

Solution Increase the cross section of the concrete, or/and the concrete grade.

Message The cross section is not acceptable for bending (As + As2 > 0.08 * Ac)

- Event The cross sectional area of the longitudinal reinforcement is greater than 8% of the concrete cross section
- Solution Increase the cross section of the concrete, or/and the concrete grade, or/and the steel grade.

#### 6.5.7.3. Beam Reinforcement Design based on SIA 262:2003

### SIA 262:2003

Syml	pols, material properties, partial factors
f _{cd}	design value of the compressive strength of the concrete
f _{ct}	design value of the yield strength of the concrete
$\gamma_{c}$	= 1.5; partial factor of the concrete
f _{yd}	design value of flow limit of rebar steel
$\epsilon_{su}$	limiting strain of rebar steel
Es	(=200 kN/mm ² ); Young modulus of rebar steel
$\gamma_{\rm s}$	= 1.15; partial factor of the steel
k _c	= 0.6; reduction factor for compressive strength of the concrete in a cracked zone

### Shear & torsion reinforcement design of stirrups

The shear reinforcement design is based on three values of the shear resistance:

- V_{Rd} The shear resistance of the cross section without shear reinforcement.
- $V_{Rd,c}$  The maximum shear force that can be transmitted without the failure of the assumed compression bars.
- V_{Rd,s} The shear resistance of the cross section with the shear reinforcement.

No shear reinforcement is required if:  $V_d \leq V_{Rd}$ 

$$V_{Rd} = k_d \cdot \tau_{cd} \cdot d \cdot b_w$$
,  $k_d = \frac{1}{1 + k_v \cdot d}$   $d \text{ in } [m]$ ,  $k_v = 2,5$ 

The conrete cross-section does not fail if  $V_{Rd,c} \ge V_d$ 

$$V_{Rd,c} = b_w z k_c f_{cd} \sin \alpha \cos \alpha$$

If  $V_d > V_{Rd,c'}$  shear reinforcement should be designed. The stirrup distance is determined from the expression

$$V_{Rd,s} = \frac{A_{sw}}{s} \cdot z \cdot f_{sd} \cdot \cot \alpha$$
$$s = \frac{A_{sw}}{V_d} \cdot z \cdot f_{sd} \cdot \cot \alpha$$

Stirrup spacing is

Longitudinal force from shear:  $F_{td} = V_{Rd} \cdot \cot \alpha$ 

Additional longitudinal reinforcement:

which should be placed  $\frac{1}{2}$  to the tension zone,  $\frac{1}{2}$  to the compression zone.

Shear force from torsion:

Shear force in a vertical fiber:

Shear force in the horizontal fiber:

$$\begin{split} V_{d,i} &= \frac{T_d}{2 \cdot A_k} \cdot z_i \\ V_{d,h} &= \frac{T_d}{2 \cdot z_b} \\ V_{d,b} &= \frac{T_d}{2 \cdot z_h} \end{split}$$

 $\Delta A_{sl} = \frac{V_{Rd} \cdot \cot \alpha}{c}$ 

fsd

The program checks the following expression  $\frac{V_d}{V_{Rd,c}} + \frac{V_{di}}{V_{Rd,ci}} \le 1$ 

where  $V_{Rd,ci} = t_k \cdot z_h \cdot k_c \cdot f_{cd} \cdot \sin \alpha \cdot \cos \alpha$ 

Stirrup distance from torsion:  $s = A_{sw} \cdot \frac{2 \cdot z_h \cdot z_b}{T_d} \cdot f_{sd} \cdot \cot \alpha$ 

Longitudinal reinforcement from torsion:

$$A_{slT} = \frac{\sum V_{d,i} \cdot \cot \alpha}{f_{sd}} = \frac{\frac{T_d}{z_h \cdot z_b} \cdot (z_h + z_b) \cdot \cot \alpha}{f_{sd}}$$

which should be placed evenly along the cross-section contour.



The actual stirrup distance is taken into account form the summary of the torsion stirrup distance and the shear stirrup distance:

$$w = \frac{1}{\frac{1}{\frac{1}{s_{w,V}} + \frac{1}{s_{w,T}}}}$$

S

#### Beam Longitudinal Reinforcement based on SIA 262:2003

σ,ε diagrams



The limit stress is developing in the reinforcement. The depth of the compressive concrete

zone will exceed 
$$x_0 = d \cdot \frac{\varepsilon_{c2u} - \varepsilon_{c2}}{\varepsilon_{s1} - \varepsilon_{c2u}}$$
, where  $\varepsilon_{s1} = f_{yd} / E_s$ 

If from the calculation a greater height than x0 is obtained, compressive steel cross section is applied, but the sum of the compressive and tensile steel cross section cannot exceed 8% of the concrete cross section.

The software calculates for each load case and cross section the lower and upper reinforcement, and the value of the moment shifting.

Due to oblique cracks the tension reinforcement is designed for a tension force greater than calculated from M/z.

This is taken into account by shifting the moment diagram.

Minimum ( $M_{min} \le 0$ ) and maximum ( $M_{max} \ge 0$ ) values of the moment diagram, and the corresponding tension and compression reinforcements are determined. On the reinforcement diagram the tension reinforcement is displayed in blue, the compressive in red, and the minimal tension reinforcement according to the design code in grey.

The compression reinforcement is necessary even if the tension reinforcement is the critical, because at the determination of the compression reinforcement diameters and stirrup spacing is taken into account that only the 1/12 of the stirrup spacing or longitudinal rebars with greater diameter are included.

#### Construction rules considered in the program

Maximum of the stirrup distance:

$$s_{\max} = \frac{A_{sw} \cdot f_{yk}}{0.2 \cdot b_w \cdot f_{ctm} \cdot \sin \alpha} \le 400 \text{ mm}$$

#### Warnings, error messages

*AxisVM sends a warning message and does not draw any reinforcement diagram in the following cases:* 

Message

#### The cross section is not acceptable for shear/torsion

Event If the efficiency of concrete cross-section greater than 1.

Solution Increase the cross section of the concrete, or/and the concrete grade.

### 6.5.8. Punching Analysis



Punching shear control perimeters are determined based on the column cross-section and the effective plate thickness. Plate edges and holes are taken into account if they are closer to the column than six times the effective plate thickness. If column cross-section is concave a convex section is used instead.

Punching analysis can be performed based on the following design codes:

Design Codes

Eurocode 2: EN 1992-1-1:2004 DIN: DIN 1045-1:2001-07

After clicking the tool button select a column or a support with stiffnesses calculated from column parameters for analysis (if a rib element is connected to the column within the plane of the plate, analysis cannot be performed).

The following parameters can be set:



#### Materials

*Concrete,* Concrete and reinforcing steel grade used in calculation. These parameters are taken from *Rebar steel* the actual model by default and can be changed here.

*Total plate* Plate thickness is taken from the actual model by default and can be changed here, if *thickness (h) By reinforcement parameter* is turned off. In the info window the minimum mushroom head thickness is displayed as H1. The minimum mushroom head without punching shear reinforcement is displayed as H2.

#### **Parameters**

Shear reinforce- ment angle	Angle between the plate and and the punching shear rebars ( $45^{\circ}$ -90°).
Radial rebar spacing	Radial rebar spacing is the difference between the radii of two neighbouring rebar circles. The <b>OK</b> button is not available until basic design criteria are met: MSZ: $t \le 0.85 h(1 + ctg\alpha)$ ; EC2: $S_r \le 0.75 d$ ; DIN: $s_w \le 0.75 d$
Distance of the first punching rebar circle	Distance of the first punching rebar circle from the convex edge of the column

$\beta$ factor (Eurocode2 and DIN)	Calculated based on Eurocode	$1+k\cdot\frac{M_{Ed}}{V}\cdot\frac{u_1}{W}$				
			Furocodo	$\frac{7Ed}{Ed}$ DIN		
	Approximate value by column position*		Eurocode	DIN		
		Internal column	1,15	1,05		
		Edge column	1,4			
		Corner column	1,5			
	Custom	user-specified value				

*For structures where the lateral stability does not depend on frame action between the slabs and the columns, and where the adjacent spans do not differ in length by more than 25%.

- *Take soil reaction* If this option is checked soil reaction within the rebar circle is considered when calculating into account the punching force. This effect increases with the radius and can reduce the size of the necessary reinforcement area. Its values per rebar circles are listed in the Punching Analysis Results dialog.
  - Loads the saved parameters of punching Loading...

After entering all parameters control perimeters will appear and the required number of punching rebars is displayed in the info window.

AxisVM calculates the effective parts of the control perimeter based on plate edges and G. holes. Continuous lines show that reinforcement is needed. AxisVM displays the required amount of reinforcement for each line. The info window shows the amount of critical punching reinforcement. When calculating the length of the critical perimeter it is assumed that rebar spacing on the perimeter is not above 2d but the fulfillment of this requirement is not checked. If this requirement is not met, the user should choose a smaller diameter or place additional rebars.

Results for the critical perimeter are calculated first (these are displayed in the Punching analysis results dialog). Then the required amount of reinforcement is determined for reinforcement circles defined in the parameters dialog. The critical perimeter is red, reinforcement circles are black. Dashed line shows the perimeter where the distance of points from the column is six times the effective plate thickness.

A thin blue line shows the perimeter where no punching reinforcement is needed.

This is also the outline of the mushroom head which can be designed with thickness H2 and without punching reinforcement.

A thick blue line shows the perimeter where the critical punching force exceeds the compressing strength of the concrete so the plate with the original thickness cannot be properly reinforced. This is the outline of the mushroom head which can be designed with thickness H1 and with punching reinforcement. Punching capacity can be increased by setting the plate thicker, using a better concrete grade or columns with bigger cross-section area.

Saves the drawing into the Drawing Library.



Loads a saved punching parameter set.

Saves the current punching parameters under a name. You can load back the saved parameters with the button *Loading*... on Punching Parameters Dialog.

Punching parameters dialog.



Inflates the plate boundary so that the entire column cross section is within the boundary.

Fits the diagram to the window.

Ē÷γ

Column local coordinates are used.





Global coordinates are used.





Turns on and off the display of rebar circles.

### 6.5.8.1. Punching analysis based on Eurocode2

The required punching reinforcement is calculated based on the following principles: The column-plate connection does not fail if the shear stress is less than or equal to the design value of the maximum punching shear resistance along the control section and the design value of the punching shear resistance of the plate with punching shear reinforcement:

$$v_{Ed} \le v_{Rd,\max}$$
 and  $v_{Ed} \le v_{Rd,cs}$ 

- $\mathbf{v}_{Ed}$  design value of the shear stress
- $v_{Rd,max}\;$  the design value of the maximum punching shear resistance along the control section
- $v_{Rd,cs} \quad$  the design value of the punching shear resistance of the plate with punching shear reinforcement

$$v_{Ed} = \beta \cdot \frac{V_{Ed}}{u_i \cdot d},$$

where  $u_i$  is the length of the control perimeter, d is the mean effective thickness of the plate.

 $\beta$  is a factor expressing additional stress due to eccentric forces:

$$\beta = 1 + k \cdot \frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_i}{W_1}$$

Eurocode assumes that the critical section is at a distance of 2d from the edge of the crosssection. The length of the critical perimeter and the static moment is calculated considering plate edges and holes of the actual geometry.

Design value of the punching resistance of the connection without punching shear reinforcement is:

$$v_{Rd,c} = C_{Rd,c} k (100 \rho_1 f_{ck})^{1/3} + k_1 \sigma_{cv} \ge (v_{\min} + k_1 \sigma_{cv})$$

If  $v_{Ed} > v_{Rd,c}$ , then the required punching reinforcement is determined along the critical perimeter

$$v_{Rd,cs} = 0.75 \cdot v_{Rd,c} + 1.5 \cdot \frac{d}{s_r} \cdot \frac{A_{sw} \cdot f_{ywd,ef}}{u_1 \cdot d} \cdot \sin \alpha \quad \text{and} \quad v_{Ed} \le v_{Rd,cs}$$

The reinforcement for each perimeter and the perimeter where no punching reinforcement is needed is calculated based on the formula:

$$v_{Ed} = \beta \cdot \frac{V_{Ed}}{u_i \cdot d} \le v_{Rd,c}$$

Info window

Under the design code, element identifier and materials the following parameters are displayed.

