Autodesk Robot Structural Analysis

Metric Getting Started Guide

Autodesk



This manual text is printed on 40 percent postconsumer waste recycled paper © 2009 Autodesk, Inc. All Rights Reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., in the USA and other countries: 3DEC (design/logo), 3December, 3December, com, 3ds Max, ActiveShapes, Actrix, ADI, Alias, Alias (swirl design/logo), AliasStudio, Alias[Wavefront [design/logo], ATC, AUG], AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Envision, AutoCAD SQL Literia, Autodesk Insight, Autodesk Intent, Autodesk Inventor, Autodesk MapGuide, Autodesk Streamline, AutoLSP, AutoShetch, AutoTrack, Backdraft, Built with ObjectARX [logo], Burn, Buzzsaw, CAiCE, Can You Imagine, Character Studio, Cinestream, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Create>what's>Next> [design/logo], Dancing Baby [image], DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignStudio, DesignStudio, Cleasin, Leado, FrueView, DXF, EditDV, Education by Design, Our World, Design Tour World (design/logo), DWF, DWG [logo], DWG TrueView, DXF, EditDV, Education by Design, Exposure, Extending the Design Team, FBX, Filmbox, FMDesktop, Freewheel, GDX Driver, Opticore, Opus, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProjectPoint, ProMaterials, Reactor, RealDWG, Real-time Roto, Reconjez, Render Queue, Reveal, Revit, Showcase, ShowMotion, SketchBook, SteeringWheels, StudioTools, Topobase, Toxik, Visual Bridge, Visual Construction, Visual Drainage, Visual Hydro, Visual Landscape, Visual Roads, Visual Survey, Visual Syllabus, Visual Survey, Visual Syllabus, Visual Toolbox, Visual Tugboat, Visual Londst, Visual Londst, Visual Londst, Visual Syllabus, Visual Toolbox, Visual Tugboat, Visual Londst, Visual Syllabus, Visual Tugboat, Visual Londst, Visual Londst, Visual Syllabus, Visual Tugboat, Visual Syllabus, Visual Tugboat, Visual Construction, Visual Drainage, Nisual Hydro, Visual Landscape, Visual Roads, Visual Syllabus, Visual Tugboat

The following are registered trademarks or trademarks of Autodesk Canada Co. in the USA and/or Canada and other countries: Backburner, Discreet, Fire, Flame, Flint, Frost, Inferno, Multi-Master Editing, River, Smoke, Sparks, Stone, and Wire

All other brand names, product names or trademarks belong to their respective holders.

Third Party Software Program Credits

ACIS Copyright© 1989-2001 Spatial Corp. Portions Copyright© 2002 Autodesk, Inc.

Flash ® is a registered trademark of Macromedia, Inc. in the United States and/or other countries.

International CorrectSpell™ Spelling Correction System© 1995 by Lernout & Hauspie Speech Products, N.V. All rights reserved.

InstallShield™ 3.0. Copyright© 1997 InstallShield Software Corporation. All rights reserved.

PANTONE® Colors displayed in the software application or in the user documentation may not match PANTONE-identified standards. Consult current PANTONE Color Publications for accurate color. PANTONE Color Data and/or Software shall not be copied onto another disk or into memory unless as part of the execution of this Autodesk software product.

Portions Copyright© 1991-1996 Arthur D. Applegate. All rights reserved.

Portions of this software are based on the work of the Independent JPEG Group.

RAL DESIGN© RAL, Sankt Augustin, 2002

RAL CLASSIC© RAL, Sankt Augustin, 2002

Representation of the RAL Colors is done with the approval of RAL Deutsches Institut für Gütesicherung und Kennzeichnung e.V. (RAL German Institute for Quality Assurance and Certification, re. Assoc.), D-53757 Sankt Augustin.

Typefaces from the Bitstream® typeface library copyright 1992.

Typefaces from Payne Loving Trust© 1996. All rights reserved.

Printed manual and help produced with Idiom WorldServer™.

WindowBlinds: DirectSkin™ OCX © Stardock®

AnswerWorks 4.0 ©; 1997-2003 WexTech Systems, Inc. Portions of this software © Vantage-Knexys. All rights reserved.

The Director General of the Geographic Survey Institute has issued the approval for the coordinates exchange numbered TKY2JGD for Japan Geodetic Datum 2000, also known as technical information No H1-N0.2 of the Geographic Survey Institute, to be installed and used within this software product (Approval No.: 646 issued by GSI, April 8, 2002).

Portions of this computer program are copyright © 1995-1999 LizardTech, Inc. All rights reserved. MrSID is protected by U.S. Patent No.

5,710,835. Foreign Patents Pending.

Portions of this computer program are Copyright ©; 2000 Earth Resource Mapping, Inc.

OSTN97 © Crown Copyright 1997. All rights reserved.

OSTN02 © Crown copyright 2002. All rights reserved.

OSGM02 © Crown copyright 2002, © Ordnance Survey Ireland, 2002.

FME Objects Engine © 2005 SAFE Software. All rights reserved.

AutoCAD 2009 is produced under a license of data derived from DIC Color Guide® from Dainippon Ink and Chemicals, Inc. Copyright © Dainippon Ink and Chemicals, Inc. All rights reserved.

Government Use

Use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in FAR 12.212 (Commercial Computer Software-Restricted Rights) and DFAR 227.7202 (Rights in Technical Data and Computer Software), as applicable.

i

TABLE OF CONTENTS

AUTODESK ROBOT STRUCTURAL ANALYSIS

FAST OVERVIEW1
General description of the program3
Robot modules
Robot screen layout5
Basic configuration of the program
Preferences
Job Preferences7
Navigation techniques
Methods of working with Robot interface 10
System menu
 View menu
 Geometry menu12
 Loads menu12
 Analysis menu
 Results menu
 Tools menu
 Window menu14
 Help menu14
Layout System14
Entering the structural analysis data17
Analyzing the structure
Results preview
 Graphical results for beams
 Graphical results for surfaces
 Tabular results
Reports and printout composition27
Advanced analysis and design
List of shortcuts

3D FRAME STRUCTURE	31
Configuration of the program Model Definition	
 Bars definition (frame 2D) 	
 Supports definition 	
 2-bay frame definition 	
 Load case definition 	
 Loads definition for particular load cases 	
 Copying existing frame 	
 Definition of lateral beam 	43
 Definition of cross bracings 	44
 Copying defined bars (lateral beam and bracings) 	46
Structure Analysis	47
Results Preview	48
 Displaying beam results graphically 	48
 Displaying results on bars in tabular form 	50
 Stress analysis 	51
Preparation of printouts	54
 "Capturing" views and data for the calculation note 	
 Preparing printout composition 	
 Printing and exporting the calculation report 	57

RC AND STEEL MIXED STRUCTURE 61

Config	uration of the program	63
Model	definition	64
•	Definition of structural axes	64
	Section definition	67
	Bars definition	70
	Supports definition	76
	Load cases definition	77
	Definition of loads for predefined load cases	79
	Changing the structure type	89
•	Definition of additional structural axis	89

 Copying existing frame 	
 Definition of lateral beams 	
 Definition of slab 	
Offset definition	
 Definition of a wall 	
 Definition of wall support 	
 Meshing parameters definition 	110
 Definition of slab loads 	
Structure Analysis	115
Results Preview	
 Panels results in map form 	
 Deformation of the structure 	
 Results on panels in tabular form 	

Integration of Autodesk Robot Structural Analysis with

Revit® Structure	
------------------	--

Export Revit model to Robot	131
 Opening project in Revit[®] Structure 	131
 Sending data to Robot 	132
Structure Analysis in Robot	141
 Displaying items on the screen 	142
 Presentation of load cases transferred from Revit[®] Structure 	143
 Meshing parameters definition 	145
Calculations	148
 Results preview - displaying panel results in map form 	
 Results preview - displaying results on bars in diagram form 	152
Modification of the Structure in Robot	155
 Replacing sections 	155
 Deleting bars 	
 Adding new elements 	159
Update Revit Model from Robot	164
 Updating Revit[®] Structure project 	164
 Model changes presentation 	172

AUTODESK ROBOT STRUCTURAL ANALYSIS FAST OVERVIEW

Synopsis:

The purpose of this Manual is introduce the novice user to the Autodesk Robot Structural Analysis system and to provide some guidance into the program configuration, menu system and navigation techniques. It will also show the many and varied methods of input of data and extraction of results.

It is assumed that Autodesk Robot Structural Analysis 2010 is installed on the PC.

GENERAL DESCRIPTION OF THE PROGRAM

What is Autodesk Robot Structural Analysis?

Autodesk Robot Structural Analysis (Robot) is a single integrated program used for modeling, analyzing and designing various types of structures. The program allows users to create structures, to carry out structural analysis, to verify obtained results, to perform code check calculations of structural members and to prepare documentation for a calculated and designed structure.

Robot - key features of the Commercial version are shown below:

- Linear, nonlinear and dynamic (modal, spectral, seismic, time history, push over, P-Delta, buckling deformation, plasticity) structure analysis
- working in a multilingual environment (15 languages independently set to user interface, designing and calculation notes)
- working in a multinational environment designing according to over 50 design codes
- frames, plates and shells, plus a powerful GUI modeler and mesher allows the user to define virtually any shape or configuration of structure – you analyze the true structure geometry
- quality bi-directional integration with Revit[®] Structure, plus integration through IFC, CIS2 etc.
- an open API to allow the user to interface their own applications for pre and/or post processing

ROBOT MODULES

Robot is a single product with many functions and a common user enviroment.

Once Robot is activated (click on the appropriate icon on the desktop or choose the appropriate command from the taskbar), the window shown below (Figure 2.1) will appear on the screen. The window is used to select the type of structure that will be analyzed or to load an existing structure.

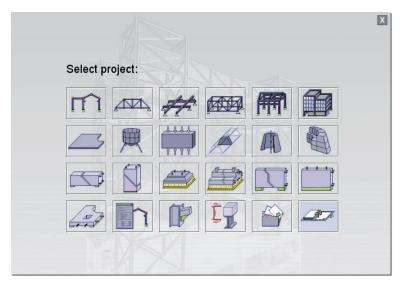


Figure 2.1 - Robot modules window

NOTE: When the cursor is positioned on an icon, a short description of its function is displayed.

The most commonly used icons are described below:

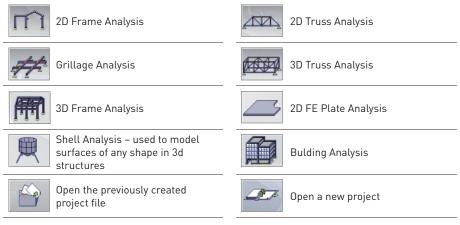


Table 2.1 - Robot basic modules

4

ROBOT SCREEN LAYOUT

STANDARD TOOLBAR

- these options are mainly associated with non structural items, such as print, save, undo etc.

STRUCTURE MODEL TOOLBAR - sections, nodes, members, supports, loads etc.

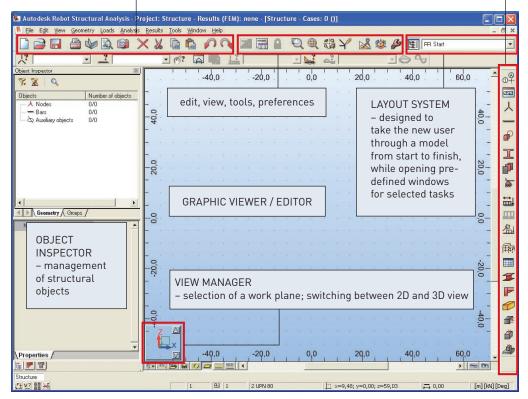


Figure 3.1 - Robot typical screen layout

BASIC CONFIGURATION OF THE PROGRAM

The two options, **Preferences** and **Job Preferences**, allowing the user to set program parameters in the Robot system, are available from the menu by opening toolbar **Tools** and pressing the appropriate icon:

ne - [Structure]	Tools icon		
w <u>H</u> elp]	- @ ×
🔎 🕥 🖬 🔚 🔍 🍭 🏭 🌱	2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	R Start	-
	194		
30,0 -20,0 -10,0 0,0 10,0	20,0 30,0	40,0	
Tools			
👯 0º 🎝 🖬 🗁 🕼		*** >*< **;	3 人
		1. A 1.	0
Preferenc	es icon Jo	b Preference	es icon

Figure 4.1 - Tools toolbar

Preferences

The **Preferences** dialog box presented below is used to define basic parameters in the program:

a Preferences			? ×	
🗃 🖩 🗙 🔆 STANDAR	ID			
Languages General Parameters View Parameters Desktop Settings Toolbar & Menu Printout Parameters	<u>R</u> egional settings:	Great Britain		REGIONAL SETTINGS adjust the
Authorization Key Advanced	Working language: Printout language:	English 💌		databases (profiles, materials), units and codes to the standards of
I I Update Preferences on exit		Accept Cancel	Help	a country
Robot is a multilingua languages for input a	1 0	ser can independently set di red	fferent	

Figure 4.2 - Preferences menu

The most regularly used options are:

- languages selection of regional settings (definition of the country whose codes materials and regulations - e.g. code combination regulations - will be used during the design process, calculations and structure design) and working and printout language
- general parameters (saving parameters, number of recently used structures, sound on/off etc.)
- display parameters (colors and fonts for screen components)
- toolbar and menu (menu type and the type of toolbars)
- printout parameters (colors and fonts for printouts, scale and symbols, line thickness)
- protection parameters (protection, authorization) for changing the system protection
- COM interface presentation of the registered additional programs/modules

Job Preferences

The **Job Preferences** dialog box, presented below, allows you to define general program parameters to be used in a given project:

🔜 Job Preferences			? ×
🖻 🔚 🗙 送 🛛	EFAULTS		•
Units and Formats Dimensions Forces Unit Edition Materials Databases Design codes Work Parameters	Eorce: Moment: Stresses:	kN ▼ 0.21 kN*m … 0.21 MN/m2 … 0.21	4 > E 4 > E
🙀 <u>O</u> pen defa	ult parameters		
Save current pa	arameters as default	OK Cano	el Help

Figure 4.3 - Job Preferences menu

The most important functions are:

- number units and formats (dimensions, forces, possibility of unit edition)
- materials (selection of material set, according to the country and the possibility of creating user-defined material)
- section database (selection of the appropriate database with member sections)
- structure analysis parameters (selection of the static analysis method and definition of basic parameters for dynamic and non-linear analysis; selection of analysis types, possibility of saving results for seismic analysis combination of seismic cases)
- parameters for generation of surface finite element meshes for plates and shells

NAVIGATION TECHNIQUES

In the Robot software, various mechanisms have been introduced to make structure definition simple and more efficient. According to the type of operation performed, the mouse cursor changes its shape to:

- "hand" a selection mode for highlighting entities
- "cross pointer" during node and bar definition, to define precise points e.g. start and end points of members
- shape of the appropriate feature e.g. when adding sections the cursor is in shape of an I section, when adding supports a support icon appears etc.