- plate thickness h:
- *d:* effective plate thickness
- $\alpha$ : angle between the plate and the punching reinforcement
- distance of reinforcement circles s_r:
- minimum plate thickness required with punching reinforce-H1: ment
- H2: minimum plate thickness required without punching reinforcement
- N_{Ed}: design value of the punching force

 $M_{Edx}$ ,  $M_{Edz}$ design value of the moment

- β*: calculated excentricity factor
- control perimeter at the column perimeter  $u_0$ :
- critical control perimeter at 2d **u**₁:
- $v_{Ed0}$ : shear stress along the u₀ perimeter
- $v_{Ed}$ : shear stress along the  $u_1$  perimeter
- maximum of shear stress v_{Rdmax}:
- shear stress without reinforcement v_{Rdc}:
- efficiency on the critical control perimeter  $v_{Ed}/v_{Rdmax}$ :

 $v_{Ed0}/v_{Rdmax}$ :

- efficiency on the u₀ perimeter  $v_{Ed}/v_{Rdc}$ : efficiency (tension in concrete)
  - distance between the first rebar circle and the convex column  $\mathbf{r}_1$ : edge
  - tension in the punching reinforcement f_{ywdeff}:
    - punching reinforcement area on the critical control perimeter  $A_{sw}$ :
    - N_{sr}: number of reinforcement circles

	×
Eurocode	
Beam 1 (finite element),	Node 9
C35/45	
B500A	
n[mm] =	200
d[mm] =	170
x[°] =	90
s _r [mm] =	128
H1[mm] =	200
H2[mm] =	215
Load Case : Co #3	3
$N_{ed}[kN] = -$	-371,12
M _{Edy} [kNm] =	8,13
$M_{Edz}$ [kNm] =	10,13
3* =	1,053
u _o [m] =	1,606
u ₁ [m] =	3,201
- / _{Ed0} [N/mm ² ] =	1,43
/ _{Ed} [N/mm ² ] =	0,72
$I_{Bdmax}[N/mm^2] =$	6,02
$I_{Rdc}[N/mm^2] =$	0,66
$v_{Ed} / v_{Rdmax} = 0.12$	<= 1
$v_{Ed0} / v_{Rdmax} = 0.24$	<= 1
$v_{Ed} / v_{Rdc} = 1,09 >$	1
Punching reinforcem	ent
is needed.	
[mm] =	51
	292,50
$A_{sw}[mm^2] =$	208
V _{er} =	13

#### Punching analysis based on DIN 1045-1 6.5.8.2.

The required punching reinforcement is calculated according to the following principles: The column-plate connection does not fail if the shear stress is less than or equal to the design value of the maximum punching shear resistance along the control section and the design value of the punching shear resistance of the plate with punching shear reinforcement:  $v_{sd} \leq v_{Rd}$ 

The design value of the shear stress is  $v_{sd} = \beta \cdot \frac{V_{sd}}{u \cdot d}$ , where  $\beta$  is a factor expressing addi-

tional stress due to eccentric forces.

DIN 1045-1 assumes that the critical section is at a distance of 1,5d from the edge of the cross-section.

Design value of the punching resistance of the connection without punching shear reinforcement is determined using the formula ۱

$$\begin{aligned} v_{Rd} &= f \left( v_{Rd,ct}, v_{Rd,cta}, v_{Rd,\max}, v_{Rd,sy} \right) \\ v_{Rd,ct} &= \left( 0.14 \cdot \eta_1 \cdot \kappa \cdot (100 \cdot \rho_1 \cdot f_{ck})^{1/3} + 0.12 \cdot \sigma_{cd} \right) \cdot d , \\ v_{Rd,ct,a} &= \kappa_a \cdot v_{Rd,ct} \end{aligned}$$

The design value of the maximum punching shear resistance is  $v_{Rd,max} = 1.7 \cdot v_{Rd,ct}$ 

On the first perimeter at a distance of  $r_0 = 0.5 \cdot d$  from the cross-section edge the required

amount of punching shear  $v_{Rd,sy0} = v_{Rd,c} + \frac{\kappa_s A_{sw0} \cdot f_{yd}}{u_0}$ .

Design value of the punching resistance of the connection with punching shear reinforcement is

$$v_{Rd,sy} = v_{Rd,c} + \frac{\kappa_s A_{sw} \cdot f_{yd} \cdot d}{u_i \cdot s_w}$$

If  $v_{sd} > v_{Rd,ct}$ , the required amount of punching shear reinforcement is calculated along the critical perimeter using the requirement  $v_{sd} \leq v_{Rd,sy}$ .

Info window

h: d:

α: sw:

H1:

H2:

N_{Ed}:

β:

 $u_0$ :

 $u_1$ :

 $v_{Ed}$ : V_{Rdmax}:

v_{Rdct}:

 $v_{Ed}/v_{Rdmax}$ :  $v_{Ed}/v_{Rdct}$ :

Kappa_s

 $M_{Edx}$ ,  $M_{Edz}$ 

Under the design code, element identifier and materials the following parameters are displayed. plate thickness effective plate thickness	DIN 1045-1 (Germ Beam 1 (finite element) C40/50 Bst 500 (A)	an) ), Node 9
angle between the plate and the punching reinforcement distance of reinforcement circles minimum plate thickness required with punching rein-	h[mm] = d[mm] = α[°] = s _w [mm] = H1[mm] =	200 170 90 128 200
minimum plate thickness required without punching rein- forcement	H2[mm] = Load Case : Co +	210 #3
design value of the punching force design value of the moment	$M_{Edy}[kNm] =$ $M_{Edy}[kNm] =$	8,13 10,13
excentricity factor control perimeter at the column perimeter	β* = u ₀ [m] = u ₁ [m] =	1,053 1,606 2,745
critical control perimeter at $2d$ shear stress along the $u_1$ perimeter	v _{Ed} [kN/m] = v _{Rdmax} [kN/m] = v _{Rdct} [kN/m] =	142,350 204,950 136,640
shear stress without reinforcement efficiency on the critical control perimeter	v _{Ed} / v _{Rdmax} = 0,69 v _{Ed} / v _{Rdct} = 1,04 Punching reinforcer	9 <= 1 > 1 ment
efficiency (tension in concrete)	is needed. Kappa _s = r. [mm] =	0,700
correction factor: $1 + \sqrt{\frac{d}{d}} \le 2$	$A_{sw}[mm^2] =$	393

 $N_{sr} =$ 

distance between the first rebar circle and the convex col- $\mathbf{r}_1$ : umn edge

2 |

	A _{sw} : N _{sr} :	number of reinforcement circles
		Warnings and error messages
Message		Compression force in plate is too high.
Event Solution		The applied force is so high that the concrete plate fails irrespectively of the reinforcement. The most efficient solution is to increase plate thickness.
		The critical punching area can be extended by increasing plate thickness and/or column size (reducing the design value of the specific shear force this way).
		Choose a higher grade concrete.

#### 6.5.9. **Footing design**

æ

AxisVM can determine the necessary size and reinforcement of rectangular spread foundations (with or without pedestal), and can check the footing against sliding and punching according to Eurocode7 and MSz. It determines the settlement of the foundation as well.

The size of the foundation can be entered or let AxisVM calculate it. If AxisVM calculates the Footing size size a maximum value must be specified.

> Using the soil profile and the internal forces this module determines the necessary size of the foundation in an iterative process. Then it calculates the effective area of the foundation for load cases and combinations, the design forces, moments and resistances, determines the settlement (for load cases and Service Limit State [SLS] combinations), efficiencies and the shear reinforcement if necessary. The module also checks the stability of the footing. Step sides must not be bigger than the respective side of the foundation.

The coordinate system used in footing calculations is the coordinate system of the support.

Click the *Footing design* icon and select one or more nodal supports with a vertical or slanted column. (If supports have been already selected, the dialog is displayed at the first click). Footing desgn parameters have to be specified in a dialog.

At the Footing tab select the footing type (simple plate / stepped / sloped) and set the geometry parameters and the friction coefficient between the footing and the blind concrete.



Footing

Footing design

parameters

Symmetry of footing



# Square footing

*b* is the side length, the column is concentric, value or upper limit of b must be entered

### **Rectangular footing**

bx and by are the sides,

the column is concentric,

value or upper limit of *bx* and *by* must be entered

## Single eccentric rectangular footing

the column is eccentric in x direction, concentric in y direction  $x_1$  and  $x_2$  are the distance of the column axis from the edges of the footing value or upper limit of  $x_1$ ,  $x_2$  and by must be entered



### Single eccentric rectangular footing

the column is eccentric in *y* direction, concentric in *x* direction  $y_1$  and  $y_2$  are the distance of the column axis from the edges of the footing value or upper limit of  $y_1$ ,  $y_2$  and bx must be entered

### Double eccentric rectangular footing

the column is eccentric in both directions

 $x_1$  and  $x_2$  are the distance of the column axis from the edges of the footing in x direction

 $y_1$  and  $y_2$  are the distance of the column axis from the edges of the footing in y direction

value or upper limit of  $x_1$ ,  $x_2$ ,  $y_1$ ,  $y_2$  must be entered

If the lock button beside the edit field is down (closed), the entered value is given (it is checked). If the lock icon is up (open) the entered value is the upper limit (it is determined by the program). If Check is turned on, all values will be closed and cannot be opened until Check is turned off.



### For stepped and sloped footings:

 $dx_1$  and  $dx_2$  are the distance of the edges of the step or the upper base of the frustum from the column axis in x direction.  $dy_1$  and  $dy_2$  are the distance of the edges of the step or the upper base of the frustum from the column axis in y direction. These are always given values.

Footing parameters:

Concrete material of the footing

- t foundation depth (distance between the bottom of the base plate and the 0 level)
- $h_2$ step height (height of the step or the frustum)
- $h_1$ base plate thickness
- blind concrete thickness  $h_b$
- friction coefficient between the footing and the blind concrete  $\Phi_{cvk}$

Under the edit fields the footing and the column is displayed in top view. Given sizes are drawn as continuous lines, upper limits as dashed lines. The forces appear as red crosses placed according to their eccentricities. This diagram is for orientation purposes only because the actual eccentricities are calculated taking into account the self weight of the footing and the backfill reducing the eccentricity.



If the button Show all support forces is down, the view is scaled to show all force crosses. If the button is up only crosses within the bounding rectangle of the footing are displayed.



Reinforcement

On the *Reinforcement* tab reinforcement calculations can be turned on. Rebar steel grade, x and y top and bottom rebar diameters and concrete covers must be specified.

ooting Footing	design param Reinforcement	eters Soil	1
	alculate reinforce	Rebar	r steel B500A
Con	crete cover		Diameter
	$\times_{top}$ [mm] = 40	•	Ø [mm] = 20 💌
	y _{top} [mm] = 60	•	Ø [mm] = 20 💌
×b	ottom [mm] = 40	-	Ø [mm] = 20 💌
УЪ	ottom [mm] = 60	•	Ø [mm] = 20 💌

At the *Soil* tab you can specify the soil profile and the properties of the backfill. Soil profiles can be saved under a name and can be reloaded.

Properties of the selected layer is displayed in the *Soil* group box. Properties of the backfill is displayed in the *Backfill* group box.

Soil layer properties can be changed. These changes can be applied to the soil layer clicking the *Modify layer* button. Layer name and description can be modified. Layer color can be changed clicking the small color rectangle beside the name. Soil library icon is placed beside the color rectangle. Clicking this icon a soil library is displayed with predefined layer properties.

Footing design parameters	x
Footing design parameters         Footing Reinforcement       Soil         Soil profile       3         V       LK10         LK10       1         LS10       Sodorható, kövér agy         Soil type       Fine         Y [kg/       Q         4       LP10       C [kN/         5       Layer thickness       r         Modify la       Modify la	Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector         Image: Solid vector       Image: Solid vector </th
	OK Cancel

Soil layers have the following properties:

 $\begin{array}{ll} \mbox{Soil type} & \mbox{coarse, coarse underwater or fine} \\ \gamma [kg/m^3] & \mbox{mass density} \\ \phi [^{\circ}] & \mbox{internal angle of friction} \\ \phi_t [^{\circ}] & \mbox{Angle of friction between the soil and concrete} \\ E_0 [N/mm^2] & \mbox{Young modulus of the soil} \\ \mu [] & \mbox{Poisson coefficient of the soil} \\ c [kN/m^2] & \mbox{cohesion (only for fine soils)} \end{array}$ 

Soil

### Soil database

Clicking the *Soil database* icon two tables are displayed. After selecting a soil and clicking the OK button (or double clicking the soil) properties of the selected soil are copied to the *Soil* or *Backfill* group box.

		_										
Coarse		dry or damp humid		underv	underwater		Fine		Void ratio	consistence		ce eo#
les,	Loose	ASL	ANL	AV	L	ł			0.4	SUIT IKA	154	son
vel	Solid	AST	ANT	AV	т		Silt		0.5	IK5	104	ID5
on-sitty,	Loose	BSL	BNL	BV	L			0,5	IK7	155	IP7	
gravel	Solid	BST	BNT	BV	т			1.0	IK10	1510	IP10	
enous,	Loose	CSL	CNL	CV	L	ŀ			0.4	JK4	1010	
d medium	Solid	CST	CNT	CV	т			0,5	JK5	JS5		
na	1	DCI	DNI	D1/			Lean clay		0,7	JK7	JS7	JP7
ty sand	Colid	DSL	DNL	DV	L T				1,0	JK10	JS10	JP10
oue fine	Loopo	DS1 ECI	DINT	DV EV		ľ			0,4	KK4		
v sand	Colid	ECT	ENIT	EV	- -		Medium clay		0,5	KK5	KS5	
,	Loose	ESI	ENI	EVI					0,7	KK7	KS7	KP7
e sand	Colid	FOL	ENIT	EV/	- T				1,0	KK10	KS10	KP10
	Loose	051	ONI	FVI OV4			Fat clay		0,4	LK4		
sitty sand	Colid	OSL	ONT	01	GVI				0,5	LK5	LS5	
	Joliu	031	ON						0,7	LK7	LS7	LP7
									1,0	LK10	LS10	LP10
									_			
γ[kg/m ³ ]	φ[°]	φ	(")	φ _{ZS} [*]	E _D (N	l/mr	n²]	нU				
2100	38.00	32	.00	27.00		98		0.10				
Solid, dry, sandy gravel											ж	
	Coarse les, vel an-sitty, gravel enous, d medium ad ty sand ous fine y sand e sand sitty sand e sand sitty sand	Coarse         Loose           Vel         Solid           on-sitty,         Loose           on-sitty,         Loose           on-sitty,         Loose           on-sitty,         Loose           ous rine         Solid           vs and         Loose           solid         Solid           ous rine         Loose           solid         Loose           solid         Solid           sitty sand         Loose           Solid         Solid           y (kg/m²)         Qt(1)           2100         38.00	Coarse         dry or dry or dry or ASL           Loose         ASL           Solid         ASL           Solid         ASL           solid         ASL           gravel         Solid         BSL           gravel         Solid         BSL           gravel         Solid         CSL           dimedum         Solid         CSL           thedum         Solid         DST           solid         ESL         Solid           solid         CST         Solid           solid         EST         Solid           solid         FSL         Solid           Solid         Solid         GST           y[kg/m ² ]         w[1"]         wp           y1(sg/m ² )         038.00         32           Solid         GST         Solid	Coarse         dry or damp         humid damp           Solid         ASL         ANL           Solid         AST         ANT           Solid         AST         ANT           nn-sity, on-sity, dimedum         Loose         SSL         BNL           Solid         BST         BNT         Not           Solid         CST         ONT         Not           dimedum         Solid         CST         DNL           Solid         DST         DNT         Not           y and         Solid         EST         ENL           Solid         FST         FNT         Solid         FST           Solid         FST         FNL         Solid         FST           Solid         FST         FNL         Solid         SST           Solid         SST         SNT         SOL         SOL           Y(Mg/m ² )         W(1 <b>P</b> (1         SOL         SOL <tr< td=""><td>Coarse         dry or dry or solid         humid ASL         underv ANL           Loose         ASL         ANL         AV           Vel         Solid         AST         ANT         AV           solid         AST         ANT         AV         AV           yn-sity, solid         Loose         ASL         BNL         BV           solid         AST         ENT         BV         BV           gravel         Solid         CSL         CNL         CVI           dmedum         Solid         CST         CNT         CV           ys and         Solid         CST         ENT         DV           y sand         Solid         EST         ENL         EV           solid         EST         ENL         EV         EV           solid         EST         ENL         EV         EV           solid         FST         ENT         EV         EV           solid         FST         ENL         EV         EV           solid         FST         ENL         EV         EV           Solid         GST         ENL         EV         EV              Solid</td><td>Coarse         dry or damp         numid         underwater           ies, vel         Lose         ASL         ANL         AVL           Solid         AST         ANI         AVT           pn-sity, vel         Loses         BSL         BNL         BVL           solid         EST         BNT         BVT         BVT           pn-sity, versity         Loses         BSL         BNL         BVL           graved         Solid         EST         BNT         BVT           graved         Solid         CSL         CNL         CVL           dmedum         Solid         CST         DNT         DVT           ysand         Solid         EST         ENL         EVL           solid         EST         FNL         FVL         solid           ysand         Losse         CSL         ENL         EVL           solid         FST         FNL         FVT           Solid         FST         FNL         FVT           solid         GST         GNT         QVT           y(kgtm²)         QfT         Qr         Qr         Solid           Solid         GST         GNT</td><td>Coarse         dry or damp         humid damp         underwater humid           ies, ies, ies, ies, ies, ies, ies, ies,</td><td>Coarse         dry or damp         humid         underwater underwater           ies, vel         Solid         ASL         ANL         AVL           Solid         AST         ANL         AVL           yn-sitty, vn-sitty, dimedum         Loose         SSL         BNL         BVL           Solid         BST         BNT         BVL         BVL           dimedum         Solid         CSL         CNL         CVL           did         Solid         CSL         NIL         DVL           did         Solid         CSL         CNL         CVL           did         Solid         CSL         NIL         DVL           did         Solid         CSL         DNL         DVL           grand         Solid         CST         NIT         FVL           solid         FST         FNL         FVL           solid         CST         NIT         GVT           solid         CST         NIT         FVT           solid         CST         NIT         FVT           solid         CST         NIT         GVT           solid         CST         NIT         GVT      <t< td=""><td>Coar se         dry or damp         humid         underwater underwater         Fine           ies, vel         Solid         ASL         ANL         AVL         Solid         Solid         Solid         Solid         Main         AVT         Solid         Solid<td>Coarse         dry or damp         humid         underwater underwater         Fine         Void ratio           ies, vel         Solid         ASL         ANL         AVL         AVL         0,4           solid         AST         ANL         AVL         AVL         0,4         0,5           on-sitty, vel         Loose         BSL         BNL         BVL         10         0,4           formous, vel         Loose         CSL         CNL         CVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,6         0,7           solid         FST         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         GVL         0,5         0,7         0,7           stitly sond         Loose         GSL         GNL         GVL</td><td>Coarse         dry or damp         humid         underwater underwater         Fine         Voidi value         coal of stribut           ies, ies, isodi         ASL         ANL         AVL           Solid         AST         ANL         AVL           Solid         ASL         ANL         AVL           Solid         ASL         ANL         AVL           Solid         BST         BNL         BVL           Immous, ind medum did         Loose         CSL         CNL         CVL           Solid         CST         ONL         DVL         Lean clay           Solid         CST         NIL         PVL         0.4         K4           vand         Solid         CST         NIL         DVL         0.4         K4           vand         Solid         CST         NIL         PVL         0.4         K4           vand         Solid         FST         FNL         FVL         0.7         K7           vand         Solid         FST         FNL         FVL         0.7         K7           Solid         FST         FNL         FVL         0.4         LK4           Solid         SST<!--</td--><td>Coarse         dry or damp         humid         underwater underwater         Fine         Void rabit         consisten rabit           ies, vel         Sold         ASL         ANL         AVL         Multicit         Iff         Iff</td></td></td></t<></td></tr<>	Coarse         dry or dry or solid         humid ASL         underv ANL           Loose         ASL         ANL         AV           Vel         Solid         AST         ANT         AV           solid         AST         ANT         AV         AV           yn-sity, solid         Loose         ASL         BNL         BV           solid         AST         ENT         BV         BV           gravel         Solid         CSL         CNL         CVI           dmedum         Solid         CST         CNT         CV           ys and         Solid         CST         ENT         DV           y sand         Solid         EST         ENL         EV           solid         EST         ENL         EV         EV           solid         EST         ENL         EV         EV           solid         FST         ENT         EV         EV           solid         FST         ENL         EV         EV           solid         FST         ENL         EV         EV           Solid         GST         ENL         EV         EV              Solid	Coarse         dry or damp         numid         underwater           ies, vel         Lose         ASL         ANL         AVL           Solid         AST         ANI         AVT           pn-sity, vel         Loses         BSL         BNL         BVL           solid         EST         BNT         BVT         BVT           pn-sity, versity         Loses         BSL         BNL         BVL           graved         Solid         EST         BNT         BVT           graved         Solid         CSL         CNL         CVL           dmedum         Solid         CST         DNT         DVT           ysand         Solid         EST         ENL         EVL           solid         EST         FNL         FVL         solid           ysand         Losse         CSL         ENL         EVL           solid         FST         FNL         FVT           Solid         FST         FNL         FVT           solid         GST         GNT         QVT           y(kgtm ² )         QfT         Qr         Qr         Solid           Solid         GST         GNT	Coarse         dry or damp         humid damp         underwater humid           ies, ies, ies, ies, ies, ies, ies, ies,	Coarse         dry or damp         humid         underwater underwater           ies, vel         Solid         ASL         ANL         AVL           Solid         AST         ANL         AVL           yn-sitty, vn-sitty, dimedum         Loose         SSL         BNL         BVL           Solid         BST         BNT         BVL         BVL           dimedum         Solid         CSL         CNL         CVL           did         Solid         CSL         NIL         DVL           did         Solid         CSL         CNL         CVL           did         Solid         CSL         NIL         DVL           did         Solid         CSL         DNL         DVL           grand         Solid         CST         NIT         FVL           solid         FST         FNL         FVL           solid         CST         NIT         GVT           solid         CST         NIT         FVT           solid         CST         NIT         FVT           solid         CST         NIT         GVT           solid         CST         NIT         GVT <t< td=""><td>Coar se         dry or damp         humid         underwater underwater         Fine           ies, vel         Solid         ASL         ANL         AVL         Solid         Solid         Solid         Solid         Main         AVT         Solid         Solid<td>Coarse         dry or damp         humid         underwater underwater         Fine         Void ratio           ies, vel         Solid         ASL         ANL         AVL         AVL         0,4           solid         AST         ANL         AVL         AVL         0,4         0,5           on-sitty, vel         Loose         BSL         BNL         BVL         10         0,4           formous, vel         Loose         CSL         CNL         CVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,6         0,7           solid         FST         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         GVL         0,5         0,7         0,7           stitly sond         Loose         GSL         GNL         GVL</td><td>Coarse         dry or damp         humid         underwater underwater         Fine         Voidi value         coal of stribut           ies, ies, isodi         ASL         ANL         AVL           Solid         AST         ANL         AVL           Solid         ASL         ANL         AVL           Solid         ASL         ANL         AVL           Solid         BST         BNL         BVL           Immous, ind medum did         Loose         CSL         CNL         CVL           Solid         CST         ONL         DVL         Lean clay           Solid         CST         NIL         PVL         0.4         K4           vand         Solid         CST         NIL         DVL         0.4         K4           vand         Solid         CST         NIL         PVL         0.4         K4           vand         Solid         FST         FNL         FVL         0.7         K7           vand         Solid         FST         FNL         FVL         0.7         K7           Solid         FST         FNL         FVL         0.4         LK4           Solid         SST<!--</td--><td>Coarse         dry or damp         humid         underwater underwater         Fine         Void rabit         consisten rabit           ies, vel         Sold         ASL         ANL         AVL         Multicit         Iff         Iff</td></td></td></t<>	Coar se         dry or damp         humid         underwater underwater         Fine           ies, vel         Solid         ASL         ANL         AVL         Solid         Solid         Solid         Solid         Main         AVT         Solid         Solid <td>Coarse         dry or damp         humid         underwater underwater         Fine         Void ratio           ies, vel         Solid         ASL         ANL         AVL         AVL         0,4           solid         AST         ANL         AVL         AVL         0,4         0,5           on-sitty, vel         Loose         BSL         BNL         BVL         10         0,4           formous, vel         Loose         CSL         CNL         CVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,6         0,7           solid         FST         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         GVL         0,5         0,7         0,7           stitly sond         Loose         GSL         GNL         GVL</td> <td>Coarse         dry or damp         humid         underwater underwater         Fine         Voidi value         coal of stribut           ies, ies, isodi         ASL         ANL         AVL           Solid         AST         ANL         AVL           Solid         ASL         ANL         AVL           Solid         ASL         ANL         AVL           Solid         BST         BNL         BVL           Immous, ind medum did         Loose         CSL         CNL         CVL           Solid         CST         ONL         DVL         Lean clay           Solid         CST         NIL         PVL         0.4         K4           vand         Solid         CST         NIL         DVL         0.4         K4           vand         Solid         CST         NIL         PVL         0.4         K4           vand         Solid         FST         FNL         FVL         0.7         K7           vand         Solid         FST         FNL         FVL         0.7         K7           Solid         FST         FNL         FVL         0.4         LK4           Solid         SST<!--</td--><td>Coarse         dry or damp         humid         underwater underwater         Fine         Void rabit         consisten rabit           ies, vel         Sold         ASL         ANL         AVL         Multicit         Iff         Iff</td></td>	Coarse         dry or damp         humid         underwater underwater         Fine         Void ratio           ies, vel         Solid         ASL         ANL         AVL         AVL         0,4           solid         AST         ANL         AVL         AVL         0,4         0,5           on-sitty, vel         Loose         BSL         BNL         BVL         10         0,4           formous, vel         Loose         CSL         CNL         CVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,4         0,6           formous, vel         Loose         DSL         DNL         DVL         0,6         0,7           solid         FST         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         FVL         0,4         0,5         0,7           solid         FSL         FNL         GVL         0,5         0,7         0,7           stitly sond         Loose         GSL         GNL         GVL	Coarse         dry or damp         humid         underwater underwater         Fine         Voidi value         coal of stribut           ies, ies, isodi         ASL         ANL         AVL           Solid         AST         ANL         AVL           Solid         ASL         ANL         AVL           Solid         ASL         ANL         AVL           Solid         BST         BNL         BVL           Immous, ind medum did         Loose         CSL         CNL         CVL           Solid         CST         ONL         DVL         Lean clay           Solid         CST         NIL         PVL         0.4         K4           vand         Solid         CST         NIL         DVL         0.4         K4           vand         Solid         CST         NIL         PVL         0.4         K4           vand         Solid         FST         FNL         FVL         0.7         K7           vand         Solid         FST         FNL         FVL         0.7         K7           Solid         FST         FNL         FVL         0.4         LK4           Solid         SST </td <td>Coarse         dry or damp         humid         underwater underwater         Fine         Void rabit         consisten rabit           ies, vel         Sold         ASL         ANL         AVL         Multicit         Iff         Iff</td>	Coarse         dry or damp         humid         underwater underwater         Fine         Void rabit         consisten rabit           ies, vel         Sold         ASL         ANL         AVL         Multicit         Iff         Iff



Add new soil layer

Move up Move down

Delete

The function available on the *Soil* toolbox are: *Add new soil layer, Move up, Move down, Delete.* 

Adds a new soil layer with the properties and layer thickness set in the group box. The new layer always gets to the bottom of the soil profile.