The cursor operation in a viewer by means of the third mouse button (or wheel) is identical to that in the AutoCAD[®] program. The following cursor support modes are available:

- wheel rotation zoom in / out
- wheel rotation + Ctrl key horizontal pan
- wheel rotation + Shift key vertical pan
- pressing the third button pan
- double-click with the third button initial view

The user should take note of the work capabilities in 3D views when the menu option **Dynamic View** (View \rightarrow Dynamic View \rightarrow Dynamic View) is switched on. 3D viewing enables work in one of five modes:

- four simple modes: 3D rotation, 2D rotation, zoom and pan
- one multi-function mode

The user may switch from one work mode to another by selecting an appropriate option in the View / Dynamic View menu, on the View toolbar and in the context menu. After choosing a work mode, the mouse cursor movement (with mouse left button pressed) brings about the relevant change in the 3D view:

- 3D Rotation rotates a structure in all planes
- 2D Rotation rotates a structure in the plane parallel to the screen plane
- Zoom movement down the view zooming in / zooming out a structure to / from the screen plane
- Pan movement in the view plane (structure shift with respect to the screen center)

The multi-function mode (Rotation / Zoom / Pan) enables work using all the modes at the same time. The viewer of 3D view is divided into quarters and each of them is ascribed one of the modes:



Table 5.1 - Cursor modes

Once the cursor is positioned in the relevant quarter of the screen, the cursor shape changes (see the icons above).

Navigation tool (ViewCube) is also available in the program. ViewCube is a 3D interactive tool that lets you reorient a view and set the current orientation of a structure model. Clicking a predefined face, edge or corner on the ViewCube, you can reorient the view of a model. Moreover, clicking and dragging the ViewCube lets you reorient the model to different directions. Access the ViewCube option by selecting the ViewCube - Properties.



Figure 5.1 - ViewCube

The ViewCube also uses the compass to indicate a direction from which you view a model. To change the viewpoint of a model, click a selected direction on the compass (N, S, E, W). You can show or hide the compass from the ViewCube context menu after you right-click on the ViewCube and select the Show compass option.

METHODS OF WORKING WITH ROBOT INTERFACE

There are two methods to work with Robot - by using System Menu to entering data, or special Layout System.

System menu

The system menu consists of two parts: a text menu and toolbars with appropriate icons. They can be used interchangeably, according to the users' needs and preferences. Both are displayed in the same way - as a horizontal bar at the top of the screen (additionally, for some layouts in the Robot system, another toolbar with most frequently used icons is displayed on the right side of the screen).

Basic options available within the modules are accessible both from the text menu and the toolbar.

Though contents of the text menu and toolbars for successive modules vary, the main options are always available regardless of which module is active.

The figure below illustrates both types of menus (the main menu that appears once the Start layout is selected is shown as an example):

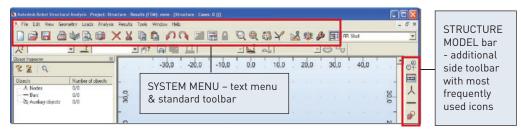


Figure 6.1 - System menu

Options available in the text menu are grouped as follows:

<u>File menu</u>

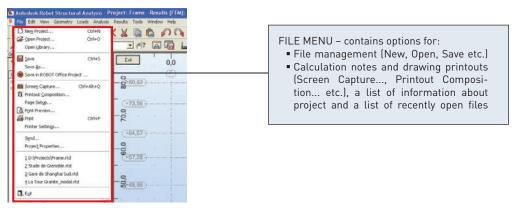


Figure 6.2 - File menu window

Edit menu

EDIT MENU – contains options for:

- Elements edition (Undo, Redo, Cut, Copy etc.)
- Selection (Select...,Select All)
- Selection using filters (Select Special)
 Model modification (Edit, Complex)
- Edit..., Divide, Intersect etc.)

D Undo Ctrl+: XXDDDO (+ Redo Ctrl+Y · 6? A . L 17 X cue Chiefe Copy Ctrl+C Exit 108 12 0,0 Y C Paste Ctrl+V X Clear Delete Create OLE Type Link Select. Select All 01544 Previous Selection 2+80,02 -Select Special Edit (+73,56) Complex Edit Substructure Modification +84.57 + Divide. 1 Intersect 8+57,28 Trim... Extend... > Corgect ... Detailed Correct. +48.95-Correct Drawing Model.

Project: Structure - Results (FE

Autodesk Robot Structural Analy

Edt N

Figure 6.3 - Edit menu window

Autodesk Robot Structural Analysis - Getting Started Guide

View menu

A ^o Bedraw	X B B P R
Dynazyc Verv Zoom Zan Projection	
bjects Rotation	•
Nodes Bass Work in 20	•
Bar Bar Bar Bar Bar Grid	· - 3 *80.02)
Bar Bar Isbles	80
Bar Bar History	(+73,56)
Re C	

Figure 6.4 - View menu window

Geometry menu

VIEW MENU – contains options for:

- 2D/3D view management of a structure's model (Dynamic View, Zoom, Pan etc, Work in 3D)
- Structure attributes to be presented on the screen (Display); definition of grid parameters (Grid)
- Selection of tables with data or results (Tables) and saving defined views of a structure (History)

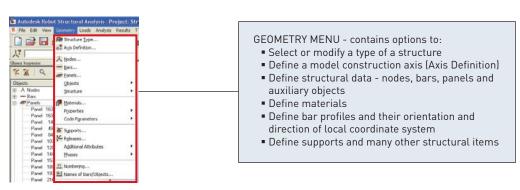


Figure 6.5 - Geometry menu window

Loads menu

LOADS MENU – contains options to define lo	ad
cases and combinations	

File Edit View Geom	etry Loads Analysis Results Tools V	Window He
Chyect Inspector		
Objects	Load Table Nun Combination Table D 11 Mass Table	0. (F
Bar 1 Bar 2 Bar 3 Bar 4 Bar 5	Select Cases Select Cases Component Select Modes Select Result Type	
- Bar 6 - Bar 7	Special Lgads	

Figure 6.6 - Loads menu window

Analysis menu

Autologikki fölipid Structural Analysis. Project: Structura - Results (EAB): ner rei 6. dk. Hove Commy: Loadi Condexe Sexalka Tools Window Help rei 6. dk. Hove Commy: Loadi Condexe Sexalka Tools Window Help rei 6. dk. Hove Condexe Sexalka Tools Window Help rei 6. dk. Hove Condexe Sexalka Tools Window Help rei 6. dk. Hove Condexe Sexalka Tools Window Help rei 6. dk. Hove Condexe Sexalka Tools Window Help rei 6. dk. Hove Condexe Sexalka Tools Window Help Rev 1 Rev 1 Rev 1 Rev 1 Rev 3	 ANALYSIS MENU – contains options to: Start the calculation process Change from linear to non linear, P-Delta or buckling Set up dynamic analyses Design concrete beams, columns and surfaces
--	--

Figure 6.7 - Analysis menu window

Results menu

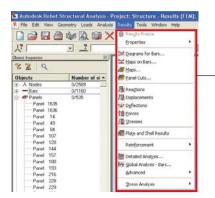


Figure 6.8 - Results menu window

<u>Tools menu</u>

TOOLS MENU – contains different types of configuration options (Preferences, Job Preferences...) and possibility to define user's interface, menu, shortcuts and form of calculation notes

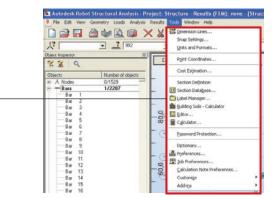


Figure 6.9 - Tools menu window

RESULTS MENU – contains options to:

- Display beam results graphically
- Display detailed results for beams graphically
- Display results for surfaces
- Display tables that easily be edited, sorted, filtered, exported to MS Excel[®] etc.

Window menu

Autodesk Robel Structural Analysis. Project: Structure Results (FLM): none. [Structure Autodesk Robel Structural Analysis. Project: Structure New Autodesk Robel Structure New Autode	
Rest 1/2287 6.4 The spectral Base 1/2287 6.4 0.5 0.5 Base 1/2287 6.4 0.5 0.5 Base 1/2287 6.4 0.5 0.5 Base 1.2287 6.4 0.5 0.5	WINDOW MENU – offers options to ma- nage and arrange the graphic windows and an option to activate/deactivate the "Object Inspector"

Figure 6.10 - Window menu

<u>Help menu</u>

🕒 Autodesk Robot Structural Analysis - Project: Structure - Results (FEM): none - [Structure - (🕅 Elle Egit Yew Geometry Loads Analysis Results Tools Wind W Help	
Image: Constraint of the part of the	HELP MENU – contains help options and various product information items

Figure 6.11 - Help menu window

Layout System

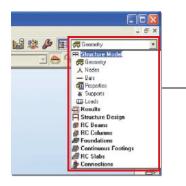
The second method of work with Robot is by using the special layout system.

Robot has been equipped with a layout mechanism that simplifies the design process. The layouts in Robot are specially designed systems of dialog boxes, viewers and tables that are used to perform specific defined operations.

Layouts available in Robot were created to make consecutive operations leading to defining, calculating, and designing the structure easier – the layouts guide the user through the process from model generation to results.

In order to make the system as easy to use as possible, each layout has its own predefined set of windows which are automatically opened on entering the layout and closed on exit.

Layouts are available from the selection list found in the right, upper corner of the screen:



SYSTEM LAYOUT SELECTION WINDOW A list of standard layouts available in Robot. This example shows the layouts for a shell structure and the layouis vary depending on the structure type

Figure 6.12 - System layout selection window

The layout order and arrangement follows a chronological process, starting from defining nodes, beams, then supports, sections etc.

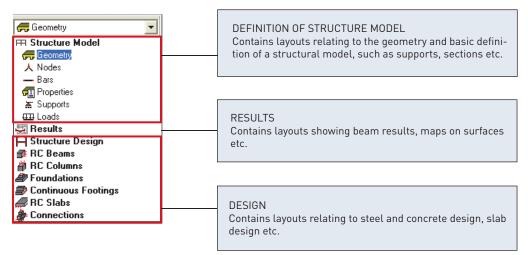
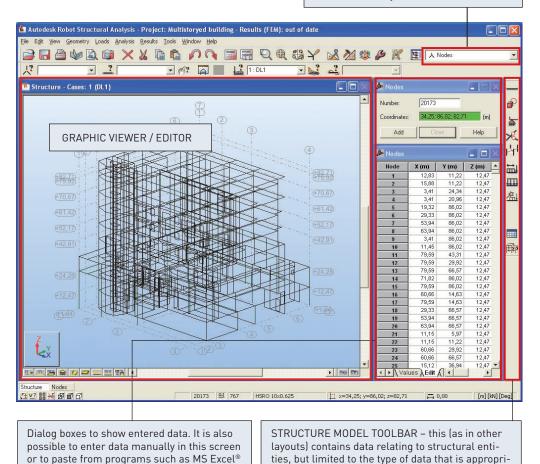


Figure 6.13 - System layout selection window - chronological layout order

A typical layout for nodes is shown – note that each window cannot be closed until a new layout is selected.



LAYOUT SELECTION PULL DOWN MENU - in this case layout Nodes is selected

Figure 6.14 - Typical layout for nodes

However it is not necessary to define the structure according to the layout order. This may be done in any order chosen by the user. The layout system was introduced in such a way that Robot structure definition is intuitive and efficient.

ate to node definition

All Robot operations may be performed without using the defined layouts but by using system menu instead or also taking advantage of both methods (simultaneously) according to the user's needs and preferences.

ENTERING THE STRUCTURAL ANALYSIS DATA

There are 3 ways to enter data:

- 1. By entering data using the appropriate text dialog box or direct in a table (or pasted from MS Excel[®])
- 2. By entering data in the Robot GUI using tools for graphic structure definition i.e. snap grids or structural axes.

Examples of data entering using the System Layout mechanism (for Bars and Loads options) are presented below:

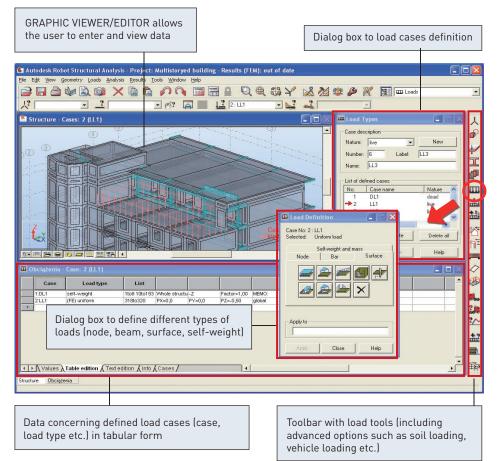


Figure 7.1 - Examples of loads data entering with System Layout mechanism

Ν	0	ΓE	
	-	_	-

The updating of data is dynamic – tables reflect graphics and vice versa at all times.