Moves the selected soil layer up within the soil profile.

Moves the selected soil layer down within the soil profile.

Deletes the selected soil layer from the soil profile.

Calculations

Soil rupture The size of the footing is increased until the design bearing pressure is smaller than the bearcheck ing resistance:  $q_{Ed} \le q_{Rd}$ .

## Warnings and errors:

If the bigger size of the footing exceeds 10 times the thickness a warning appears.

Reinforcement of the foundation base plate

*f* If rebar positions and diameter are specified the module determines the necessary amount
 *i* of top and bottom reinforcement in *x* and *y* direction according to the following diagram.
 *i* The minimum requirement is always taken into account.



The necessary rebar spacing is calculated from the rebar diameter. **Warnings and errors:** 

The program sends a warning if compression reinforcement is required or the calculated amount is more than the maximum allowed ( $A_s > 0.04 \cdot A_c$ ).

*Sliding check* The module determines if the design stress caused by horizontal force is under the sliding resistance between 1) the soil and the blind concrete, 2) the blind concrete and the foundation calculated from the effective area.

 $\tau_{Ed} \leq \tau_{Rd}$  and  $\tau_{Ed2} \leq \tau_{Rd2}$ 

*Calculating* according to *Eurocode* 7

Eurocode 7 allows different design approaches (DA). These are certain combinations of partial factors for actions, material properties and resistances. Partial factor sets applied to actions are referred to as A1, A2, sets applied to material properties are M1, M2, sets applied to resistances are R1, R2, R3. (See *EN 1997-1:2004, Annex A*) Each design approach combine these partial factor sets.

Design Approach		Combination	Actions	Material properties	Resistances
DA1	Combination 1	ULS	A1	M1	R1
	Combination 2	SLS	A2	M2	R1
DA2		ULS	A1	M1	R2
DA3		SLS	A2	M2	R3

The program checks A1+M1+R1 (DA1 / 1) and A1+M1+R2 (DA 2) for critical ULS combinations, A2+M2+R1 (DA1 / 2) and A2+M2+R3 (DA3) for critical SLS combinations. So for each critical combination two results are calculated.

If design was performed for a user-defined load combination set this combination to ULS or SLS otherwise the footing may be overdesigned.

Bearing resistance is  $q_{Rd} = s_{\gamma} \cdot \gamma' \cdot B' \cdot N_{\gamma} \cdot i_{\gamma} \cdot b_{\gamma} \cdot 0, 5 + s_{q} \cdot q \cdot N_{q} \cdot i_{q} \cdot b_{q} + s_{c} \cdot c \cdot N_{c} \cdot i_{c} \cdot b_{c}$ 

Sliding check calculates if the footing meets the following criterion between the footing and the blind concrete and between the blind concrete and the soil:

$$H_d \le R_d + R_{p;d}$$

where  $H_d$  is the design value of the horizontal force,  $R_d$  is the design shear resistance,  $R_{p;d}$  is the passive soil resistance at the side of the footing.

Design shear resistance is obtained from the formula

 $R_d = V_d \cdot \tan \delta_d \,,$ 

where  $V_d$  is the design vertical action,  $\delta_d$  is the design angle of friction:

$$\delta_d = \arctan\left(\frac{\tan\varphi}{\gamma_\varphi}\right).$$

where  $\varphi$  is the angle of interface friction,  $\gamma_{\varphi}$  is the partial factor of shearing resistance, prescribed by the design approach.

*Punching check* The module checks the shear resistance of the foundation ( $v_{Rd,max}$ ), at the perimeter of the column and determines the necessary amount of shear reinforcement.

The calculation reduces the punching force by the soil reaction on the effective area (and within the critical punching line).

The punching check is passed if  $v_{Ed} \leq v_{Rd}$ 

Without shear reinforcement 
$$v_{Rd} = \min \begin{cases} v_{Rd,c} \\ v_{Rd,max} \end{cases}$$
, with shear reinf.  $v_{Rd} = \min \begin{cases} v_{Rd,cs} \\ v_{Rd,max} \end{cases}$ 

#### Warning and errors:

If  $v_{Ed} \leq v_{Rd,c}$ , no shear reinforcement is necessary.

If  $v_{Rd,max} > v_{Ed} > v_{Rd,c}$ , shear reinforcement is necessary.

If  $v_{Ed} > v_{Rd,max}$ , the base plate fails due to punching. Plate thickness of column crosssection size should be increased.

If a stepped or sloped footing is designed, the size of the pedestal is determined checking the punching requirements so efficiency for punching is not calculated.

Predicting the settlement of footing

AxisVM calculates the elastic settlement caused by additional stress in soil layers. Loads cause the following stress at depth of *z* under the center of the centrally loaded rectangle of the footing (after Boussinesq-Steinbrenner):

$$\sigma_{z} = 4 \frac{\sigma_{0}}{2\pi} \left\{ \arctan\left[\frac{b}{z} \cdot \frac{a(a^{2} + b^{2}) - 2az(R - z)}{(a^{2} + b^{2})(R - z) - z(R - z)^{2}}\right] + \frac{bz}{b^{2} + z^{2}} \cdot \frac{a(R^{2} + z^{2})}{(a^{2} + z^{2})R} \right\},$$

where

*a* is the bigger side the centrally loaded rectangle of the footing,

*b* is the smaller side the centrally loaded rectangle of the footing,

 $\sigma_0$  is the soil stress at the footing base plane caused by loads (including the self-weight of the footing and the backfill minus the weight of the removed soil above the base plane),

and 
$$R = \sqrt{a^2 + b^2 + z^2}$$
.

This stress calculation is valid for a homogeneous half space. In case of soil layers effective layer thicknesses must be calculated:

$$h_{hi} = h_i \cdot \left(\frac{E_{si}}{E_{s0}} \cdot \frac{\rho_0}{\rho_i}\right)^{2/5},$$

where

 $h_{hi}$  is the effective thickness of the soil layer *i* 

 $h_i$  is the thickness of the soil layer i

 $E_{s0}$  is the Young modulus of the the base layer

 $E_{si}$  is the Young modulus of the soil layer *i* 

 $\rho_0$  is the density of the base soil layer

 $\rho_i$  is the density of the soil layer *i* 

AxisVM breaks up the user defined soil layers into 10 cm sublayers and calculates the stress due to soil weight and the stress caused by loading at the bottom of the sublayer. The change in sublayer thickness is calculated according to the following formulas:

$$\Delta h_i = h_i \cdot \frac{\sigma_{ai}}{E_{si}}$$
, where  $\sigma_{ai} = \frac{\sigma_{i-1} + \sigma_i}{2}$ 

 $\sigma_{ai}$  is the average stress caused by loading in sublayer i

 $\sigma_{i-1}$  is the average stress caused by loading at the top of sublayer i

 $\sigma_i$  is the average stress caused by loading at the bottom of sublayer i

 $E_{si}$ : the Young modulus of the sublayer i

The predicted settlement at a given depth is calculated as the sum of the changes in sublayer thicknesses for the sublayers above the level.:

$$s_m = \sum_{i=0}^m \varDelta h_i$$

AxisVM calculates the limit depth, where  $\sigma = 0.1 \cdot \sigma_{ob}$  (i.e. the extra stress caused by loading falls under the 10% of the stress due to soil self weight.

If this condition is not met at the bottom of the layer structure a settlement estimation is made based on the settlement at this point and the stress ratio (> 0.1) is calculated. If the stress caused by loading at the footing base plane is smaller than the stress due to the original soil layers settlement is not calculated.

AxisVM calculates the settlement for all load cases and SLS combinations. Stress and settlement functions are displayed for the selected load case. Settlement function s(z) is the total settlement of layers above z.

Results

320

The designed foundation will be displayed in top view with soil layers, punching circles and places dimension lines automatically. The 3D model can be zoomed in and out, shifted and rotated just like the main model.



If the display of settlement is activated (see *Display parameters*) a thick blue diagram plots the total soil stress against depth. Thin diagrams show the stress due to loading and the self-weight of the soil. The first one is decreasing, the second one is increasing with depth. Horizontal lines show the sublayers. The gray diagram on the other side of the axis is the settlement function.

The settlement displayed in the info window is the value of the settlement function at the limit depth (where the stress caused by loading is 10% of the stress due to self weight fo the soil).

If this condition is not met at the bottom of the layer structure a settlement estimation is made based on the settle-

ment at this point and the stress ratio (> 0.1) is calculated. If stress caused by loading at the bottom of the layer structure is still more than 10% of the stress due to soil self weight the limit depth cannot be determined as the further structure of the soil is unknown.

In this case the info window displays the value of the settlement function at the bottom of the layer structure as *>value*.

To improve the estimation further soil layer information must be added.



*Footing* This table displays the forces of the selected supports and the most important results in*internal forces* cluding calculated geometry.

As support forces are calculated in the local system of the support the *x* and *y* directions are the local *x* and *y* directions of the support. If the supports are global these are the global X and Y directions. Symbols:

Rx, Ry, Rz,	support forces
Rxx, Ryy,	
Rzz	
$q_{Ed}$	design bearing pressure
$q_{Rd}$	design bearing resistance
$q_{Ed}/q_{Rd}$	soil utilization factor
axb	local <i>x</i> direction bottom reinforcement (if calculated)
ayb	local <i>y</i> direction bottom reinforcement (if calculated)
axt	local <i>x</i> direction top reinforcement (if calculated)
ayt	local <i>y</i> direction top reinforcement (if calculated)
$ au_{Ed}/ au_{Rd}$	efficiency based on footing displacement relative to the blind concrete
$ au_{Ed2}/ au_{Rd2}$	efficiency based on blind concrete displacement relative to the soil
$v_{Ed}/v_{Rd}$	efficiency based on punching (for simple plate footings)
Settlement	predicted settlement of the footing
bx, by	footing base plate size in <i>x</i> and <i>y</i> direction
dx*, dy*	pedestal (step or frustum) size in <i>x</i> and <i>y</i> direction
ex*, ey*	eccentricity of the pedestal's center of gravity in <i>x</i> and <i>y</i> direction

Detailed Displays the data in the table of *Footing internal forces* and the following results:

# internal forces

design approach used to calculate the results of the line
<i>x</i> and <i>y</i> size of the effective rectangle
eccentricity of action in <i>x</i> and <i>y</i> direction
rebar scheme in bottom <i>x</i> direction (if calculated)
rebar scheme in bottom <i>y</i> direction (if calculated)
rebar scheme in top <i>x</i> direction (if calculated)
rebar scheme in top $y$ direction (if calculated)
design shear stress between the footing and the blind concrete
design shear resistance between the footing and the blind concrete
design shear stress between the soil and the blind concrete
design shear resistance between the soil and the blind concrete
minimum shear design resistance without punching reinforcement
maximum shear design resistance without punching reinforcement
shear design resistance with punching reinforcement
length of the critical line
shear reinforcement along the punching line
ratio of stress caused by loading and the stress due to self weight of the
soil (if limit depth is below the bottom of the layer structure its value is
determinded at that point and is greater than 0.1, otherwise it is 0.1)
the depth where stress ratio is 0.1 (if limit depth is greater than the bot-
tom of the layer structure a ? is displayed)

Copies image to the Clipboard

暍

⇔

Đ

Prints image to the Clipboard

Saves the drawing into the Drawing Library

Display parameters Turns on and off symbols of the drawing.

Display Parameters	x
Soil layers	
Eooting dimension lines	
Soil layer position symbols	
Reinforcement circles	
Force rectangles for individual forces	
🔽 Units	
Settlement of footing	
OK Cancel	

# 6.5.10. Design of COBIAX slabs



If the AxisVM configuration includes the COBIAX (CBX) module, it is possible to place void formers into slabs achieving weight reduction (and concrete reduction) and making larger spans available. For definition of COBIAX slabs see 4.9.4.1. COBIAX-domain.

Design codes

This design is available according to Eurocode, DIN 1045-1 and SIA (Swiss) design code.

COBIAX design must take into account that void formers reduce the stiffness and shear resistance of the slab. The effect of smaller bending stiffness can be seen in the results. Where shear forces would exceed the reduced shear resistance, placing of void formers must be avoided.

If the user defined the surface reinforcement parameters AxisVM calculates the design results used in reinforcement design. One of these design components is the difference between the actual shear force and the shear resistance. If actual reinforcement is also defined AxisVM calculates with the actual reinforcement.

Clicking on the Cobiax icon *vSz–vRd,c* will be displayed setting the color legend to show positive values (where shear force exceeds the resistance) in red and negative values in blue. No void formers should be placed into the red zones. In other words, these must be converted to solid areas.

Defining solid areas A toolbar appears to help solid area definition.

Existing solid areas and their polygon vertices can be moved.

Clicking on the *Update model* button converts solid areas into new domains without void formers. Due to changes in the model all resuls will be cleared.





Based on shear force isolines

AxisVM determines where to form solid areas based on the isolines of vSz-vRd,c.





*Update model* replaces solid areas with domains without void formers. Running the analysis again it can be checked whether any void former falls into a red zone. If so, new solid areas must be added or existing areas (domains) must be converted to solid areas and extended to remove void formers from red zones. The cycle of running the analysis and checking the distribution must be repeated until all void formers are removed from red zones.

### 6.6. Steel Design



### 6.6.1. Steel beam design based on Eurocode 3

**EUROCODE 3** The steel beam design module can be applied to the following shapes:

Rolled I shapes Welded I shapes Box shapes Pipe shapes Single-symmetric I shapes Tee shapes Rectangular (solid) shapes Round (solid) shapes Arbitrary shapes, some checks are not performed

Among elements with cross-section class 4, single- and double-symmetric I shaped, rectangular and box shaped cross-sections can be designed with this module. Effective section properties are calculated in the cases of uniform compression and uniform bending. These properties can be found in the Table Browser under Steel design, in the table Design Resistances, or in the pop-up window after clicking on the element:

- A_{eff} area of the effective cross section when subjected to uniform compression
- $e_{N,y}$  the shift of the y neutral axis when the cross-section is subjected to uniform compression (will be zero if the section is symmetric to axis y). Negative shift will cause a negative  $\Delta My = N \cdot e_{N,y}$  moment in the actual cross-section.
- W_{eff,min} elastic section mudulus (corresponding to the fibre with the maximum elastic stress) of the effective cross section when subjected only to moment about the relevant axis.
- W_{eff,(-),min} refers to sections where the moment is positive

 $W_{eff,(+),min}$  refers to those where the moment is negative

It is important to know, that these section properties are calculated when the section is in class 4. I might happen that there is no stress causing buckling, but the properties will still be available in the TableBrowser.

It is assumed that the cross-sections do not have holes in them and are made of plates with a thickness less than or equal to 40 mm.

The cross section should be constant or tapered. It is also assumed that the loads on singlesymmetric cross-sections act in the plane of symmetry, that is the plane of bending. For general shapes with no plane of symmetry only Axial Force-Bending-Shear (N-M-V) and Compression-Bending-Buckling (N-M-Buckling) is checked.

*AxisVM performs the undermentioned checks only. All the other checks specified in the design code like constrained torsion, strutting forces, joints, etc. has to be completed by the user.* 

The principal axes of an arbitrary cross section have to be coincident with the local y and z axes.


Classes of Cross- Sections	The program is identifying the class of the cross-section bas considering coexisting compression and bending.	ed on EN 1993-1-1, Table 5.2,				
Checks	Axial Force-Bending-Shear [N-M-V]	(EN 1993-1-1, 6.2.1, 6.2.8)				
	Compression-Bending-Buckling (flexural in plane or torsional	) [N-M-Buckl.]				
		(EN 1993-1-1, 6.3.3)				
	Axial force-Bending-Lateral Tors. Buckling [N-M-LTBuckl.]	(EN 1993-1-1, 6.3.3)				
	Shear /y [V _v ]	(EN 1993-1-1, 6.2.6)				
	Shear $/z [V_z]$	(EN 1993-1-1, 6.2.6)				
	Web Shear-Bending-Axial Force [V _w -M-N]	(EN 1993-1-1, 6.2.1, 6.2.8)				
Resistances	Plastic resistance (axial) [N _{pl,Rd} ]	(EN 1993-1-1, 6.2.4)				
	Effective resistance (when subjected to uniform compression) [N _{eff,Rd} ]					
		(EN 1993-1-1, 6.2.4)				
	Plastic Shear Resistance /y axis [V _{pl.v.Rd} ]	(EN 1993-1-1, 6.2.6)				
	Plastic Shear Resistance /z axis $[V_{pl.z.Rd}]$	(EN 1993-1-1, 6.2.6)				
	Shear Web Buckling $[V_{b,Rd}]$	(EN 1993-1-5, 5.2-3)				

Elastic Moment Resistance /yy [M _{el,y,Rd} ]	(EN 1993-1-1, 6.2.5)
Elastic Moment Resistance /zz [M _{el,z,Rd} ]	(EN 1993-1-1, 6.2.5)
Plastic Moment Resistance /yy [M _{pl,y, Rd} ]	(EN 1993-1-1, 6.2.5)
Plastic Moment Resistance /zz [M _{pl,z,Rd} ]	(EN 1993-1-1, 6.2.5)
Moment Resistance for effective cross-section subjected to	bending around axis y [M _{pl,y, Rd} ] (EN 1993-1-1, 6.2.5)
Moment Resistance for effective cross-section subjected to	bending around axis z [M _{pl,z,Rd} ] (EN 1993-1-1, 6.2.5)
Minimal Buckling (flexural in plane or torsional) Resistance	[N _{b,Rd} ]
	(EN 1993-1-1, 6.3.1)
Lateral Torsional Buckling Resistance [M _{b,Rd} ]	

These informations are given by the program as auxiliary results. The checks are mostly defined by interaction formulae. The definition and the detailed conditions of the application of the variables contained by the equations can be found in the design code.

In the following,  $N_{Rk} = f_y A$ ,  $M_{y,Rk} = f_y W_y$  and  $M_{z,Rk} = f_y W_z$ , where  $W_y = W_{pl,y}$  and  $W_z = W_{pl,z}$  for class 1 or 2 cross sections,  $W_y = W_{el,y}$  and  $W_z = W_{el,z}$  for class 3 cross sections and  $W_y = W_{eff,y}$  and  $W_z = W_{eff,z}$  for class 4 cross sections.

*Axial Force-* The member can be in tension or in compression. The check is performed on the basis of *Bending-Shear* EN 1993-1-1, 6.2.1 (7).

$$\frac{\frac{N_{Ed}}{N_{Rk}}}{\frac{\gamma_{M_0}}{\gamma_{M_0}}} + \frac{\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma_{M_0}}} + \frac{\frac{M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M_0}}} \le 1$$

 $\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$ : it differs from zero only when the cross section is in class 4 and the original cross section is asymetric to axis y.

#### High shear

If the shear force is greater than 50% of the shear resistance, the effect of shear force is considered as detailed below.

For section class 1. and 2. allowance is made on the resistance moment accoding to EN 1993-1-1, 6.2.8.

For section class 3. and 4. stresses are calculated and the general and conservative formula in EN 1993-1-1, 6.2.1 (5) is applied. This is done for section types: I, T, C, box and pipe. For other section types (L shape, rectangular and round sold shapes, and user defined shapes) the effect of hight shear has to be calculated by the user.

#### Plastic resistance check

For I, pipe and box shaped sectors in section class 1. and 2., the resistance check is performed according to EN 1993-1-1 6.2.10. Allowance is made for the effect of both shear force and axial force on the resistance moment. Besides resistance check of pure axial force and pure shear force, the following criterion should be satisfied:

$$\frac{M_{y,Ed}}{M_{N,y,Rd}} \le 1$$
$$\frac{M_{z,Ed}}{M_{N,z,Rd}} \le 1$$

where  $M_{N,y,Rd}$ ,  $M_{N,z,Rd}$ : reduced moment resistances based on the effect of shear force and axial force (EN 1993-1-1 6.2.8. and 6.2.9.1). For pipe sections, the reduced moment is calculated as follows:

$$M_{N,y,Rd} = 1,04 \cdot (1 - \rho - \frac{n^{1,7}}{(1 - \rho)^{0,7}}) \text{ where } n = \frac{N_{Ed}}{N_{pl,Rd}} \text{ and } \rho = \left(2\frac{V_{Ed}}{V_{pl,z,Rd}} - 1\right)^2$$
$$M_{N,z,Rd} = 1,04 \cdot (1 - \rho - \frac{n^{1,7}}{(1 - \rho)^{0,7}}) \text{ where } n = \frac{N_{Ed}}{N_{pl,Rd}} \text{ and } \rho = \left(2\frac{V_{Ed}}{V_{pl,y,Rd}} - 1\right)^2$$

For bi-axial bending the criterion in EN 1993-1-1 6.2.9.1. (6) should be satisfied:

$$\left[\frac{M_{y,Ed}}{M_{N,y,Rd}}\right]^{\alpha} + \left[\frac{M_{z,Ed}}{M_{N,z,Rd}}\right]^{\beta} \le 1$$

Bending-Buckling

*Compression*- The check is based on EN 1993-1-1, 6.3.3 (6.61) and (6.62):

$$\frac{N_{Ed}}{\chi_{y}} + k_{yy} \frac{M_{y,Ed} + \Delta M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma_{M_{1}}}} + k_{yz} \frac{M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M_{1}}}} \le 1$$

$$(\chi_{LT} = 1,0)$$

$$\frac{N_{Ed}}{\chi_{z}} + k_{zy} \frac{M_{y,Ed} + \Delta M_{y,Ed}}{\frac{M_{y,Rk}}{\gamma_{M_{1}}}} + k_{zz} \frac{M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M_{1}}}} \le 1$$

 $\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$ : it differs from zero only when the cross section is in class 4 and the original cross section is asymetric to axis y.

Axial Force-Bending-Lateral Torsional Buckling When determining the lateral-torsional buckling resistance it is assumed that the cross section is constant and symmetric for the local z axis. It is also assumed the the loads act in the plane of symmetry, that is the plane of bending. The value of k (ENV 1993-1-1, F1.2) is taken equal with  $K_z$  (buckling length factor). The weak axis should be the local z axis.

The check is based on the form of equations (6.61) and (6.62) of EN 1993-1-1, 6.3.3 :

$$\frac{N_{Ed}}{\chi_y \frac{N_{Rk}}{\gamma_{M_1}}} + k_{yy} \frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} \frac{M_{y,Rk}}{\gamma_{M_1}}} + k_{yz} \frac{M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M_1}}} \le 1$$

$$\frac{N_{Ed}}{\chi_z \frac{N_{Rk}}{\gamma_{M_1}}} + k_{zy} \frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} \frac{M_{y,Rk}}{\gamma_{M_1}}} + k_{zz} \frac{M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M_1}}} \le 1$$

 $\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$ : it differs from zero only when the cross section is in class 4 and the original cross section is asymptric to axis y.

 $\chi_{LT}\,$  is calcualted according to EN 1993-1-1 6.3.2.2 or 6.3.2.3.

The determination of the interaction factors of  $k_{yy}$ ,  $k_{yz}$ ,  $k_{zy}$  and  $k_{zz}$  is based on EN 1993-1-1, Appendix B Method 2 (Tables B.1 and B.2).

The equivalent uniform moment factors  $C_{my}$ ,  $C_{mz}$ ,  $C_{mLT}$  are listed in Table B.3.

For tensile axial force, the check is performed using the effective moments based on ENV 1993-1-1, 5.5.3.

*Shear /y* The check is performed on the basis of EN 1993-1-1, 6.2.6.

$$\frac{V_{y,Ed}}{V_{c,y,Rd}} \le 1$$

Shear /z The check is performed on the basis of EN 1993-1-1, 6.2.6.

$$\frac{V_{z,Ed}}{\min\left(V_{c,z,Rd}, V_{b,Rd}\right)} \le 1$$

 $V_{b,Rd} = V_{bw,Rd}$ . The resistance is calculated with the contribution of the web but not the flanges.

*Web Shear-* The check is performed for cross-sections with web (I and box sections) based on EN 1993-1-*Bending-Axial* 5 7.1, 6.2.8, 6.2.9 assuming that the web is parallel with the local z axis.

Force

$$\frac{M_{Ed}}{M_{pl,Rd}} + \left(1 - \frac{M_{f,Rd}}{M_{pl,Rd}}\right) \cdot \left(2\frac{V_{Ed}}{V_{bw,Rd}} - 1\right)^2 \le 1$$

In case of high shear force or high axial force formulas in EN 1993-1-1 6.2.8, 6.2.9 are applied.

## **Basic section types**

Section type		N-M-V Stress	N-M- Buckling	N-M- LT buckling	Shear Vy	Shear Vz	Shear buckling	Effec- tive section
Ι	Ι	~	~	~	~	~	~	~
Single symmetric I	Ι	~	~	V	~	~	~	~
Т	Т	~	~	~	~	~	-	-
Box		~	~	~	~	~	~	>
Welded box	Π	~	~	~	~	~	~	>
Pipe	0	~	~	~	~	~		
L	L	~	~	-	~	~	-	-
L equal	L	~	~	in case of normal force (no bending)	~	v	-	-
U	L	~	~	if bending acts in the plane of symmetry	~	~	-	-
С	С	~	~	if bending acts in the plane of symmetry	~	~	-	-
Round		~	~	~	~	~		
Rectangular		~	~	~	~	~	-	-

## **Double-sections**

Section type		N-M-V Stress	N-M- Stability	N-M- LT buckling	Shear Vy	Shear Vz	Shear buckling	Effec- tive section
2I	II	~	~	-	~	~	-	-
2I	Π	~	~	~	~	~	~	~
if a=0 (*)		•	•	•	•	•	•	•
2L	זר	~	~	-	~	~	-	-
2L	Τ			1			_	-
if a=0 (*)		V	•	•	•	•		
2U opened ][	30	~	~	-	~	~	-	-
2U opened ][	Ι							
if a=0 (*)		V	•	V	~	~		V
2U close []	[]	~	~	-	~	~	-	-
2U close []				./				
if a=0 (*)		V		•				•

#### Other section types

Section type		N-M-V Stress	N-M- Stability	N-M- LT buckling	Shear Vy	Shear Vz	Shear buck- ling	Effective section
Ζ	l	~	~	-	~	~	-	-
J	L				-			
Asymmetric C	L				-			
Asymmetric Z	С				-			
S	l				-			
Arc	(	~	~	-	~	~	-	-
Half circle	D	~	~	-	~	~	-	
Reg. polygon shape	0	~	>	-	~	~	-	-
Wedged I	Ŧ	~	>	~	~	~	~	~
Complex/ Other (**)	<b>↓</b>	~	~	-	~	~	-	-

- (*) For double-section types if the distance between the two sections is zero, the program will assume that the connection between the elements is continuous and will replace the two with one section (I, T or box). The connection needs to be calculated by the user.
- (**) These sections are designed only if local coordinates are the same as principal directions.
  - *[¬]* If the manufacturing process of the section is *cold-formed* or *other*, the member is not designed.