3. By entering data in another application and importing into Robot - several file formats are supported including:

DXF, DWG, IFC, SDNF (steel detailing neutral file), CIS/2. A dynamic link to and from Revit[®] Structure also provides bi-directional integration.

As the preparation of the structural model progresses, the user can control exactly what is seen on the screen by using the **Display** settings (it is available by pressing 📰 icon from context menu or direct from the bottom, right corner on the Robot screen):

Display Template: Standard	J 🖻		× ×
Favorites Model Nodes Bars Panels / FE Mark with colors Loads View (Open-GL version) View (version without Open Structure	Name Node numbers Bar description Panel description Supports Section - shape Local systems Releases Offsets Load symbols		
Symbol size: 30	Load values Structural axes Display attributes of selected objects OK Cancel	- [200y

DISPLAY allows the user to switch on and off a wide selection of items, including node and members numbers, section shapes, supports, FE mesh and also sets up hidden line and render options. A wide range of customization options allows to the user to define a view of structure as required exactly to user's preferences.

Figure 7.2 - Display window

After defining the model data, the user now proceeds to the analysis stage. However, prior to this stage, the model must be "discretized" into finite elements. Robot has wide ranging capabilities for "automeshing" the structure and meshing is generally a very fast process for even the largest of models. Some of the "meshing parameters are shown below:

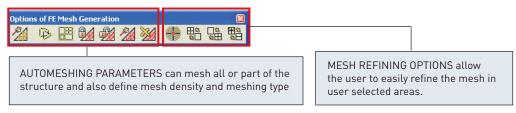


Figure 7.3 - FE Mesh Generation toolbar

Example of structure with meshed panels is shown below:

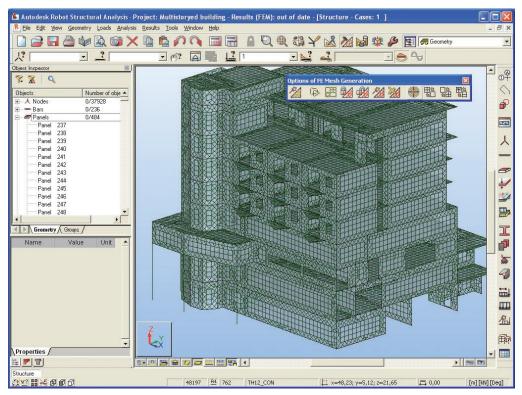


Figure 7.4 - Example of meshing

ANALYZING THE STRUCTURE

Structural analysis can be started by selecting one of the two "calculation" buttons in the horizontal toolbar.

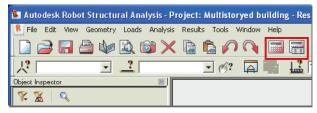


Figure 8.1 - Calculation icons

The icon on the left side starts the calculation process. The second icon is used to set different analysis parameters. This option allows the user to change specific analysis types from linear to non linear or to set up dynamic analysis.

No.	Name	Analysis Type	Case: DEAD LOAD
▶ 1	DEAD LOAD	Static - Linear	Previously: Static - Linear
2	LIVE LOAD	Static - Linear	r reviously. State - Einear
3	HORIZONTAL LIVE LOAD	Static - Linear	New analysis type
4	HORIZONTAL LIVE LOAD X DIRE	Static - Linear	⊙ <u>S</u> tatics
5	ULTIMATE WIND Y DIRECTION	Static - Linear	O <u>B</u> uckling
6	ULTIMATE WIND Y DIRECTION	Static - Linear	○ <u>H</u> armonic analysis
7	ULTIMATE WIND -Y DIRECTION	Static - Linear	Modal taking into account static forces
8	ULTIMATE WIND -Y DIRECTION	Static - Linear	
9	ULTIMATE WIND X DIRECTION	Static - Linear	
10	ULTIMATE WIND -X DIRECTION	Static - Linear	O Push over
11	WIND SERVICE Y DIRECTION UP	Static - Linear	O Harmonic in the frequency domain (FRF)
12	WIND SERVICE Y DIRECTION D	Static - Linear	
<	1111	3	OK Cancel Help
New	Parameters Char	nge analysis type Delete	
Operations	on selection of cases		
Case list			
	arameters Change analysi	s type Delete	

Figure 8.2 - Changing analysis parameters

As a default, all analysis is set as "linear static", unless the user has included some members or parameters such as cables, tension only, hinges etc. In such a case the analysis type is automatically changed to "non linear" and Robot will apply the loads incrementally to ensure the true structural equilibrium and a final exact geometry is reached. Robot has many non linear parameters that the user can set in the case of non convergence of analysis data, including the options to set full or modified Newton-Raphson and "step halving". In addition to general non linear calculations, the user can also set the following analysis options:

- P-Delta
- Modal
- Seismic (to codes such as IBC, UBC, EC etc.)
- Spectral response
- Time history
- Buckling

RESULTS PREVIEW

The Results Layout shows a wide range of analysis results presentation.

NOTE:

All of results preview options are also available through the RESULTS pull-down menu.

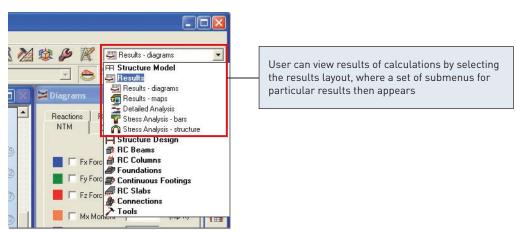


Figure 9.1 - Results type selection from results layout

Results can be split into various categories, each having their own characteristics.

Graphical results for beams

(graphical results for individual beams, or selection of beams, or groups of beams presentation - bending moment, shear, stress, deflection animation of results in .avi format etc.)

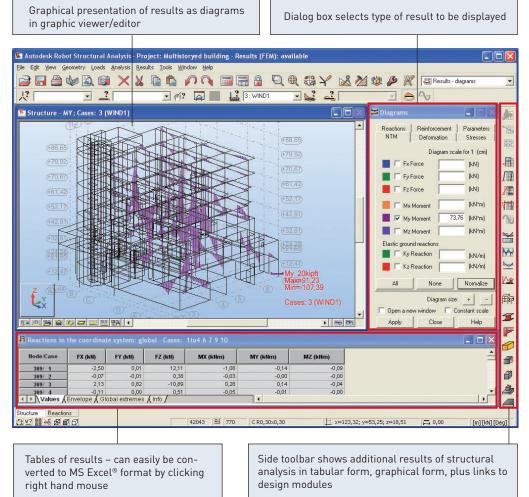


Figure 9.2 - Results in the diagrams form

There is also easy acces to calculation results for single bars. In context menu of the bar (right mouse click on the element, then select Object Properties option) user can see diagrams and values for selected quantity of internal forces.

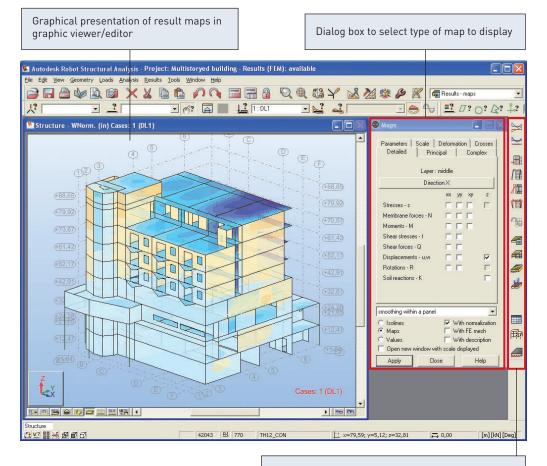
	T	
		-
4	Len	ath (m)
0.00 30.00	40.00 50.00	60.00
MY	Diagram	
	Fx 📕	П Мх
168	Fy Fy	My My
(=0,0 (m)	Fz Fz	∏ Mz
	📕 🗖 Smax 📕	🗖 🗐 Smin
Extremes		
	MY -4,39 168 (=0,0 (m)	.00 30.00 40.00 50.00 MY

A right mouse click on any bar displays its properties and results (if analysis has been carried out)

Figure 9.3 - Calculation results for single bar

Graphical results for surfaces

(graphical results for surfaces, contour maps for bending, deflection, stress, animations; "reduced results" showing global forces in surface cuts, including direct reduced results for cores and stiff diaphragms)



Side toolbar shows additional results of structural analysis in tabular form, graphical form, plus links to design modules

Figure 9.4 - Results in the maps form

Tabular results

🗈 Autodesk Robot Structural Analysis - Project: Multistoryed building - Results (FEM): available							
🔁 🖬 🎒 🥼 🖄 🗶 🕼 🔺 🌡 💁 🌈 🦳 📰 🚟 🔒 🔍 🍭 🖏 🌱 🕍 🏄 🅸 🌽 🦉 🖷 Results-maps 📃							
🙏 11o294 298 co 🗴 📝 11o171 180lo249 2 🗾 🎻? 🛄 🔛 🔛 11o10 🔄 🛀 🕹 💭 📃 🗁 🖓 🛄 🖉 🦓							
🛛 FE Results Direction X - Cases: 1to4 6 7 9 10							
Panel/Node/Case	MXX (kNm/m)	MYY (kNm/m)	MXY (kNm/m)	Definition		Detailed Principal Complex Parameter	
237/ 19/ ULS/20	9,26>>	13,13	-0,19		*1.40		
237/ 4802/ULS/8	-5,19<<	-1,96	-0,01	1*1.20 + 3*1.60 + 4	*0.50	Cano	cel
237/ 19/ ULS/20 237/ 8/ ULS/9	9,26	13,13>> -4,68<<	-0,19 2,00	1*1.20 + 3		Layer : Layer middle	
237/ 8/ ULS/9 237/ 8/ ULS/9	-4,00	-4,68	2,00	1*1.20 + 3		Direction X Hel	P
237/ 22057/ ULS/8	1,99	-0,04	-2,25<<	1*1.20 + 3*1.60 + 4		xx yy xy z	
238/ 19/ ULS/20	9,66>>	10,68	-0,15		*1.40		
238/ 4769/ULS/6	-11,97<<	-7,27	-3,63	1*1.20 + 2*0.50 + 3*1.60 + 4	*0.50	Stresses - s Filte	rs 1 💷
238/ 19520/ ULS/6	5,48	12,51>>	1,07	1*1.20 + 2*0.50 + 3*1.60 + 4	*0.50	Membrane forces · N F F F Extrem	mes l om
238/ 4769/ ULS/20	-5,48	-7,46<<	-1,89		*1.40	Moments · M 🔽 🔽 🗸	
238/ 21500/ ULS/20	0,07	7,26	4,02>>		*1.40	Shear stresses - t Case	
238/ 4769/ULS/6	-11,97	-7,27	-3,63<<	1*1.20 + 2*0.50 + 3*1.60 + 4		Shear forces - 0	- 4
239/ 1/ ULS/6	2,20>>	1,52	0,26	1*1.20 + 2*0.50 + 3*1.60 + 4		and the second	Æ
239/ 3285/ ULS/6	-3,84<<	-1,27	-0,31	1*1.20 + 2*0.50 + 3*1.60 + 4		Displacements - u,w 🔽 🔽	*****
239/ 1/ ULS/9	2,18	1,53>>	0,26	1*1.20 + 3		Rotations - R 🔽 🗖	
239/ 3285/ULS/6	-3,84	-1,27<< 0,55	-0,31 0,42>>	1*1.20 + 2*0.50 + 3*1.60 + 4	*0.50	Soil reactions - K	
239/ 21/ ULS/20 239/ 3274/ ULS/6	-0,22	0,55	-0.57<<	1*1.20 + 2*0.50 + 3*1.60 + 4			1
240/ 19520/ ULS/20	4,84>>	8,79	-0,57		*1.40	All None	
240/ 4724/ULS/6	-11.88<<	-7.95	1,88	1*1.20 + 2*0.50 + 3*1.60 + 4			
240/ 11/ ULS/20	-1.08	13.73>>	-1.68		*1.40		
240/ 4724/ULS/20	-6,85	-8,70<<	0,34	1	*1.40		
240/ 4724/ULS/6	-11,88	-7,95	1,88>>	1*1.20 + 2*0.50 + 3*1.60 + 4	*0.50		12000
240/ 19516/ ULS/20	-0,47	3,65	-3,02<<		*1.40		龠
241/ 35560/ ULS/6	9,13>>	4,93	2,58	1*1.20 + 2*0.50 + 3*1.60 + 4	*0.50		
241/ 4879/ ULS/1	-7,21<<	-16,05	-0,15	1*1.20 + 2*1.60 + 4		FE Results	
241/ 19541/ ULS/20	4,50	8,68>>	0,50		*1.40	For the active table, columns selected on this tab	
241/ 17/ ULS/1	0,61	-16,50<<	-1,08	1*1.20 + 2*1.60 + 4		will be added will replace existing ones	
241/ 12/ ULS/20	0,36	5,52	4,25>>	1*1.20 + 2*1.60 + 4	*1.40		
241/ 35584/ ULS/1 242/ 5466/ULS/7	6,74 3,31>>	8,66	-2,50<<	1*1.20 + 2*0.50 + 3 1*1.20 + 2*0.50 + 3			
242/ 5466/ ULS// 242/ 5708/ ULS/6	-2.49<<	-0,05	-1,04 0,24	1*1.20 + 2*0.50 + 3*1.60 + 4			
Values Envelope	2.00	2 0255	0.65	4#4 00 - 0#0 60 - 0#4 60 - 4			
A Auros Vennelope V	olopai extrem	es Visaueia	•				
Structure FE Results							
Tables can easily be edited, enveloping results, setting limits, filtering data by load case etc. Dialog box used to define contents of the table. It is available from the menu activated							
	by pressing the right mouse button when the cursor is located in the table, and selecting					5 5	

Figure 9.5 - Tabular results

Robot tables can be easily transferred to MS Excel[®] by pressing the right hand mouse button.

TABLE COLUMNS

Inside Robot, the tables can be manipulated in many ways, in the same way as input tables:

- filtering data for specific load cases, members or groups of members
- filtering data inside or outside user defined limits showing global maxima and minima for selected members, nodes or surfaces

REPORTS AND PRINTOUT COMPOSITION

Robot has a built in "report generator" which permits the user to create a user defined printout for a Project.

There are two options on the menu bar - "Printout Composition" and "Screen Capture" which help to prepare the final calculations notes, tables and graphic printouts.

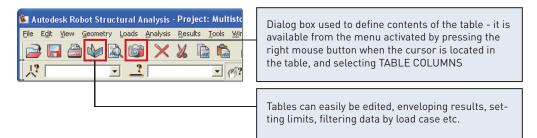


Figure 10.1 - Printout Composition and Screen Capture icons

The **Standard** tab lists a set of predefined options for printout, which can be mixed with screen captures to the user's requirements.

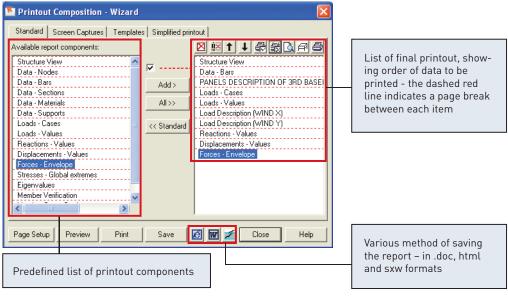


Figure 10.2 - Printout composition

A key feature of Robot is that all of the data in the program is saved in a common binary file – this data includes input, output, design plus also printout data. When structural data is altered or amended the output and printout (tables and graphics) are automatically updated on re-analysis to reflect the new results of the analysis. This is particularly useful when considering a schematic design or "sensitivity study" since the printout is automatically updated at every analysis step to reflect the current structural configuration.

ADVANCED ANALYSIS AND DESIGN

Following analysis of the structure, the user may wish to take advantage of many of the additional features – such as:

- Steel design
- Concrete design for beams, columns, floors
- Advanced analysis P-Delta, dynamic etc.

These items are beyond the scope of this "Getting Started Manual" and are covered in more depth in the help for Robot.

LIST OF SHORTCUTS

In order to	Press
select all	Ctrl + A
copy a text or a drawing	Ctrl + C
open a new project	Ctrl + N
open an existing project	Ctrl + 0
start printing	Ctrl + P
save the current project	Ctrl + S
cut a text or a drawing	Ctrl + X
repeat the last operation	Ctrl + Y
paste a text or a drawing	Ctrl + V
undo the last operation	Ctrl + Z
display the 3D view of a structure (3D XYZ)	Ctrl + Alt + 0
project a structure on XZ plane	Ctrl + Alt + 1
project a structure on XY plane	Ctrl + Alt + 2
project a structure on YZ plane	Ctrl + Alt + 3
zoom in the structure view on screen	Ctrl + Alt + A
display the initial view of the structure (defined by the initial an-	Ctrl + Alt + D
gles and scale)	
"exploded" view of structure elements (on/off)	Ctrl + Alt + E
zoom window	Ctrl + Alt + L
turn on/off section drawing display	Ctrl + Alt + P
screen capture	Ctrl + Alt + Q
zoom out structure view on screen	Ctrl + Alt + R
turn on/off section symbol display	Ctrl + Alt + S
rotate continuously around the X axis	Ctrl + Alt + X
rotate continuously around the Y axis	Ctrl + Alt + Y
rotate continuously around the Z axis	Ctrl + Alt + Z
delete a text or a drawing	Del
call Robot Help system for the active option in the active dialog	F1
box	
call text editor	F9
reduce structure attributes (supports, numbers of nodes, bars,	PgDn
loads) presented on screen	
enlarge structure attributes (supports, numbers of nodes, bars,	PgUp
loads) presented on screen	

3D FRAME STRUCTURE

Synopsis:

The purpose of this example is to show the ease of definition, analysis and report generation for a simple steel 3D frame in Robot. For this example, it is assumed that Autodesk Robot Structural Analysis 2010 is installed on the PC.

CONFIGURATION OF THE PROGRAM

Following installation of Robot, the user may configure the working parameters or "Preferences". To do this:

- 1. Start the Robot program (click the appropriate icon or select the command from the taskbar).
- 2. On the opening window, displayed on the screen click the last but one icon in the first row (Frame 3D Design). Other options are for 2d and 3d frames, models with surfaces such as walls and floors, access to stand alone design modules etc.
- 3. Select **Tools** > **Preferences** from the text menu or click on the \checkmark , then icons on the toolbars to open the "Preferences" dialog box. Preferences allow the user to set up working and printout languages, fonts, colors etc.

and Preferences			? 🛛
Canquages General Parameters View Parameters Desktop Settings Toolbar & Menu Printout Parameters Authorization Key Advanced	D Begional settings: Working language: Printout language:	United States	
✓ Update Preferences on exit		Accept Cancel	Help

Figure 1 - Regional settings in Preferences dialog box.

4. Select the first preferences option - Languages (tree on the left part of the window), then as the **Regional settings** choose **United States**.

Regional settings set the default databases (profiles, materials), units and codes to the standards of a country. In the example above, we have chosen American section database (AISC) and metric data units: [m],[cm], [kN].

5. Click Accept to close the window.

NOTE:

You can check active units in right, bottom corner of the screen. In this example should be the display: [m] [kN] [Deg].

MODEL DEFINITION

Bars definition (frame 2D)

In this step, you create frame members consisting of 2 columns and a beam.

- 1. Click icon (right side of the screen) to open the **Bars** dialog box.
- 2. Set **Bar type**: as **Column** whatever is selected is not important for analysis, but affects the design parameters for subsequent member design, such a buckling length, position of restraint etc.
- 3. Define Section: as W14x211.

If the **W14x211** section is not available on the list, you should click the (...) button located beside the **Section** field and add this section from the database. In opened **New section** dialog box, in **Section selection** field for **Database: AISC** – choose **Family: W** then Section: **W14x211**. Click **Add** and **Close** the box.

INFO

There are many extra options that may be entered for fabricated members, tapering sections and also for beams that the user wants to define as able to exhibit plasticity.

- 4. Enter the following points in the **Beginning** and **End** field:
 - to define first column of frame: (0;0;0) and (0;0;3) Add
 - to define second column: (7;0;0) and (7;0;3) Add
- 5. Set **Bar type**: as **Beam**.
- 6. Define **Section**: as **W14x211**.
- 7. To define a beam in the structure, enter the following points in the **Beginning** and **End** field: (0;0;3) and (7;0;3) **Add**.
- 8. Close the Section window.

Supports definition

In this step, you create supports for the frame structure.

- 1. Click icon (right side of the screen) to open the **Supports** dialog box.
- 2. To choose structure nodes, the user can either select a support type from the list and then select a node or nodes by click or window, or the user can directly input

the node number and apply. To make a window selection, press the right hand mouse key and then the **Select** menu option and window around the nodes to be supported and click inside the selection box.

- 3. From the Supports dialog box select the Fixed support icon.
- 4. Click Apply.

INFO

2-bay frame definition

In this step, we create a 2 bay frame by mirroring the existing frame.

 Select all bars (by window or CTRL+A) and mirror them Edit > Edit > Vertical Mirror by vertical axis of the right column (just click on this axis).

There are many editing possibilities: copy, move, divide, intersect, rotation etc. to make modeling of structure easier and more effective.

- 2. Click icon (top of the screen) to show the whole structure.
- 3. To display bars and nodes numbers click 📰 icon (bottom left corner of the screen) then in the Display dialog box tick on Favorites > Node numbers and Favorites > Bar description > Bar numbers.
- 4. Click Apply and OK.
- 5. Click 🚔 , 💹 icon at the bottom of the screen to display supports symbol and sections shape.
- 6. The frame should appear as below:

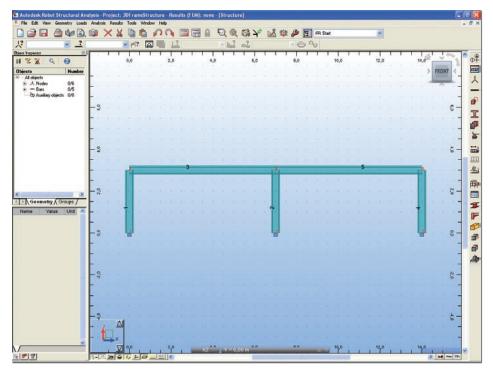
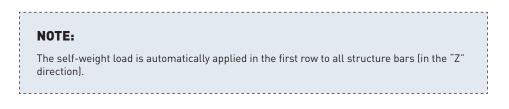


Figure 2 - Frame with supports and sections shape view.

Load case definition

In this step, you define load cases (their number, nature and description).

- 1. Click icon (right side of the screen) to open the **Load Types** dialog box.
- 2. Click New to define a dead load (self-weight) with a standard name DL1.



- 3. Choose in the **Nature** field **live**, then **New** to define a **live load** with a standard name **LL1**. We have now defined 2 load cases.
- 4. Choose in the **Nature** field **wind**, then **New** to define a **wind load** with a standard name **WIND1** (in the same way define **WIND2**). We have now defined 4 load cases.
- 5. Close the dialog box.

Loads definition for particular load cases

In this step, you define loads for each created load case.

1. To define loads for LL1 case select the 2^{nd} load case in the list of defined cases field:

🖺 Autodesk Robo	ot Structural Analysis	s - Project: S	itructure - R	lesults (FEM	i): out of da	te - [Struct	ure - Cas	es: 2 (LL	.1)]	
R Eile Edit View	Geometry Loads Ana	ilysis <u>R</u> esults	Tools Windo	w <u>H</u> elp						
		\times %		20				Y	×	墩 🖉
1.1	?	1	· @? 🗛			•	2 2			-
Object Inspector	8				1 : DL1					
7 🛣 🔍					3 : WIND1 4 : WIND2	1				
Objects 	Number of obje 0/12				Simple Case	es				

Figure 3 - Load case LL1 selection.

- 2. Click icon (right side of the screen) to open the Load Definition dialog box.
- 3. On the **Bar** tab click icon to open the **Uniform load** dialog box, then type value of load **-45.0** in the **pZ** field, click **Add** (**Uniform load** dialog box is automatically closed).
- 4. Select beam span (no.3) indicate them or just type 3 in field Apply to.
- 5. Click Apply.

6. On the **Bar** tab click icon to open **Trapezoidal load** dialog box, then type values of loads and coordinates as shown below, click **Add** and close the box. This generates a varying load on the beam.

🆽 Trapezoidal	load 📃 🗖 🔀						
	¹ p ² ¹ x ² +						
Type: Trape	ezoidal load (2p) 🛛 🔽						
Direction of load action							
Coord. system:	💿 Global 🛛 🔿 Local						
Values	Coordinates						
(kN/m)	 O Relative (x/L) ○ Absolute (m) 						
p1 = -30,00	x1 = 0,25						
p2 = -45,00	x2 = 0,75						
Angles (Deg) X = 0,0 Y = 0,0 Z = 0,0							
Projecte	d load						
Add	Close Help						

Figure 4 - Trapezoidal load definition.

- 7. Select beam span (no.5) indicate the beam or just type 5 in field Apply to.
- 8. Click Apply and close Load Definition dialog box.
- 9. To define loads for WIND1 case select the 3rd load case in the list of defined cases field.
- 10. On the **Node** tab in **Load Definition** dialog box click icon to open **Nodal Force** dialog box, then type value of load **9.0** in the **FX** field, click **Add** and close the box.
- 11. Select the upper node of the left column (no.2) graphically or by cross-window or just type **2** in field **Apply to**.

- 12. Click Apply and close Load Definition dialog box.
- 13. To define loads for WIND2 case select the 4th load case in the list of defined cases field.
- 14. On the **Bar** tab in **Load Definition** dialog box click icon to open **Uniform load** dialog box, then type value of load -20.0 in the pX field (pY=0; pZ=0), click **Add** and close the box.
- 15. Select the right column (no.4) indicate it or just type 4 in field Apply to.
- 16. Click Apply and close Load Definition dialog box.
- 17. Close the Load Types dialog box.
- 18. Click icons at the bottom of the screen to turn displaying loads symbols and loads values description on.
- 19. The loads can also be seen in tabular form click Tables icon (right side of the screen) then tick Loads on or select View > Tables > Loads command (see below).

INFO

All tables in Robot can be exported to MS Excel® by simply clicking the right hand mouse button in the table.

Case	Load type	List						
1:DL1	self-weight	1to5	Whole structu	-Z	Factor=1,00	MEMO:		
2:LL1	uniform load	3	PX=0,0	PY=0,0	PZ=-45,00	global	not project.	absolute
2:LL1	trapezoidal load (2p)	5	X1=0,25	PX1=0,0	PY1=0,0	PZ1=-30,00	X2=0,75	PX2=0,0
3:WIND1	nodal force	2	FX=9,00	FY=0,0	FZ=0,0	CX=0,0	CY=0,0	CZ=0,0
4:WIND2	uniform load	4	PX=-20.00	PY=0,0	PZ=0.0	global	not project.	absolute

Figure 5 - Load cases data in tabular form.