#### **Design Parameters**

For the design based on Eurocode 3, the following design parameters should be defined and assigned to the structural members:

Classification can be automatic or defined explicitly.

#### **Stability Parameters**

Buckling (flexural)

 $k_y$ ,  $K_z$ : buckling length factors corresponding to the *y* and *z* axis, respectively.

If a support is continuous along the member, constraining the buckling about an axis, the corresponding buckling length factor could be taken as nearly zero. In a similar case, when there are intermediate supports, constraining the buckling about an axis, the buckling length factor could be taken as the ratio of the corresponding buckling length (between the intermediate supports) and the length of the structural member.

Design Parameters - Eurocode 🛛 🔀
Material Steel Cross-Section IPE 240, HE 200 B, IPE 360, O 40, IPE 6
Automatic classification
Buckling Coefficients
Flexural Buckling
K _y = K ₂ = V
Lateral-Torsional Buckling
K _w = 1,00 💌 🔽 Konzol
Load position C Iop C Center of gravity
C Bottom C Custom $Z_a = $
Web Shear Buckling
No Stiffeners     a _{max} = 2,268 m
C Transversal Stiffeners a (m) = 0,080
Member preferences
Pick Up >> OK Cancel

# Lateral Torsional<br/>BucklingK_o: is a factor related to the constrain against warping. Its value must be between 0.5 and 1.-if warping is not constrained it is 1.0.

- if warping is constrained at both ends of the beam, it is 0.5.
- if warping is constrained at one of the ends of the beam, it is 0.7.

See in detail: Appendix F1 of ENV 1993-1-1.

 $C_1$ ,  $C_2$ ,  $C_3$ : are factors depending on the ratio of the end moments of the structural element, on  $K_z$  factor, and on the type of loading.  $C_1$  is calculated automaticaly. When external loads are applied to the structural member and the point of application of them is not coincident with the shear center of the cross section, a value for  $C_2$  shall be entered, based on ENV 1993-1-1, Table F1.2. In case of single-symmetric cross section  $C_3$  shall also be entered, based on ENV 1993-1-1, Table F1.2..

 $Z_a$ : is the z coordinate of the point of application of the transversal load (relative to the center of gravity of the cross-section), based on ENV 1993-1-1, Figure F1.1. The positions of the center of gravity and the top or bottom of the cross section can also be chosen by radio-buttons.

*Web Shear* For shapes with webs, the web can be supported or not with stiffeners:

Buckling No Stiffeners: assumes no transversal stiffeners along the structural member.

**Transversal Stiffeners:** there are transversal stiffeners at distance a each from the other along the structural member.

In any cases the program assumes that there are transversal stiffeners (non-rigid end post) at the ends of the structural members (e.g. at the supports).

*Steel structural* The design is performed on structural elements that can consist of one or more finite elements (beams and/or ribs). A group of finite elements can become a structural element only if the finite elements in the group satisfy some requirements checked by the program: to be located on the same straight line, to have the same material, cross-section, and to have parallel local coordinate systems.

The program allows two methods to define structural members as follows:

 Structural elements for steel design are not the same as the structural members (See... 3.2.12 Assemble structural members)

Any node of a selection set of finite elements where another finite element is connected will become an end-point of a structural element within the selection set of finite elements.



The finite elements in the selection set become only one structural element irrespective of other finite elements connecting to its nodes.



#### Diagrams

You can display the diagrams corresponding to all the checks by clicking on the structural member.



## 6.6.2. Bolted Joint Design of Steel Beams

AxisVM calculates the moment-curvature diagram, the resistance moment and initial strength of steel column-beam bolted joints based on Eurocode3 (Part 1.8 Design of Joints).

Type of joints

- The above type of joints can be calculated:
  - 1. beam to column joint

2. beam to beam joint



*Assumptions:* 

- The beam and column cross-sections are rolled or welded I shapes.

- The beam end plate connect to the flange of the column.
- The pitch range of the beam is beetwen  $\pm 30^{\circ}$ .
- The cross-section class should be 1, 2 or 3.
- The normal force in the beam should be less than  $0.05^* N_{pl,Rd}$

The program checks if these requirements are met.

The steps of the design

Select the beam and one of its end nodes.

(We can select several beams in one process if the selected beams have the same material and cross-section properties and connected columns also have the same material and cross-section properties.)



#### Click on the Joint Design icon.

The Bolted Joint Designer will appear:



#### Lets you assign the parameters of the joint in three steps.

*Bracings* We can assign horizontal, diagonal bracing plates and web thickening plates to increase the strength of the connection.



t2: thickening plate thickness on the beam web

*Web shear area* The program calculate the web shear area including the thickening plate area. If there is a hole in the web near to the connection you can decrease this value in the data field depending on the hole size.



End plate

Parameters of the end plate:

- thickness
- material
- welding thickness
- width of the end plate (a)
- height of the end plate (c)
- distance between top flange of the beam and top of the end plate (b)
- bolts in the extension of the end plate

Bolt rows can be assigned to the tensile part of the end plate.



The program places bolts in two columns symmetrical to the beam web. The same type of bolts is used in the connection.

Bolt parameters:

- material

- size

- number of rows
- distance of bolt columns (d)

In case of automatic positioning of bolts the program places bolt rows in equal distances. The program checks the required minimal distances between bolts and from the edge of plates.

Turn off the option Use default positions to place the bolt rows individually.

*An error message will appear if the distances does not meet the requirements.* 

Minimal bolt distances are checked based on EC2:

1. Between bolts:	2,2 d
2. From edge of plate	1,2 d
3. In a direction perpendicular to the force	1,2 d

**Results** When we click on the *Result* tab AxisVM calculates the Moment-curvature diagram, the design resistant moment  $(M_{rD})$  and the initial strength of the connection  $(S_{i,init})$ .



Bolted Join	t Designer - Ne E E E	w joint 						_ [] >
Model	M [kNm] 90.00 \$86.40	(Node 5,	ST1_1)			— (Node !	5, ST1_1)	<b></b>
	61.00+ 72.00+ 63.00+ 54.00+			-				
	45.00 - 36.00 - 27.00 - 18.00 -	0 5	4	7 0.41 3	o u	2 <del></del>		
<b>B</b> . ²⁴⁺ <b>B</b> .18+	9.00 + 9.00 + -9.00 & • • • •	- 0.1	+ 0.3	+ 0.3 + 0.3	+0+	Ω 0 φ  	°]	• •
						ок	Ca	ancel

[☞] A warning message will appear if the resistant moment is less than the design moment. The calculation method considers shear forces and normal forces together with the moments. As a consequence we can get different resistant moments (M_{rD}) for the same connection depending on the load cases (or combinations). Therefore AxisVM checks the M_{rD} ≥ M_{sD} condition in all load cases.





Load the connection parameters.



Save the connection parameters. Saved parameters can be loaded and assigned to other beam-end joints later.

Prints the displayed diagram. See... 3.1.10 Print



Copies the diagram to the Clipboard



Saves the diagram to the Gallery

The result table contains the followings:

- node number
- beam number
- name of the load case or combination
- design moment ( $M_{sD}$ )
- design resistant moment (M_{rD})
- a summary of the calculation results and intermediate results

## 6.7. Timber Beam Design

EUROCODE 5 (EN 1995-1-1:2004) The timber beam design module can be applied to the following cross-sections and timber materials:

- *a*) Rectangle for solid timber, Glued laminated timber (Glulam) and for Laminated veneer lumber (LVL)
- *b*) Round for solid timber



*Material* The material database contains the solid, Glulam and LVL timber material properties acproperties cording to the related EN standard. (Solid timber: EN338, Glulam: EN 1194)

Characteristic strength	Notation
Bending strength	$f_{m,k}$
Tensile strength parallel to grain	$f_{t,0,k}$
Tensile strength perpendicular to grain	$f_{t,90,k}$
Compression strength parallel to grain	$f_{c,0,k}$
Compression strength perependicular to grain in	f _{c,90,k,y}
<i>y</i> direction [*]	
Compression strength perependicular to grain in	$f_{c,90,k,z}$
z direction [*]	-
Shear strength perpendicular to the grain in y	$f_{v,k,y}$
direction [*]	•
Shear strength perpendicular to the grain in z	$f_{v,k,z}$
direction	

^{*}In case of solid and Glulam timber  $f_{v,k,z} = f_{v,k,y} = f_{v,k}$  and  $f_{c90,k,z} = f_{c90,k,y} = f_{c90,k}$ 

Modulus of elasticity	Notation
Mean value parallel to grain	E _{0,mean}
Mean value perpendicular to grain	E _{90,mean}
5% value of modulus parallel to grain	$E_{0,05}$
Mean value of shear modulus	G _{mean}

Density	Notation
Characteristic value of density	$ ho_k$
Mean value of density	$ ho_{mean}$
Partial factor	Notation
Partial factor for material	<b>ү</b> м
Size effect factor	Notation
for LVL timber	S

*Timber classes* Timber elements must have a service class. Service class can be set in the line elements definition dialog, at Service Class field. **See...** 4.9.7 Line Elements Service classes (EN 1995-1-1, 2.3.1.3):

**Service class 1** – where the average moisture content in most softwoods will not exceed 12%. This corresponds to a temperature of  $20^{\circ}$ C and a relative humidity of the surrounding air only exceeding 65% for a few weeks per year.

**Service class 2** – where the average moisture content in most softwoods will not exceed 20%. This corresponds to a temperature of 20°C and a relative humidity of the surrounding air only exceeding 85% for a few weeks per year.

Service class 3 – where the average moisture content in most softwoods exceeds 20%.

Design strength and other design properties of the timber materials depend on the service class.

*Load-duration* Timber design module requires information on the load duration. So if a timber material has *classes* been defined in the model load case duration class can be entered. **See...** 4.10.1 Load Cases, Load Groups

*Design strength* The design values of strength is calculated from the characteristic values of strength accord*components* ing to the following formulas:

In case of  $f_{t,90,d}$ ,  $f_{c,0,d}$ ,  $f_{c,90,d}$ ,  $f_{v,d}$  (Solid, Glulam, LVL timbers):

$$f_{d} = \frac{k_{\text{mod}} \cdot f_{k}}{\gamma_{M}}$$
  
In case of  $f_{m,d}$  (Solid, Glulam, LVL timbers):  
 $f_{d} = \frac{k_{\text{mod}} \cdot k_{h} \cdot f_{k}}{\gamma_{M}}$   
In case of  $f_{t,0,d}$  (Solid and Glulam timbers):  
 $f_{d} = \frac{k_{\text{mod}} \cdot k_{h} \cdot f_{k}}{\gamma_{M}}$   
In case of  $f_{t,0,d}$  (LVL timber):  
 $f_{d} = \frac{k_{\text{mod}} \cdot k_{l} \cdot f_{k}}{\gamma_{M}}$  where,

$$\gamma_M$$

 $k_{mod}$  modification factor (EN 1995-1-1, 3.1.3)

- *k*_{*h*} depth factor (EN 1995-1-1, 3.2, 3.3, 3.4)
- $k_l$  length factor for LVL timber (EN 1995-1-1, 3.4)
- $f_k$  characteristic strength
- $\gamma_M$  partial factor of material (EN 1995-1-1, Table 2.3)
- $k_h$  factor The  $f_{m,k}$  and  $f_{t,0,k}$  characteristic strength values are determined for a reference depth of beam. In case of solid and Glulam timber if the depth (*h*) of the cross-section less than the reference value, the design strength is multiplied with the following factor.

Solid timber:  $k_h = \min\left\{ \left(\frac{150}{h}\right)^{0,2}; 1,3 \right\}$  (if  $\rho_k \le 700 \text{ kg/m}^3$ ) Glulam:  $k_h = \min\left\{ \left(\frac{600}{h}\right)^{0,1}; 1,1 \right\}$  In case of LVL timber if the depth (*h*) of the cross-section not equal to the reference value, the design strength is multiplied with the following factor.

LVL: 
$$k_h = \min\left\{\left(\frac{300}{h}\right)^s; 1, 2\right\}$$
 (where *s* is the size effect exponent)

*h* is the cross-section depth in mm.

Reference depths are the following,

- solid timber: 150 mm
- Glulam: 600 mm
- LVL: 300 mm
- $k_l$  factor The  $f_{t,0,k}$  characteristic strength value of LVL timber is determined for a reference length of beam. If the length (*l*) of the beam not equal to the reference length, the design strength is multiplied with the following factor.

$$k_l = \min\left\{\left(\frac{3000}{l}\right)^{\frac{s}{2}}; 1, 1\right\}$$
 (where *s* is the size effect exponent)

l is the beam length in mm. Reference length: 3000 mm.