20. In the list of defined load cases choose the load case for which the loads will be displayed:



21. All defined loads cases will be displayed together (as shown below):

		\mathbf{X}	1 B	1	no			6 5	2 Q	部子	° 📩	墩	S E	FR Stat			*				
4		ato:	- 6	? 🗖		12 St	nple Cases		-12	2		1	0								
hapector 🐰		۰.	-2.0	-	1	1	2,0	T,	1	1	1	1	1	1	1	1	12,0	-		T	*
			-2,0		0,0		2,0		4,0		6,0		8,0		10,0		12,0		14,0		
Al objects Number	F.																			FRON	-
 人 Nodes 1/6 一 Bass 0/5 一 Auxiliary objects 0/0 	-2																			12	e
CO ADMINISTRATION DATE																					
	Ē																				1
	-\$																				s-
	ŀ																				-
	-9								pZ=-4	5.00						pZ=-41	5.00;-30	00			-
	1		E	x=9.00													T				6
	F		. /=		TE	1.1	1 1 1	3	de la	أسلسك			_			- 3	-	_	- în		
Geometry Groups /	- %																			x=-20.00	00-
ame Value Unit												2							, P		-
fnod 4 neral dom 235	-																				-
djoini	- %				4							4									~ -
oordi 7,00 0,0 pe o cartesian litional characteristics	ŀ																				
uppor to to																					
ompat 62	- 0'7																				20
	F																				2 2
	- 4																				÷
		. 1																	Z KG		
	1	Π.																	4 8.91	s: 1104	-

Figure 6 - All defined load cases view.

Copying existing frame

In this step, we copy the 2D frame to generate 3D structure. When we copy that frame, all attributes attached to it (loads, sections, supports etc.) are also copied:

Click icon (top of the screen) to open the View dialog box, then or choose View > Work in 3D > 3D xyz from the menu to select the isometric structure view.

- 2. Click CTRL+A to select all of the structure.
- 3. Edit then $\xrightarrow{\uparrow \to \square}$ Translation or select the Edit > Edit > Translate option from the pull-down menu to open the Translation dialog box.
- 4. In **Translation vector** field type coordinates as shown below (be sure that in **Edit mode** option **Copy** instead of **Move** is checked on and **Number of repetitions** is 1).

🛄 Translation	
Translation vector (m)	
dX; dY; dZ = 0;8;0	
Numbering increment	
Nodes:	
Elements:	
Edit mode	
💽 Сору	Drag
O Move	
Number of repetitions:	1
Execute Close	Help

Figure 7 - Copy frame parameters definition.

- 5. Click Execute second frame should be display on the screen.
- 6. Close the **Translation dialog** box.
- Click icon to show the whole structure. The structure should look as shown below:

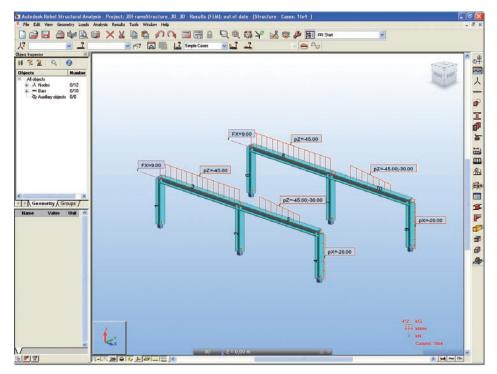


Figure 8 - 3D View of copied frame.

Definition of lateral beam

In this step, you define beams to join the selected frames together:

- 1. Click icon (right side of the screen) to open the **Bars** dialog box.
- 2. In **Bars** dialog box set **Bar type**: as **Beam**.
- 3. Define Section: as W12x190.



If the **W12x190** section is not available on the list, one should click the (...) button located beside the **Section** field and add this section to the active section list in the **New section** dialog box.

4. In the fields **Beginning** and **End** type coordinates (as shown below) or simply select beginning field and then graphically draw the start and end points of the member.

Bars							
Number: 11 Name: Beam	Step: 1						
Properties							
Bar type:	Beam 🔽 🛄						
Section:	W 12x190 🔽 🛄						
Default material:	Default material: STEEL						
⊂Node coordinates	(m)						
Beginning: 14	.00; 0,00; 3,00						
End: 14	.00; 8,00; 3,00						
	Drag						
Axis position Offset:	None 💌						
Add	Close Help						

Figure 9 - Bars definition.

5. If entered by coordinates, Click Add and close Bars dialog box.

Definition of cross bracings

 Sections then New section definition – or select the Geometry > Properties > Sections > New section definition option from the menu to open the New Section dialog box:

I New Section	
Standard Parametric Tapere	d Compound Special Ax, Iy, Iz Variable 4.16 (cm) Section selection Database: AISC American hot rolled shapes (Jan 20 Eamily: L Angles (x-y axis, parallel to legs) Section: 4x4x0.25 Elasto-plastic analysis
<u>G</u> amma angle: 0 ▼ (Deg)	Section type: Steel

Figure 10 - New section definition.

- 2. Select Section (L4x4x0.25) from American section Database (AISC) as shown above.
- 3. Click Add new section type will be added to the active Sections list.
- 4. Close New section definition and Sections dialog boxes.
- 5. Click icon (right side of the screen) to open the **Bars** dialog box.
- 6. In **Bars** dialog box set **Bar type**: as **Simple bar**.
- 7. Define Section: as L4x4x0.25.
- In the fields Beginning and End type coordinates: (14;0;3) (14;8;0) click Add then (14;8;3) (14;0;0) and press Add. Or draw in the GUI.
- 9. Close **Bars** dialog box.

Copying defined bars (lateral beam and bracings)

In this step, you copy the recently defined bars to finish the definition of the frame:

- 1. Select the three recently defined bars just indicate them (bars no.11,12,13) multiple selections can be made by holding down CTRL key.
- 2. Edit then $\overrightarrow{\Box} \leftarrow \overrightarrow{\Box}$ Translation or select the Edit > Edit > Translate option from the menu to open the Translation dialog box.
- 3. In **Translation vector** field type coordinates: (-7;0;0) or select 2 points on the screen that represent the translation vector such as the bottom of 2 columns (be sure that in **Edit** mode option **Copy** is checked on) and in **Number of repetitions** type **2**.
- 4. If entered by coordinate, click **Execute** button then close **Translation** and **Edit** dialog boxes.
- 5. The structure should look as shown below:

47

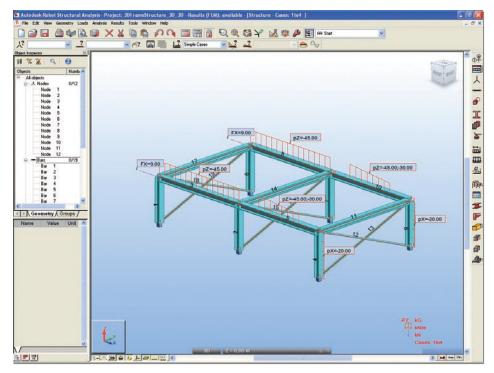


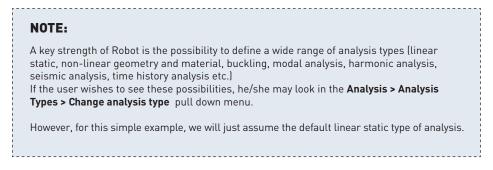
Figure 11 - Complete 3D frame structure view.

STRUCTURE ANALYSIS

Here we start the analysis process, but firstly we will tell Robot to make automatic code combinations (from any one of the list of Codes in Job Preferences):

- Select Loads > Automatic Combinations... option from the menu to open the Load Case Code Combinations dialog box. Select the Full automatic combinations option. The program will now automatically assign a number of combinations to find the most onerous load combination.
- 2. Click OK button automatic calculation of code combinations will be done.

- 3. Calculations click this icon or select the Analysis > Calculations option from the menu to start calculation process.
- 4. Once the calculations are completed the information: **Results (FEM): Available** should be displayed at the top of the screen.



RESULTS PREVIEW

Displaying beam results graphically

In this step, we display My bending moment on bars for selected load case:

- 1. Click 🗾 icon at the bottom of the screen to switch off the section shape and simply display a "stick model" for all members.
- 2. In the list of defined load cases choose the desired load case to display:



Select Results > Diagrams for Bars...
 Diagrams for Bars...
 from the menu to open
 Diagrams dialog box.

4. On the **NTM** tab check **My Moment** (this shows the major axis bending moment on the beams), then click **Normalize** button (to auto scale) and click **Apply** to display bending moments diagrams for all beams:

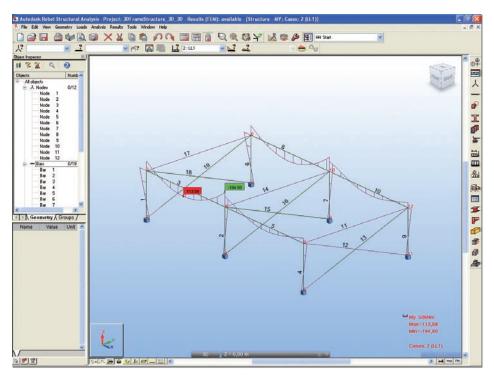


Figure 12 - Diagram of bending moment.

NOTE:

To display numerical values of internal forces in the **Diagrams** dialog box, click on tab **Parameters, Diagram description** tick labels and **Apply**.

- 5. In a similar way, diagrams that exhibit other values available from the **Diagrams** dialog box can be viewed.
- 6. Tick off My Moment and Apply.

Displaying results on bars in tabular form

In this step, you display internal forces for bars for particular load cases and combinations:

 Select View > Tables to open Tables: Data and Results dialog box, then tick on Forces and OK to view all information about internal forces (Values, Envelope, Global extremes).

File Edit View Geometry Loads	XXDA				8 🕸 🖉 🖭	FR Start	~	
	o19 of contractions of the contraction of the contr			· · · · · · · · · · · · · · · · · · ·			M	
at hapestor	and the second	1			1			
7 X Q @	Bar Node Case	FX (MI)	FY (kll)	FZ (kil)	MX (hilm)	MY (kilim)	MZ (kilm)	1
	1/1/1	30,62	5,65	-4,27	0,00	4,09	5,59	
iects Numb A	1/ 1/ 2	148,26	0,00	-62,90	0,00	57,40	0,01	
All objects 는 人 Nodes 0/12	1/ 1/ 2	-0,77	-0,00	2,88	-0,00	-5,67	-0,00	
E Å Nodes 0/12 Node 1	1/ 1/ 4	2,04 273,96	0,00	-7,15 0,76	0,00	13,95 96,74	0,00 7,02	
Node 2	1/ 1/ ULS-	26,33	5.00	-105.76	0,00	-5.40	5,00	
Node 3	1/ 1/ SLS+	180.92	5.65	-105,75	0,00	75,43	5,60	
Node 4	1/ 1/ SLS+	29,86	5,65	-74,33	0,00	-1,59	5,59	-
Node 5	1 2 1	21,39	5,65	-4,27	0,00	-8,73	-11,35	
Node 6	1 2 2	148,28	0,00	-62,90	0,00	-131,30	-0.00	
Node 7	1 2 3	-0,77	-0,00	2,00	-0,00	2,95	0,00	
Node 0	1/2/4	2,04	0.00	-7,15	0.00	-7,52	-0.00	
Node 3	1/ 2/ ULS+	262,00	7,90	0,76	0,00	-3,13	-10,21	
Node 10	1/ 2/ ULS.	18,02	5.08	-105,76	0,00	-220,55	-15,89	
Node 11	1 2 51.5+	171,60	5,65	-1,40	0,00	-5,77	-11,35	
Node 12	1 2 SLS-	20,62	5,65	.74,33	0,00	.147,54	-11,35	
Bast 0/19	2 3 1	43,02	-5,65	-0.00	-0.00	0,00	-5,59	
Dar 1	2/ 3/ 2	244,64	-0,01	-20,73	-0,00	31,52	-0,02	
Bar 2	2/ 3/ 3	0,07	+0,00	-3,60	-0,00	6,14	-0,00	
Bar 3	2/ 3/ 4	-0,76	0,00	9,46	0,00	-16,58	0,00	
Bar 4	2/ 3/ ULS+	443,70	-5,08	15,13	0,00	50,43	-5,03	
Bar 5	2/ 3/ ULS-	38,22	-7,90	-45,97	-0,00	-26,50	-7,82	
Bar 6	2/ 3/ SLS+	200,34	-5,05	9,46	0,00	37,66	-6,59	
Dar 7 🔛	2/ 3/ 5L5-	43,06	-5,65	-32,33	-0,00	-16,56	-5,60	
3	2 4 1	34,59	-5,65	-0,00	-0,00	-0,00	11,35	
Geometry Groups /	2 4 2	244,44	-0,01	-28,73	-0,00	-54,67	0,01	
ame Value Unit	2 4 3	0,07	-0,00	-3,60	-0,00	-4,67	0,00	
ame vaide ond	2 4 4	-0,76	0,00	9,46	0,00	11,82	-0,00	
	2/ 4/ ULS+	432,61	-5,08	15,13	0,00	10,91	15,09	
	2/ 4/ ULS-	29,91	-7,90	-45,97	-0,00	-87,47	10,21	
	2/ 4/ SLS+	279,10	-5,65	9,45	0,00	11,82	11,35	
	2: 4 SLS-	33,83	-5,85	-32,33	-0,00	-59,34	11,35	
	3 2 1	4,27 62,90	-0,00	9,96 148,46	-0,00	-8,73	-0,00	
	¥ 2 2		0,00	-0.77	-0.00	2.95	0.00	
	3/ 2/ 3	6,12 7,16	+0,00	-0,77 2,04	0,00	-7,52	-0,00	
	3 7 01.5+	105,77	-0,00	249,48	-0,00	-7,52 -3,13	-0,00	
	2 2 ULS-	3,84	-0,00	7,74	-0,00	-220,58	-0,02	
11 H	3/ 2/ SLS+	74,33	-0,00	160,46	-0,00	-5,77	-0,02	
	3/ 2/ SLS-	4,27	-0.00	9,19	-0.00	-147.55	-0.01	
	3 4 1	4,27	-0,00	-11,59	-0,00	-14,42	0,00	
	3 4 2	62,90	-0.00	-166.54	-0,00	-194,60	0.00	
	2 4 3	6,12	0,00	-0,77	-0,00	-2,42	-0,00	
	2 4 4	7,16	-0,00	2,04	0,00	6,79	0,00	
	3 4 ULS+	105.77	-0.00	-7,16	-0.00	-2.11	0.00	
	3 4 ULS-	3,04	-0,00	-200,37	-0,00	-320,67	-0,00	
	3 4 SLS+	74,33	-0.00	-9,54	-0.00	-7.63	0.00	
	3 4 51.5	4,27	-0,00	-178,90	-0,00	-211,44	-0,00	
	4 5 1	30.62	-5.65	-4,27	-0.00	4.09	-5,59	
		62.04	0.00	94.17	0.00	96.30	0.00	*

Figure 13 - Example of results presentation in tabular form.

NOTE:

In a similar way, tables that exhibit other values available from the **Tables: Data and Results** dialog box can be viewed.

The tables can easily be exported to MS Excel® (by right hand mouse click) and tables can also be sorted by choosing certain load cases, member types etc. Advanced users can also define "groups" and display results only for these groups. Column values may be placed in order by simply clicking the appropriate column header, for example **FX**.

51

- 2. Close the **Forces** table.
- 3. Click 🗾 icon at the bottom of the screen to turn sections shape back on.

Stress analysis

In this step, we learn how to obtain and analyze stress diagrams and maps for the entire bar structure:

- 1. Switch layout to **Stress Analysis Structure**. To do this go to the Robot layouts selection combo-box (right top screen corner)
- 2. Click the Results option and later the Stress Analysis Structure.



Figure 14 - Layout selection pull-down menu (stress analysis selection).

NOTE:

Now the screen is divided into three main parts:

- the graphic viewer where the structure is displayed,
- the Stress Analysis -Structure dialog box to managing stress analysis. This dialog box is used for selecting stresses and determining the manner of stress graphical presentation
- the Stress Analysis -Structure results table to view actual values of stresses for bars

52

3. In the list of defined load cases choose the load case for which the results will be displayed:



4. On the **Diagrams** tab located in the **Stress Analysis** - **Structure** dialog box select the **Max** option from the **Mises** field, then **Apply** (the result should look as shown below):

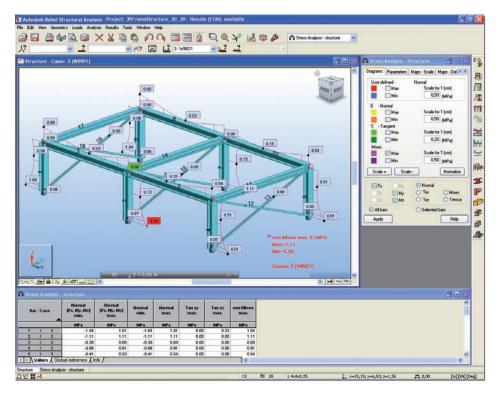


Figure 15 - Stress analysis in diagram form.

NOTE:

This dialog box allows selecting a user-defined stress and a set of basic stress types: normal, tangent, Mises and Tresca.

- 5. On the **Stress Analysis Structure** dialog box tick off the **Max** option from the **Mises** field, then **Apply**.
- 6. On the **Maps Deformation** tab located in the **Stress Analysis Structure** dialog box tick the **Deformation** option on, then **Apply**.
- 7. Stress Analysis for structure maps click this icon or select the Results
 > Stress Analysis > Stress Maps option from the menu to view the structure together with section shapes and accurate detailed stress maps on these sections (as shown in the figure below).

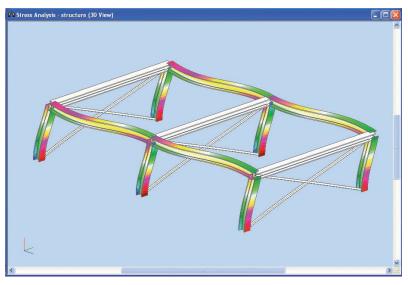


Figure 16 - Stress analysis in maps form.

The size of member sections in stress maps is by default greater than the real size of member sections so that the stress maps presented for these sections are more readable. Real proportions between the member length and dimensions of the member cross section can be obtained after pressing the **Home** key on the keyboard. These proportions can be modified using the following keys on the keyboard: **PgUp** and **PgDn**.

8. Close the Stress Analysis – structure (3D View) window.

PREPARATION OF PRINTOUTS

Robot has a powerful built in report generator that allows the user to compile and manipulate standard data, plus screen captures of graphical and tabular data.

"Capturing" views and data for the calculation note

- 1. In layouts selection combo-box (right top screen corner) switch to **Start** layout. The picture of the model is shown on the screen with no other windows. Let's assume we want to print this image.
- 2. Click icon (top of the screen on the standard toolbar) to open the Screen Capture dialog box.

Screen Capture		
Structure View		ОК
		Cancel
		Help
 Screen capture update View updated upon pi Current view (PNG) Whole structure 	rinting	
Screen capture orientation		rizontal
Scale • Automatic 1 cm on paper	O Us	er-defined m

Figure 17 - Screen capture parameters definition.

3. In the above dialog box change **Screen capture orientation** to **Horizontal** and type name of the picture as a **Structure View**.

The Screen capture update enables the user to choose automatic update of screenshots - this means that the images and data are automatically updated should the structure be changed. Alternatively, they can be saved in JPG format.

- 4. Click **OK** to close the box.
- 5. In the list of defined load cases choose the **2:LL1** load case.



- 6. Click icon (top of the screen on the standard toolbar) to open the **Screen** Capture dialog box.
- 7. In this dialog box change Screen capture orientation to Horizontal and leave name of the picture as a Structure - Cases 2: (LL1).
- 8. Click **OK** to close the box.
- 9. In the same way prepare pictures: Structure Cases 3: (WIND1) and Structure - Cases 4: (WIND2)

Preparing printout composition

Here we prepare the layout of the report:

- 1. Click icon (top of the screen on the standard toolbar) to open the **Printout** Composition - Wizard dialog box.
- 2. On the Standard tab, select report components (click them while pressing CTRL key) as shown below:

Printout Composition - Wizard	
Standard Screen Captures Template Available report components: Structure View Data - Nodes Data - Nodes Data - Sections Data - Sections Data - Sections Data - Sections Data - Supports Cases Loads - Values Reactions - Values Forces - Envelope Stresses - Global extremes Eigenvalues Member Verification Member Verification Member Group Design	s Simplified printout
Page Setup Preview Print	Save 🚺 👿 🚄 Close Help

Figure 18 - Report components selection.

- 3. Click **Add** to transfer the marked components to the right panel which shows the actual layout of the printout.
- 4. On the Screen Captures tab select all screenshots.
- 5. Click **Add** to transfer marked components to the right panel.
- 6. Using icons **1 u** move particular components to the order shown below:

Printout Composition - Wizard Standard Screen Captures Template	s Simplified printout
Available report components: Structure View Structure - Cases: 2 (LL1) Structure - Cases: 3 (WIND1) Structure - Cases: 4 (WIND2)	Image: Construction of the second
Page Setup Preview Print	Save 🖉 👿 🚄 Close Help

Figure 19 - Screen captures selection.

Printing and exporting the calculation report

In this step, we learn how to print and save the calculation note:

1. Click **Preview** to display the report:

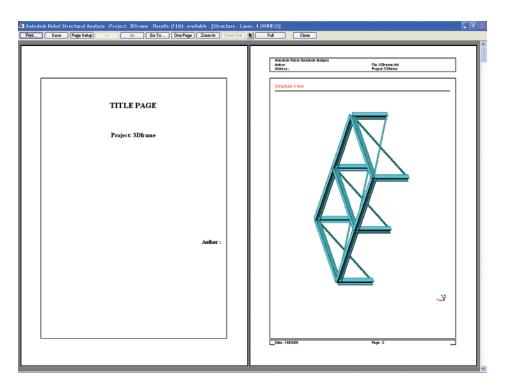


Figure 20 - Report preview.

Before printing of the report, save your project. In this case we save it as "**Frame 3D**". Thanks to this operation actual information about project will be used to generate the documentation (i.e. project name on the title page – as shown above). To change this information (and many more) later we can use **File > Project properties** option from the top menu.

2. Using buttons browse each page of the Report.

If you need to make some corrections to the layout or tabular content you can edit them

directly in this preview window. To do this, you should press icon (top bar) and double click on the appropriate item to be edited. The general Robot window will be open, which allows the user to make changes or corrections. After that press button **Return to Preview** to continue the preview of the calculation report.

- 3. Click **Page setup**. The **Page** and **Margins** tab allows the user to set various parameters.
- 4. Click **Header/footer** tab. In this tab you can define the look of the title page, header and footer.

Page Setup Page M	argins	Header/I	ooter Parameters	×
I Itle page I Header I Eooter I Table of contents	Edit Edit Edit	Restore Restore Restore Restore	Frame Restore Seperation lines Frames: Footer Text Header	Cancel Help
<u>S</u> elect te	mplate:		Save 1	Delete

Figure 21 - Page setup settings.

- 5. Click **OK** to close the dialog box.
- 6. Close the preview of report printout window.
- 7. If desired, In **Printout Composition Wizard** dialog box click on **Print** button to print all reports.
- 8. To save the calculation note, click on icon icon to export the report to .html

format, we to export to .rtf (MS Word[®]) format or to save in .sxw (Open Office) format.

RC AND STEEL MIXED STRUCTURE

Synopsis:

The purpose of this example is to show the ease of definition, analysis and presentation of results for a 3D frame and shell structure (mixed RC and steel) in Robot.

For this example, it is assumed that Autodesk Robot Structural Analysis 2010 is installed on the PC.

CONFIGURATION OF THE PROGRAM

Following installation of Robot, the user may configure the working parameters or "Preferences". To do this:

- 1. Start the ROBOT program (click the appropriate icon or select the command from the taskbar).
- 2. On the opening window, displayed on the screen click the first icon **i** in the first row (**Frame 2D Design**). Other options are for 3d frames and models with surfaces such as walls and floors, access to stand alone design modules etc
- Select Tools > Preferences from the text menu or click on the icons on the toolbars to open the Preferences dialog box. Preferences allow the user to set up working and printout languages, fonts, colors etc.

B Preferences			? 🔀
Constant of the second	Begional settings: Working language: Printout language:	United States	
☑ Update Preferences on exit		Accept Cancel	Help

Figure 1 - Regional settings in Preferences dialog box.

4. Select the first preferences option - Languages (tree on the left part of the window), then as the **Regional settings** choose **United States**

Regional settings set the default databases (profiles, materials), units and codes to the standards of a country. In the example above, we have chosen American section database (AISC) and metric data units: [m],[cm], [kN].

5. Click Accept to close the window



You can check active units in right, bottom corner of the screen. In this example should be the display: [m] [kN] [Deg].

MODEL DEFINITION

Definition of structural axes

In this step, we define a rectangular axis grid in the Cartesian coordinate system.

INFO

The axes of the structure create an additional grid which can be used to define different elements of the structure and select structure components.

The grid intersections form points that facilitate the users work by guiding cursor movements during graphical structure definition.

Press I icon (right side of the Robot screen) or select Geometry > Axis Definition... from the menu to open the Structural Axis dialog box.

- 2. On the **X** tab in the **Position** field type **-1** and press **Insert** button, in the same way enter following axis coordinates as shown below (Figure 2):
- 3. Set Numbering as A B C...
- 4. On the **Z** tab in the **Position** field type **0** and press **Insert** button, in the same way enter following axis coordinates as shown below (Figure 3):
- 5. Set Numbering as 1 2 3...

🚭 Structura	ıl Axis	
Name:	Structure axis	*
Cartesian	Arbitrary	
	Advanced paramete	rs
Xz		
Position: 15.00 Label	No. of repet.: m) 0 🗢 Position	Distance: 1 (m)
A B C D E	-1.00 0.00 7.00 14.00 15.00	Insert Delete
		Delete all
		Single out
<		
Numbering:	АВС 💌	
New	Axis r	manager
Apply	Close	Help

Figure 2 - Definition of structural axis in X direction.

🗣 Structural Axis	×					
Name: Structure axis	~					
Cartesian Cylindrical Arbitrary	ן					
Advanced parameters						
XZ	-					
Position: No. of repet.: Distance: 9,00 (m) 0 1 (m)						
Label Position						
1 0.00 2 4.00 3 7.00 4 8.00 5 9.00	ןנ					
4 8.00 5 9.00 Delete	ןן					
Delete all	ןן					
Single out	ןנ					
Stories						
Numbering: 123 💌]					
New Axis manager						
Apply Close Help						

Figure 3 - Definition of structural axis in Z direction.

- 6. Apply and Close the Structural Axis dialog box.
- 7. Defined axes should appear as shown below:

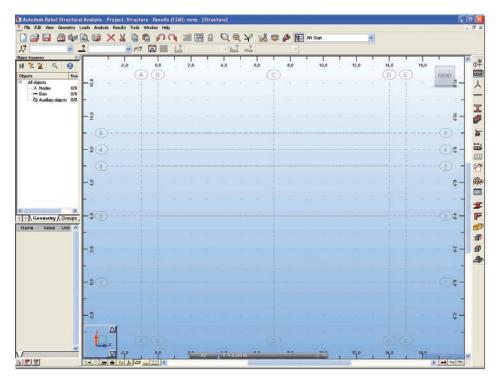


Figure 4 - View of defined structural axis.

Section definition

In this step, we learn how to add new sections to the list of available sections.

- Press icon (right side of the screen) or select Geometry > Properties > Sections from the menu to open the Sections dialog box.
- 2. Check presence of the following sections:

CR30x30, BR30x60, W8x28, W10x45.