Moduluses

for analysis

tions

Analysis type	Modulus (SLS)	Modulus (ULS)
First order,	$E_{mean.fin} = \frac{E_{mean}}{(1+k_{def})}$	$E_{mean.fin} = \frac{E_{mean}}{(1 + \psi_2 k_{def})}$
intear elastic	$G_{mean.fin} = \frac{G_{mean}}{(1 + k_{def})}$	$G_{mean.fin} = \frac{G_{mean}}{(1 + \psi_2 k_{def})}$
Second order,	$E_d = \frac{E_{mean}}{\gamma_M}$	$E_d = \frac{E_{mean}}{\gamma_M}$
linear elastic	$G_d = \frac{G_{mean}}{\gamma_M}$	$G_d = \frac{G_{mean}}{\gamma_M}$
Frequency	$E_{mean}$ , $G_{mean}$	$E_{mean}$ , $G_{mean}$

Conservative way  $\psi_2 = 1,0$  is used.

- *Design assump-* There is no hole or other weaking in the beams.
  - The cross-section constant (rectangle, round) or linear changing depth along the beam (tapered beam).
  - The grain parallel with the beam *x* axis.
  - In case of tapered beam the grain paralel one of the longitudinal edge.
  - The dominant bending plane is the *x*-*z* plane of the beam (moment about *y* axis).
  - $I_y \ge I_z$
  - In case of Glulam the laminates are parallel with the *y* axis of the cross-section.
  - in case of LVL the laminates are parallel with the *z* axis of the cross-section.



Checks	Normal force-Bending [N-M]	(EN 199	95-1-1, 6.2.3, 6.2.4)
	Compression-Bending-Buckling (in plane) [N-M-Buckling]	(EN 199	95-1-1, 6.3.2)
	Normal force-Bending-Lateral tors. buckling [N-M-LT buckling]	(EN 199	95-1-1, 6.3.3)
	Shear /y -Torsion /x $[V_y-T_x]$	(EN 199	95-1-1, 6.1.7, 6.1.8)
	Shear /z -Torsion /x $[V_z-T_x]$	(EN 199	95-1-1, 6.1.7, 6.1.8)
	Moment /y - Shear /z (tensile stress perpendicular to the grain) [	$[M_y-V_z]$	(EN 1995-1-1, 6.4.3)
Calculated	$\lambda_{rel,y}$ Relative slenderness ratio ( <i>y</i> ) /in <i>z</i> - <i>x</i> plane of the beam/ []		
parameters	$\lambda_{\text{rel},z}$ Relative slenderness ratio (z) /in y-x plane of the beam/ []		
	$k_{c,y}$ Buckling (instability) factor (y) /in z-x plane of the beam/ []	(EN 199	95-1-1, 6.3.2)
	$k_{c,z}$ Buckling (instability) factor (z) /in x-y plane of the beam/ []	(EN 199	95-1-1, 6.3.2)
	k _{crit} Lateral torsional buckling factor []	(EN 199	95-1-1, 6.3.3)
	k _h Depth factor []	(EN 199	95-1-1, 3.2, 3.3, 3.4)
	k _{mod} modification factor []	(EN 199	95-1-1, 3.1.3)
	$\sigma_{t,90,d}$ (tensile stress perpendicular to the grain) [N/mm ² ]	(EN 19	95-1-1, 6.4.3)

AxisVM performs the following checks only. All the other checks specified in the design code like supports, joints, etc. has to be completed by the user.

*Normal force-* The design value of normal force can be tension or compression.

Bending Tension and moment (EN 1995-1-1, 6.2.3)

$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$
$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + k_m \frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

Compression and moment (EN 1995-1-1, 6.2.4)

$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$
$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + k_m \frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

where,

 $k_m = 0.7$  in case of rectangle cross-section  $k_m = 1.0$  in all other cases

Compression-Moment-Buckling

(EN 1995-1-1, 6.3.2)  

$$\frac{\sigma_{c,0,d}}{k_{c,y} \cdot f_{c,0,d}} + \frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

$$\frac{\sigma_{c,0,d}}{k_{c,z} \cdot f_{c,0,d}} + k_m \frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

where,

 $k_{c,y}$  Buckling (instability) factor (*y*) /in *z*-*x* plane of the beam/ (EN 1995-1-1, 6.3.2)  $k_{c,z}$  Buckling (instability) factor (*z*) /in *x*-*y* plane of the beam/ (EN 1995-1-1, 6.3.2) In case of tension force the  $f_{c,0,d}$  is replaced with  $f_{t,0,d}$ , and  $k_{c,y} = k_{c,z} = 1,0$  Normal force-Bending-LT buckling

*ce*- For lateral torsional buckling check the program assumtions that the beam is bending in *z*-*x ig*- plane (about *y* axis). If there is simultaneous  $M_z$  moment on the beam and the compression stress from  $M_z$  reach the 3% of the  $f_{c,0,d}$  a warning message appears.

Bending only (EN 1995-1-1, 6.3.3)

$$\frac{\sigma_{m,d}}{k_{crit} \cdot f_{m,d}} \le 1$$

Compression and moment (EN 1995-1-1, 6.3.3)

$$\left(\frac{\sigma_{m,d}}{k_{crit} \cdot f_{m,d}}\right)^2 + \frac{\sigma_{c,d}}{k_{c,z} \cdot f_{c,0,d}} \le 1$$

Tension and bending

In case of small tension and bending that lateral torsional buckling could be occur, however there is no rule in EC5 for this case.

The following conservative check is used.

$$\frac{\left|\sigma_{mt,d}\right|}{k_{crit} \cdot f_{m,d}} \le 1 \quad \text{where } \sigma_{mt,d} = \frac{M_d}{W_y} + \frac{N_d}{A} < 0 \text{ where,}$$

 $k_{crit}$  is the lateral buckling factor according to the following table:

$\lambda_{rel,m} \leq 0,75$	$k_{crit} = 1,0$
$0,75 < \lambda_{rel,m} \leq$	$k_{crit} = 1,56-$
$\lambda_{rel,m} \leq 0,75$	$k_{crit} = 1/\lambda_{rel,m}^2$

*Shear-Torsion* There is no rule in EC5 for case of simultaneous shear force and torsional moment. In this case the program uses the interaction formula according to SIA 265:2003 (Swiss stan-

> dard). Shear(*y*) and torsion

$$MAX \left[ \frac{\tau_{v,y,d}}{f_{v,d}} \; ; \; \frac{\tau_{tor,d}}{k_{shape} \cdot f_{v,d}} + \left( \frac{\tau_{v,y,d}}{f_{v,d}} \right)^2 \right] \leq 1$$

Shear(z) and torsion

$$MAX\left[\frac{\tau_{v,z,d}}{f_{v,d}} ; \frac{\tau_{tor,d}}{k_{shape} \cdot f_{v,d}} + \left(\frac{\tau_{v,z,d}}{f_{v,d}}\right)^2\right] \le 1$$

where,

 $k_{shape}$  is a factor for the shape of cross-section,

- round cross-section:  $k_{shape} = 1,2$ 

- rectangular cross-section:  $k_{shape} = \min\{1 + 0.15h/b; 2.0\}$ 

In case of curved beams the program checks the tensile stress perpendicular to the grain Moment-Shear from  $M_v$  and  $V_z$  forces. (EN 1995-1-1, 6.4.3.)

Moment(*y*)-Shear(*z*)

$$\frac{\tau_{d}}{f_{v,d}} + \frac{\sigma_{t,90,d}}{k_{dis} \cdot k_{vol} \cdot f_{t,90,d}} \le 1 \text{ where,}$$

 $k_{dis}$  is a factor which takes into account the effect of the stress distribution in the apex zone  $(k_{dis} = 1,4 \text{ for curved beams})$  $k_{vol}$  is a volume factor  $(k_{vol} = [V_0/V]^{0,2})$ 

**Design Parameters** 

For the design based on Eurocode 5, the following design parameters should be defined and assigned to the design members:



Material GL 24c ()	GLILAMD
Over Contine MAT 25	40-40
Cross-Section VKI 25,	40x40
Layer thickness	
	t (cm) =
Crain	
Grain	A - 10
O I op edge is parallel	to the grain
	^ ²
	X
Bottom edge is para	illel to the grain
to Dono <u>m</u> cago to part	nor to the grant
-Buckling Coefficients-	
bucking coefficients	
Flexural Buckling	
K _y = 1,00	✓ K ₂ = 1,00
-I steral Toreional Ruck	lina
Lood position	
Load position	K _{LT} = 1,00
C Center of gravity	
○ <u>B</u> ottom	
Member preferences	
• ·	•
<b>.</b>	
PICKUD 33	

Layer thickness In case of Glued laminated timber (Glulam) arcs thickness of one layer has to be defined.

Grain direction Set of grain direction in case of tapered beam. The grain direction can be paralel with the top edge or with the bottom edge. The top edge lays in the +z direction of the cross-section.

#### **Stability Parameters**

Buckling

 $K_y$ ,  $K_z$ : buckling length factors corresponding to the *y* and *z* axis, respectively.

$$K_y = \frac{l_{ef,y}}{l}; \quad K_z = \frac{l_{ef,z}}{l}$$
 where,

*l* is the member length

 $l_{ef,y}$  and  $l_{ef,z}$  are the buckling length of the member corresponding to the y and z axis.  $(l_{ef,y}$  is the buckling length in *x*-*z* plane of the member)

 $(l_{ef,z}$  is the buckling length in *x*-*y* plane of the member)

Lateral torsional buckling

 $\mathbf{K}_{LT}$ : lateral buckling length factors corresponding to the *z* axis.

$$K_{LT} = \frac{l_{ef}}{l}$$

where,

*l* is the member length

 $l_{ef}$  is the lateral buckling length of the member corresponding to the *z* axis.

When the load not applied to the center of gravity, the program modify the lateral buckling length according to the following:

- if the load is applied to the compression edge of the member the  $l_{ef}$  is incrased by 2h

- if the load is applied to the tension edge of the member the  $l_{ef}$  is decreased by 0,5h

## Informing values of $K_{LT}$ factor. (Some of these values can be found in EN 1995-1-1, Table 6.1)

Loading type (direct load)	$M_y$ moment distribution between the lateral supports	Lateral support condition (in <i>x-y</i> plane) OO
Pz		0,9
F ₄		0,8
$\begin{array}{c c} F_z & F_z \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline & & & \\ \hline \\ \hline$		0,96
pz		0,42
F _z		0,64
Loading type (no direct load)	$M_y$ moment distribution between the lateral supports	Lateral support condition (in <i>x-y</i> plane) 00
Loading type (no direct load)	M _y moment distribution between the lateral supports	Lateral support condition (in <i>x-y</i> plane) 00 1,0
Loading type (no direct load)	M _y moment distribution between the lateral supports	Lateral support condition (in <i>x-y</i> plane) 0 1,0 0,76
Loading type (no direct load) M M M M M M M M M	M _y moment distribution between the lateral supports	Lateral support condition (in <i>x-y</i> plane) 0,76 0,53
Loading type (no direct load) M M M M M M M M M M M M M M M M M M M	M _y moment distribution between the lateral supports	Lateral support condition (in <i>x-y</i> plane) 1,0 0,76 0,53 0,37

Loading type (cantilever)	$M_y$ moment distribution	Lateral support condition (in <i>x-y</i> plane)
p _z		0,5
F ₄		0,8

Design members The design is performed on design members that can consist of one or more finite elements (beams and/or ribs). A group of finite elements can become a design member only if the finite elements in the group satisfy some requirements checked by the program: to be located on the same straight line or arc, to have the same material, cross-section and to have joining local coordinate systems.

The program allows two methods to define design members as follows:

Any node of a selection set of finite elements where another finite element is connected will become an end-point of a design member within the selection set of finite elements.



The finite elements in the selection set become only one design member irrespective of other finite elements connecting to its nodes.



#### Result diagrams

By clicking on a design member the program displays the diagrams corresponding to all the checks.



This page is intentionally left blank.

## 7. AxisVM Viewer and Viewer Expert

AxisVM Viewer

AxisVM Viewer is a freely downloadable version of the program for viewing models without the possibility of making changes. Printing of drawings, tables or reports is not available.

This programs allows a detailed presentation of a model in an environment where AxisVM has not been installed.



If you do not want others to use your work as a basis for their models but you would like to let them see it save the model in an AxisVM Viewer (*.AXV) file format (see *File/Export*). The market version cannot read AXV files but the Viewer can. This format guarantees that your work will be protected.

AxisVM Viewer Expert Owners of the AxisVM market version can buy the Viewer Expert version which lets the user print diagrams, tables and reports or place temporary dimension lines and text boxes. No changes can be saved.

This page is intentionally left blank.

## 8. Programming AxisVM

AxisVM COM server AxisVM like many other Windows application supports Microsoft COM technology making its operations available for external programs. Programs implementing a COM server register their COM classes in the Windows Registry providing interface information.

Any external program can get these descriptions, read object properties or call the functions provided through the interface. A program can launch AxisVM, build models, run calculations and get the results through the AxisVM COM server. This is the best way to

- build and analyse parametric models,
- finding solutions with iterative methods or
- build specific design extension modules.