3. If the above sections are not present in the list of available sections press the New

section definition icon to open New Section dialog box.

I New Section
General Parameters
Gamma angle: 0 ♥ (Deg) Section type: RC column ♥ Add Close Help RC beam RC column Timber Aluminum

4. In the Section type pull-down menu choose RC Column.

Figure 5 - New RC column definition.

- 5. Set **30** in the **b** and **h** field, in the **Label** field column name: **CR30x30** will be automatically created and click **Add** (see above).
- 6. In the Section type pull-down menu choose RC Beam.
- 7. Set **30** in the **b** and **60** in the **h** field, in the **Label** field column name: **BR30x60** will be automatically created, then press **Add**.
- 8. In the **Section type** field select **Steel**.
- 9. Set AISC in the Database field, W in the Family field and W 8X28 in the Section field.

I New Section			
Standard Parametric Tapere	Section selection Database: American hot re Family:	Special Ax, Iy, Iz Variable 20,5 (cm) on AISC v olled shapes (13th Ed W v hapes (W-Shapes)	
		W 8x28 W 8x18 W 8x21 W 8x24	
Gamma angle: 0 v (Deg)	Section type	W 8x28 W 8x31 W 8x35 W 8x40 W 8x48 W 8x58	

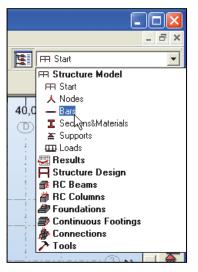
Figure 6 - New steel section definition.

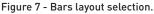
- 10. Press **Add** to add defined section to the list of the active sections.
- 11. As before, set **AISC** in the **Database** field, **W** in the **Family** field and **W 10x45** in the **Section** field.
- 12. Press Add, close New Section and Sections dialog boxes.

Bars definition

In this step, we define a 2D frame using the previously defined sections.

1. Go to the Robot layouts selection box (right top corner of screen) and switch layout to the **Bars**:





- 2. Go to the **Bars** dialog box and set: **RC Column** (Bar Type field), **CR30x30** (Section field).
- Place the cursor in the Node Coordinates > Beginning field, switch to the graphic viewer and select graphically the beginning and end of the columns by means of the coordinates of the intersection point of the defined axes: B1-B2, C1-C2, D1-D2 (see below):

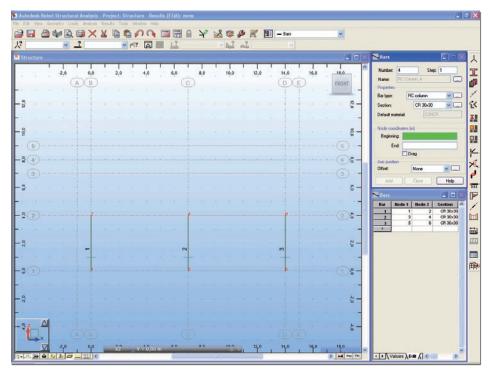


Figure 8 - View of defined columns.

- 4. Set the following options: **RC Beam** (Bar Type field), **BR30x60** (Section field).
- 5. Go to the **Node Coordinates** > **Beginning** and, as before, indicate the beginning and end of the beams: **A2-B2**, **B2-C2**, **C2-D2**, **D2-E2** (see below):

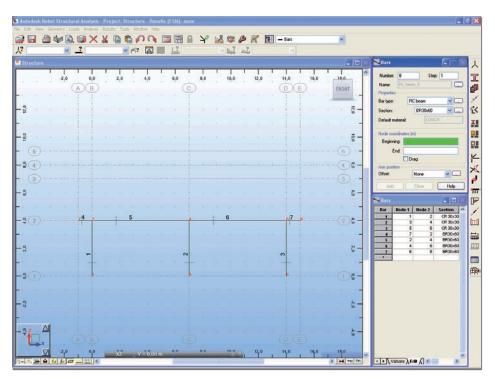


Figure 9 - View of added beams.

- 6. Set the following options: **Column** (Bar Type field), **W8x28** (Section field).
- 7. Go to the **Node Coordinates** > **Beginning** and, as before, indicate the beginning and end of the columns: **A2-A3, C2-C5, E2-E3** (see below):

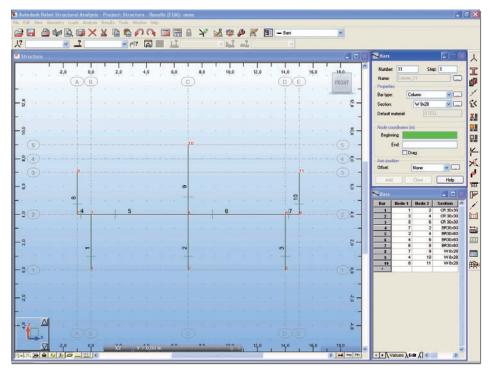


Figure 10 - View of added columns on the 2nd storey.

- 8. Set the following options: Beam (Bar Type field), W10x45 (Section field).
- 9. Go to the **Node Coordinates** > **Beginning** and, as before, indicate the beginning and end of the beams: **A3-C5**, **E3-C4** (see below):

Autodesk Ro					Structu			FEM): r	10 ME														5
284	a da D	3 101	XX		6	20				PN	8 啦	3	R 1	9	- Bars			~					
<u>با</u>		2			1032			17			13				10								
Structure																		-		Bars			1
2 A - 00 0	2,0		B	2,0		4,0		e.o			10,0		12,0	1	H,0	16,0)		18,0 FRONT		Number: 13 Name: Inc. Properties	Step m_13		
5.0									1.						1					Bartype: Section:	Beam W 10x45		2
IST II																			-	Default material			I
- (5) -			-						10										10.0	Node coordina Beginning End	es (m)		
• (3) -		e	4	-11	_				12				12	+	_	· - · · · · · · · · · · · · · · · · · ·		0	-	Axis position Other	Diag	•	* *
- 8		80							75						10				0.0	Add	Clove	Help	3
\$ <u>(</u> 2)-		4	2	+	5					+	6				7					1 2 3	1 2 3 4 5 6	CR 30x30 CR 30x30 CR 30x30	ú
																				4 5 6	7 2 2 4 4 6	BR30x60 BR30x60 BR30x60	
2,0		-						c	•						0				20	7	6 8 7 9	BR30x60 W8x20	
°(1)-															5				-	9 10 11 12	4 10 8 11 9 10 11 12	W 8x28 W 8x28 W 10x45 W 10x45	f
																			-	-			
50																		1	<u>'</u> -				
	10 - 11 1																		-				
																			-				
V 	20		20 <	32	1	10,00	m	**	•	-	10.0	-	12,0 T		40	14.0	1	18.0	-	< > Values	λean <u>κ</u>] < 0		

Figure 11 - Finished 2D frame view.

- 10. Press A, 1/2 icons at the bottom of the screen to display supports symbols and sections shapes.
- 11. Press 📰 icon at the bottom of the screen to open the **Display** dialog box.
- 12. Select the **Bars** option (left panel) and tick **Symbols** off (right panel) to turn off display of cross sections.

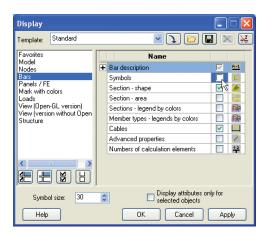


Figure 12 - Switch off bars symbols displaying.

13. Click Apply and OK to close the box. Frame 2D should look as shown below:

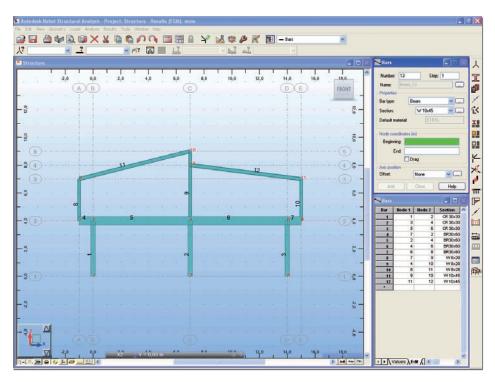


Figure 13 - 2D frame view with additional attributes - section shapes.

Supports definition

In this step, we create supports for the frame structure.

- 1. Go to the Robot layouts selection box (right top screen corner) and switch layout to **Supports**.
- 2. Go to the **Supports** dialog box and select **Fixed** in the list of supports.
- 3. Move the cursor to the graphical view. Note that a support symbol is shown that will allow the user to add supports to discrete nodes. To add to all nodes, press right mouse button and "select" and drag window round the nodes that have supports.
- Click inside "current selection" box and apply. Notice that the supports are also now shown in the tab.

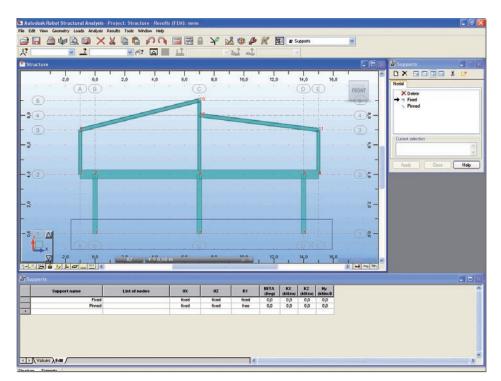


Figure 14 - Nodes selection to supports definition.

NOTE: You may also type the relevant supports numbers (1 3 5) into the **Current Selection** field and apply.

5. Defined frame with supports should look as shown below:

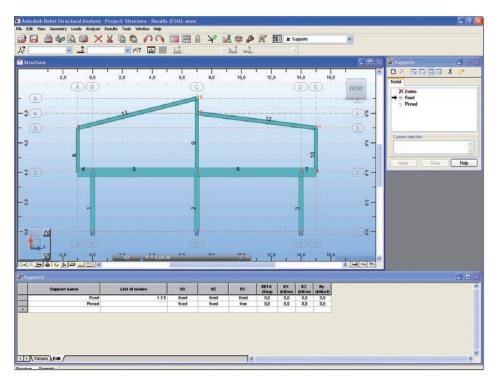


Figure 15 - Frame with defined supports view.

Load cases definition

In this step, we define names and natures of all load cases.

1. Go to the Robot layouts selection box (right top screen corner) and switch layout to the **Loads**.

? <u> </u>	🗹 (*)? 🖾 🛄 🚽	Li ~ Li ~i	2	
Structure Cases: 0 () \$ 5 6 - 4 - 3 - 2 - 2 - 3 - - - 2 - - -	00 - 50 (A B - 5 - 5 - 5 - 5 - 6 - 7 - 7 - 7 - 7 - 7 - 7 - 7 - 7		2000	Lin case description Case description Nuture: doed W New Number: 1 Labot: DL1 None: DL1 Lin cafored cases: No. Case more: Nature ModBy: Cales: Cales and ModBy: Cales: Melp
Leads - Case: 0 ()				×
Case Load type	List			

Figure 16 - Loads layout.

2. In the **Load Types** dialog box click **New** to define a dead load (self-weight) with a standard name **DL1**.

NOTE: The self-weight load is automatically applied – see the table (in the "Z" direction).

- 3. Click New once again to define a dead load with a standard name DL2.
- 4. Choose in the **Nature** field **live**, then **New** to define a live load with a standard name **LL1**.

- 5. Click New once again to define a live load with a standard name LL2.
- 6. Choose in the **Nature** field wind, then **New** to define a wind load with a standard name **WIND1**.
- 7. Choose in the **Nature** field snow, then **New** to define a snow load with a standard name **SN1**:

Ιæ	Load Ty	/pes			×
ſ	Case desc	ription			
	Nature:	snow	*	New	
	Number:	6	Label:	SN1	
	Name:	SN1			
	List of defi	ned case:	s:		
	No.	Case na	ame	Nature 🔼	
	4	LL2		live	
	5	WIND1		wind 📄	
	→6	SN1		snow 📮	
	<]	>	ĺ
	Modify		Delete	Delete all	
			Close	Help)

Figure 17 - Load types definition.

Definition of loads for predefined load cases

In this step, we define types and values of loads for particular load cases. Each load case can have several loads applied within it.

1. To define loads for **DL2** case select the 2nd load case in the list of defined load cases field (see below):

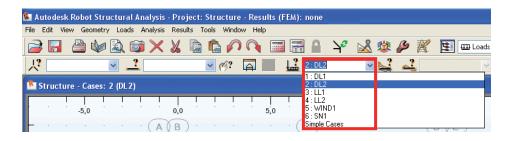


Figure 18 - DL2 load type selection.

- 2. Press icon (right side of the screen) to open the **Load Definition** dialog box.
- 3. On the **Bar** tab press icon to open **Uniform** load dialog box, then type value of load **-10** [**kN**/**m**] in the **pZ** field, click **Add** and close the box:

🖽 Uniform Load	
P 	
p (kN/m)	√ (Deg)
X: 0,00	0,0
Y: 0,00	0,0
Z: -10	0,0
Coord. system: 💿 Global 🤇	🔾 Local
Projected load	
Loads on eccentri	city
Add Close	Help

Figure 19 - Uniform load definition for DL2 case.

4. Select beams (no. 4 5 6 7) – indicate them or just type 4to7 in field Apply to and click Apply (as before its possible to window them too)

5. Press icon at the bottom of the screen to turn display Load value descriptions on.

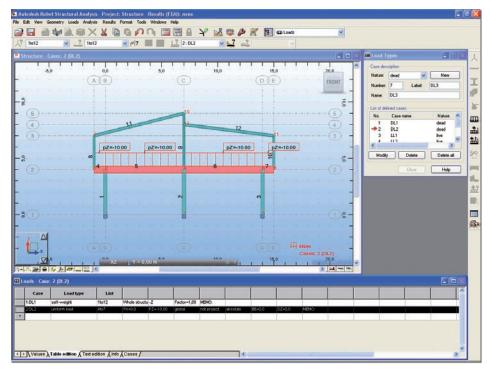


Figure 20 - DL2 load case view.