DLL modules placed in the *Plugins* folder of AxisVM are automatically included in the *Plugins* menu imitating the subfolder structure of the *Plugins* folder. The AxisVM COM server specification and programming examples are downloadable from the AxisVM website, www.axisvm.com.

This page is intentionally left blank.

## 9. Step by Step Input Schemes

## 9.1. Plane Truss Model

Geometry

Elements



Select all nodes to define nodal degrees of freedom. Choose the *Truss girder in X-Z plane* from the list.

#### Loads

1.) Define load cases and combinations.





2.) Apply loads (nodal, thermal, fault in length, dead load).



3.) Select the truss elements, which have the same load.



Static

Start a linear static analysis.

## 9.2. Plane Frame Model

#### Geometry

 Create the geometry (for example: in X-Z). Set the X-Z view.

Draw the geometry.



Elements

1.) Define beam elements.

**⇒** Beam

Select the lines, which have the same cross-section and material, to define beam elements.

2.) Loading material features from the material library

Ø ⇒

1/

Database (Steel FE 430)

- 3.) Selecting cross-section from the database
  - ÎÌ ⊨> Database
- 4.) Define support elements.
   ▲ ▲ Nodal

support

➡ Global

 $\Rightarrow$  Beam relative

(Ø76x7.0)

⇔ Local

	Support[kN/m],[kNm/rad]	
	(Kx=1E+10)	
	(Ky=0.0)	
	(Kz=1E+10)	
	(Kxx=0.0)	
	(KYY=0.0)	
	(Kzz=0.0)	T T
	Z	
4 4		• • • • •
	Y Ž	2 1

Select the nodes, which have the same properties, to define nodal support elements.

5. Define the nodal degrees of freedom.

🍜 🖒 🛛 Nodal DOF

Select all nodes to define degrees of freedom. Choose the Frame in X-Z plane from the list.

Loads

- 1) Define load cases and combinations.
  - ⊥⊥ Load case
     and
     load group



2.) Apply loads (nodal, distributed, temperature, fault in length, dead load).

<b>∲</b> ⇒	Nodal
<u></u> ♣ ⇒	Beam
₫ ⇒	Beam
<b>G</b> ⇒	Beam
	Beam
≙L <del>u</del> ⇔	Beam
<b>+</b> <u>+</u> → ⇒	Beam

3.) Select the beam elements, which have the same load.



Static



## 9.3. Plate Model

Geometry

Elements

Loads

- 1.) Create the geometry (for example: in X-Y plane). Set the X-Y view.
  - →~

Draw the element mesh.



First, select the surface elements, and then select the supported edges, to define line support elements.

If you choose relative to edge support conditions, then the edge will represent the x direction, and the y direction will be perpendicular to the edge in the surface plane (according to the right-hand rule), and the z direction will be perpendicular to the surface plane.

3.) Define the nodal degrees of freedom.

🤹 🖒 🛛 Nodal DOF

Select all nodes to define degrees of freedom. Choose the *Plate in X-Y plane* from the list.

1.) Define load cases and combinations.

Load case and load group



ĮЩ

2.) Apply loads (nodal, line, surface, dead load).



3.) Select domain, which have the same load. The direction of distributed load is perpendicular to the plane of the surface, and the sign of the load is the same as of the local *z* axis of the plate (for example:  $p_z$ =-10.00 kN/m²).



#### Elements

Static

1.) Mesh generation



-select the domain -set the avarage size of finite elements (for example.:0,5 m)



2.) Define the nodal degrees of freedom.



Select all nodes to define degrees of freedom. Choose the *Plate in X-Y plane* from the list.

Start a linear static analysis.

٩t

+U

## 9.4. Membrane Model

Geometry

 Create the geometry (for example: in X-Z plane). Set the X-Z view.

 [↑]Z v

Draw the element mesh.





Elements

1.) Define membrane elements.



#### **⇒** Membrane

Select the quad/triangle surfaces, which have the same material, local directions and thickness, to define the membrane elements.

2.) Define material features (for example: selecting from the material library)

Loading (Concrete C20/25) Concrete C20/25)

- 3.) Define the thickness ( for example: 200 mm)
- 4.) The program automatically generates the local coordinate-system of finite elements



 $n_x$ ,  $n_y$ ,  $n_{xy}$  internal forces refer to the local x,y directions



5.) Define support elements.



□□□□ Line support □→ Edge relative

⇔ Global

Tou can also define surface supports (Winkler type elastic foundation).

First, select the surface elements, and then select the supported edges, to define line support elements.

If you choose relative to edge support conditions, then the edge will represent the x direction, and the y direction will be perpendicular to the edge in the surface plane (according to the right-hand rule), and the z direction will be perpendicular to the surface plane.

6.) Define the nodal degrees of freedom.

Select all nodes to define degrees of freedom. Choose the *Membrane in X-Z plane* from the list.

Loads

- 1.) Define load cases and combinations.
  - ⊥Load case and load group
  - 🚆 🕁 Combination
- 2.) Apply loads (nodal, line, surface, dead load).
  - Image: A constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constraint of the constra

Select the elements, which have the same load.

The direction of distributed load is determined in the local *x*-*y* direction of the membrane (for example:  $p_y = -10.00 \text{ kN/m}^2$ ).





Start a linear static analysis.



## 9.5. Response Spectrum Analysis

Geometry

Elements

See... 9.1.- 7.4. Input Schemes.

Loads/1

See... 9.1 - 7.4. Input Schemes..

1.) Apply loads.

👑 🖒 Load

2.) Apply all the gravitational loads that you want to account as masses in the vibration analysis that precedes the static analysis.

Analysis/1

°<del>f∖∵</del>t

1.) Perform a vibration analysis. Vibration mode shapes for earthquake analysis are usually requested as 3 for in-plane structures and 9 for spatial structures are requested.

Include the gravitational load case described at Loads/1 point in the vibration analysis, and set the Convert loads to mass check-box enabled.



Loads/2

1.) Set a seismic load case.



2.) Specify the parameters of the seismic loads.





- 1.) Start a linear static analysis.
- 2.) When generating the seismic type load cases, two are included. One + with values included as positives, and one with values included as negatives. In addition the results corresponding to each vibration mode shape are provided (corresponding to load cases with 01, 02, ...,n suffixes), that can be used in the generation of further combinations or of critical combinations.

See... 4.10.20 Seismic Loads





## 10. Examples

## 10.1. Linear Static Analysis of a Steel Plane Frame

Input data

AK-ST-I.axs

Geometry:



Loads:



Results

AK-ST-I.axe

Cor	mponent	Analytic	AxisVM
1 Lc.	$e_{\chi}^{(C)}$ [mm]	17.51	17.51
	$M_{y}^{(A)}$ [kNm]	-20.52	-20.52
2 Lc.	$e_{X}^{(C)}$ [mm]	7.91	7.91
	$M_y^{(A)}$ [kNm]	63.09	63.09

## 10.2. Geometric Nonlinear Static Analysis of a Steel Plane Frame

Input data AK-ST-II.axs



Results

AK-ST-II.axe

C	omponent	With Stability Functions	AxisVM
1 Lc.	e ^(C) _X [mm]	20.72	20.58
	$M_{y}^{(A)}$ [kNm]	-23.47	-23.41
2 Lc.	$e_{X}^{(C)}$ [mm]	9.26	9.22
	$M_{y}^{(A)}$ [kNm]	66.13	66.25

Verify

The equilibrium must be verified taking into account the deflections.
## 10.3. Buckling Analysis of a Steel Plane Frame

Input data

```
AK-KI.axs
```

Geometry and loads:



AK-KI.axe



# 10.4. Vibration Analysis (I-Order) of a Steel Plane Frame

Input data AK-RZ-I.axs



AK-RZ-I.axe

	Frequency [Hz]	
Mode	Cosmos/M [™]	AxisVM
1	6.957	6.957
2	27.353	27.353
3	44.692	44.692
4	48.094	48.094
5	95.714	95.714
6	118.544	118.544

## 10.5. Vibration Analysis (II-Order) of a Steel Plane Frame

Input data

AK-RZ-II.axs

Geometry and loads:



AK-RZ-II.axe

	Frequency [Hz]	
Mode	Cosmos/M [™]	AxisVM
1	0.514	0.514
2	11.427	11.426
3	12.768	12.766
4	17.146	17.145
5	27.112	27.109
6	39.461	39.456

# 10.6. Linear Static Analysis of a Reinforced Concrete Cantilever

## Input data

### VT1-ST-I.axs



VT1-ST-I.axe

Component	Beam theory (shear deformations included)	AxisVM
<i>e</i> ( ^{<i>B</i>)} [mm]	15.09	15.09
$n_x^{(A)}$ [kN/m]	1800.00	1799.86

### **10.7.** Linear Static Analysis of a Simply Supported Reinforced Concrete Plate

### Input data VL1-ST-I.axs



Results

Convergence

analysis

Component	Analytic	AxisVM
-	(shear deformations	(shear deformations
	not included)	included)
e ^(A) [mm]	51.46	51.46
$m_x^{(A)}$ [kNm/m]	46.11	46.31



Meshes:



# 10.8. Linear Static Analysis of a Clamped Reinforced Concrete Plate



#### Results

VL2-ST-I.axe

Component	Analytic (shear deformations not included)	AxisVM (shear deformations included)
$e_z^{(A)}$ [mm]	16.00	16.18
$m_x^{(A)}$ [kNm/m]	22.01	22.15
$m_x^{(B)}$ [kNm/m]	64.43	63.25
$q_x^{(B)}$ [kN/m]	111.61	109.35

#### Convergence analysis



## 11. References

- 1. Bathe, K. J., Wilson, E. L., Numerical Methods in Finite Element Analysis, Prentice Hall, New Jersey, 1976
- 2. Bojtár I., Vörös G., A végeselem-módszer alkalmazása lemez- és héjszerkezetekre, Műszaki Könyvkiadó, Budapest, 1986
- 3. Chen, W. F., Lui, E. M., Structural Stability, Elsevier Science Publishing Co., Inc., New York, 1987
- 4. Hughes, T. J. R., The Finite Element Method, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1987
- 5. Owen D. R. J., Hinton E., Finite Elements in Plasticity, Pineridge Press Limited, Swansea, 1980
- 6. Popper Gy., Csizmás F., Numerikus módszerek mérnököknek, Akadémiai Kiadó · Typotex, Budapest, 1993
- 7. Przemieniecki, J. S., Theory of Matrix Structural Analysis, McGraw Hill Book Co., New York, 1968
- 8. Weaver Jr., W., Johnston, P. R., *Finite Elements for Structural Analysis*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1984
- 9. Dr. Szalai Kálmán, Vasbetonszerkezetek, vasbeton-szilárdságtan, Tankönyvkiadó, Budapest, 1990. 1998
- 10. Dr. Kollár László: Vasbeton-szilárdságtan, Műegyetemi Kiadó, 1995
- 11. **Dr. Kollár László:** Vasbetonszerkezetek I., Vasbeton-szilárdságtan az Eurocode 2 szerint, Műegyetemi Kiadó, 1997
- 12. Dr. Bölcskei E., Dr. Dulácska E.: Statikusok könyve, Műszaki Könyvkiadó, 1974
- 13. Dr. Dulácska Endre: Kisokos, Segédlet tartószerkezetek tervezéséhez, BME Építészmérnöki Kar, 1993
- 14. Porteous, J., Kermani, A., Structural Timber Design to Eurocode 5, Blackwell Publishing, 2007
- 15. Dulácska Endre, Joó Attila, Kollár László: Tartószerkezetek tervezése földrengési hatásokra, Akadémiai Kiadó, 2008
- 16. Pilkey, W. D., Analysis and Design of Elastic Beams Computational methods, John Wiley & sons, Inc., 2002
- 17. Navrátil, J., Prestressed Concrete Structures, Akademické Nakladatelství Cerm[®], 2006
- 18. Szepesházi Róbert: Geotechnikai tervezés (Tervezés Eurocode 7 és a kapcsolódó európai geotechnikai szabványok alapján), Business Media Magyarország Kft., 2008
- 19. Györgyi József: Dinamika, Műegyetemi Kiadó, 2003
- 20. Bojtár Imre, Gáspár Zsolt: Végeselemmódszer építőmérnököknek, Terc Kft., 2003
- 21. Eurocode 2, EN 1992-1-1:2004
- 22. Eurocode 3, EN 1993-1-1:2005
- 23. Eurocode 3, EN 1993-1-3:2006
- 24. Eurocode 3, EN 1993-1-5:2006
- 25. Eurocode 5, EN 1995-1-1:2004
- 26. Eurocode 8, EN 1998-1-1:2004
- 27. Paz, M., Leigh, W., Structural Dynamics Theory and Computation, Fifth Edition, Springer, 2004
- 28. Chopra, A. K., Dynamics of Structures Theory and Applications to Earthquake Engineering, Third Edition, Pearson Prentice Hill, 2007
- 29. Biggs, J. M., Introduction to Structural Dynamics, McGraw-Hill, 1964
- 30. Weaver, W., Jr., P. R. Johnston, Structural Dynamics by Finite Elements, Prentice-Hall, 1987
- 31. Bathe, K. J., Finite Element Procedures, Prentice-Hall, 1996

Notes

Notes

Notes