- 6. To define loads for LL1 case select the 3^{rd} load case in the list of defined load cases field.
- On the Bar tab press icon to open Bar Force dialog box, then type value of load -20 [kN] in the FZ field.
- 8. On the **Coordinate** field leave **x=0,50** value (defined force will be applied halfway along the member), click **Add** and **Close** the box.

🖽 Bar Force		
	F M	
Values F (kN) M (kN*m)	▽ (Deg)
X: 0,00	0,00	0,0
Y: 0,00	0,00	0,0
Z: -20	0,00	0,0
Coord. syste	m: 💿 Global Loads on eccent	O Local
Coordinate		
x = 0,50	Absolute	(x/L) (m)
Generate at the po	a calculation node int where a load is a	
Add	Close	Help

Figure 21 - Bar force definition for LL1 case.

9. Select beams (no.5) – indicate them or just type 5 in field Apply to and click Apply.

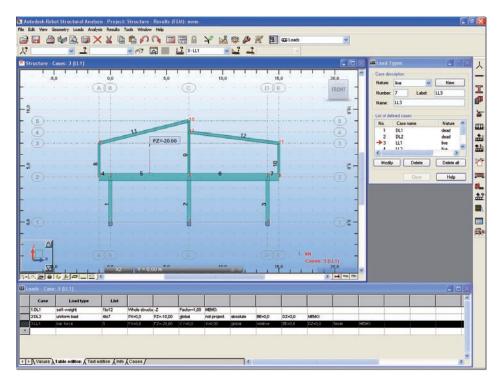


Figure 22 - LL1 load case view.

10. In the same way you may define for 4^{th} :LL2 load case bar force -30 [kN] on the half length of the bar No.6.

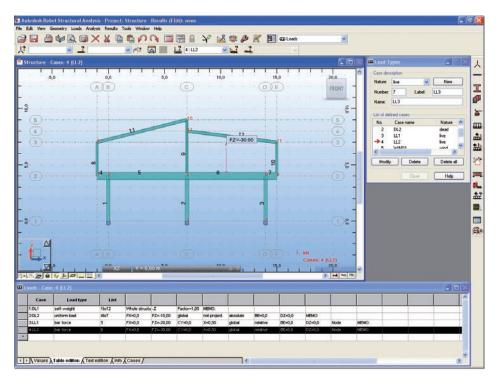


Figure 23 - LL2 load case view.

- 11. To define loads for **WIND1** case select the 5th load case in the list of defined load cases field.
- 12. On the **Bar** tab press icon to open **Uniform Load** dialog box, then type value of load **-1,5** [**kN**/**m**] in the **pZ** field, tick Coord. system **Local** on, then click **Add** and close the box.

🖽 Uniform Load	
P d Values	
p (kN/m)	▽ (Deg)
X: 0,00	0,0
Y: 0,00	0,0
Z: -1,50	0,0
Coord. system: 🔵 Global	 Local
Projected load	
Loads on eccent	tricity
Add Close	Help

Figure 24 - Uniform load definition for WIND1 case.

- 13. Select beam (no.1) indicate them or just type 1 in field Apply to and click Apply.
- 14. In the same way define uniform loads: **-2,50** [**k**N/**m**] for bar **no. 8** and **-3** [**k**N/**m**] for bar **no. 11**.

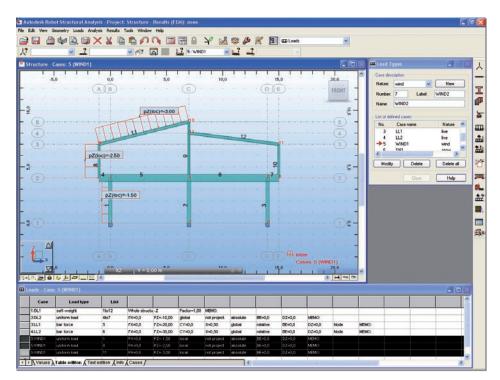


Figure 25 - WIND1 load case view.

- 15. To define loads for SN1 case select the 6^{th} load case in the list of defined load cases field.
- 16. On the Bar tab press icon to open Uniform Load dialog box, then type value of load -2,50 [kN/m] in the pZ field, tick Projected load on, then click Add and close the box.

🛄 Uniform Load	
p (kN/m)	굿 (Deg)
X: 0,00	0,0
Y: 0,00	0,0
Z: -2,50	0,0
Coord. system: 💿 Global	🔾 Local
Projected load	
Loads on eccentri	city
Add Close	Help

Figure 26 - Uniform load definition for SN1 case.

17. Select beams (no.11 and 12) – indicate them or just type 11 12 in field Apply to and click Apply.

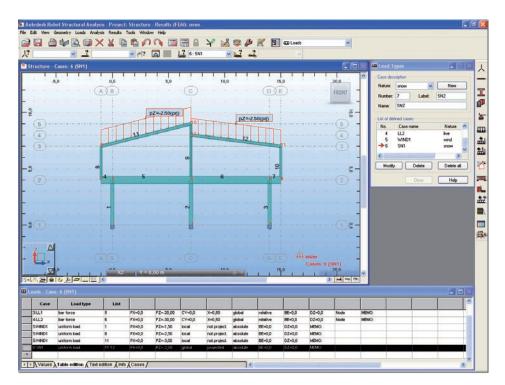


Figure 27 - SN1 load case view.

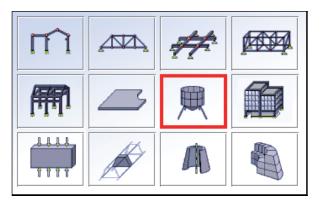
- 18. Close the **Load Definition** dialog box.
- 19. Go to the Robot layouts selection box (right top screen corner) and switch layout to **Start**.

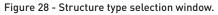
Now, using the existing 2D frame, we define a 3D frame structure with slabs and walls.

Changing the structure type

So far we have worked in 2D. In this step, we open new module **Shell** design to enable wall and slab definition and work in 3D.

1. Select **Geometry** > **Structure type** from the menu, which opens the window presented below:





2. Select **Shell** design icon (the third one from the left in the second row) to change structure type from 2D frame to shell. The program prompts you to save your work so far.

Definition of additional structural axis

In this step, w define additional structural axes on in the third axis - the Y direction.

- Press I icon (right side of the Robot screen) or select Geometry > Axis Definition... from the menu to open the Structural Axis dialog box.
- 2. Set **Numbering** as **Define** and type in the next field (on the right side) L1.

3. On the **Y** tab in the **Position** field type **0** and press **Insert** button, in the same way enter following axis coordinates as shown below:

📲 Structura	ıl Axis	🛛
Name:	Structure axis	*
Cartesiar	n Cylindrical	Arbitrary
	Advanced paramete	ers
X Y Position:	Z	Diterer
0,00	No. of repet.: (m) 0	Distance: 1 (m)
Label L1 L2 L3	Position 0.00 7.00 14.00	Insert Delete Delete all Single out
<		
Numbering:	Define 💌	L1
New	Axis	manager
Apply	Close	Help

Figure 29 - Definition of structural axis in Y direction.

4. Press Apply and close the Structural Axis dialog box.

Copying existing frame

In this step, we copy the 2D frame to generate a 3D structure. When we copy that frame, all attributes attached to it (loads, sections, supports etc) are also copied:

1. Press icon (top of the Robot screen) to open the View dialog box, then

or choose **View** > **Work in 3D** > **3D xyz** from the menu to select the isometric structure view (see picture below):

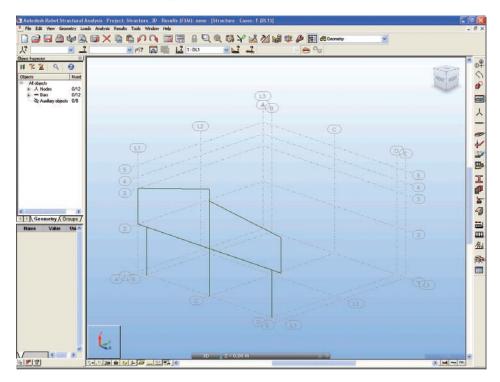


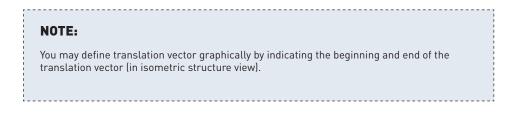
Figure 30 - 3D view of defined frame.

2. Press **CTRL+A** to select all of the structure.

- 3. Edit then → Translation – press this icons or select the Edit/Edit/Translate option from the menu to open the Translation dialog box.
- 4. In **Translation** vector type the value to copy in each axis as shown below:
- 5. In **Edit** mode tick option **Copy** on and **Number of repetitions** set as **2** (see below):

<mark>द</mark> Translation	
dX; dY; dZ = 0;7;0	
Numbering increment	
Nodes:	
© Copy	Drag
O Move	
Number of repetitions:	2
Execute Close	Help

Figure 31 - Copy of the frame - translaction vector definition.



6. Copied frames should look as shown below:

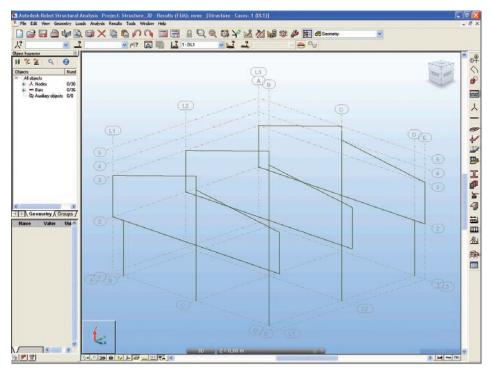


Figure 32 - Copied frame view.

Definition of lateral beams

In this step, we define beams to join the selected frames together:

- 1. Go to the Robot layouts selection box (right top screen corner) and switch layout to the **Bars**.
- 2. Click . icons at the bottom of the screen to turn node and bars numbers display off.
- 3. In Bars dialog box set Bar type: as Beam.
- 4. Define Section: as W8x15.

NOTE:

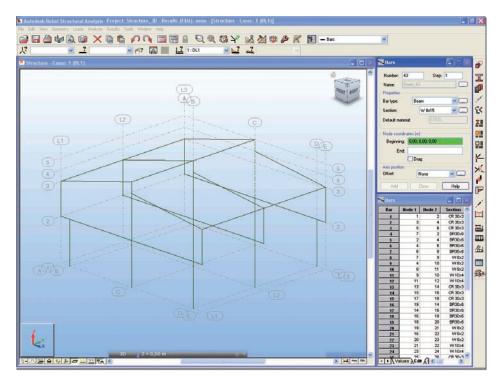
If the **W8x15** section is not available on the list, you should click the **(...)** button located beside the **Section** field and add this section from the database. In opened **New** section dialog box, in **Section** selection field for **Database: AISC** – choose **Family: W** then Section: **W8x15**. Click **Add** and **close** the box (see below)

I New Section		
Standard Parametric Tapere	ed Compound Sp	ecial Ax, Iy, Iz Variable 20,6 (cm)
W 8x15	Database:	AISC V
Color: Auto 🗸	American hot rolle Family:	ed shapes (13th Ed
G eTn	Wide-flange Shap	
L CT		8x15
Gamma angle: 0 🗸 (Deg) Section type:	Steel
Add Close	Help	STEEL

Figure 33 - New steel section definition.

- 5. Press Add new section type to be added to the active Sections list.
- 6. Close New Section dialog box.
- 7. Place the cursor in the **Node Coordinates** > **Beginning** field, then switch to the graphic viewer and select graphically the beginning and end of the bars by means of the coordinates of the intersection point of defined axes:
 - (E,L1,3) and (E,L2,3)
 - (E,L2,3) and (E,L3,3)
 - (C,L1,5) and (C,L2,5)
 - (C,L2,5) and (C,L3,5)
 - (A,L1,3) and (A,L2,3)
 - (A,L2,3) and (A,L3,3)

94



8. The structure with lateral tie beams should look as shown below:

Figure 34 - View of structure with lateral tie beams added.

9. Go to the Robot layouts selection box (right top screen corner) and switch layout to the **Geometry**.

Definition of slab

In this step, we learn how to define a slab, first we define a contour (shape) of the plate then we assign physical properties to it. Firstly, we need to define the elevation of the structure that we wish to work on – we do this with a "work plane".

- To set new a work plane press the icon provided in the bottom left corner of the screen which represents the active work plane a View dialog box will be opened.
- Press 2D and XY buttons and select "Structure axis 2" 4,00 (m) as shown below. Once these options are selected the structure is set on the XY plane at the recently defined Z coordinate (e.g. Z=12); only structure components from this plane are displayed.

۷	iew		×
Í	2D (2D/3D 3D	x y x	Close
	4,00 🔽 🔽	~	Help
1	"Structure axis 1" 0,00 (m)		
	"Structure axis 2" 4,00 (m)		
	"Structure axis 3" 7,00 (m)		
	"Structure axis 4" 8,00 (m)		
	"Structure axis 5", 9,00 (m)		

Figure 35 - New work plane definition (in axis no 2).

- 3. Close the View dialog box.
- 4. Press icon (right side of the Robot screen) or select **Geometry > Objects > Polyline-contour** from the menu to open the **Polyline-Contour** dialog box.
- 5. Indicate required points of the contour in the graphic viewer by cursor as follow: (A,L1); (E,L1); (E,L3); (A,L3); (A,L1) note, always select the start and end point to "close"the contour.

	2.00	~	- 10			Contrast of		-	र व	C ON	- T	22	3 1680	*	PE		Geometry						
Inspector III	?		-	· (%)?	A		1	: DL1	_	~	-				0	4	_		_			 	
T X Q @		4,0		-2,0		0,0		2,0	1	4,0		6,0		8,0		10,0		12,0		14,0	16,0	18.0	-
Lts Nun All objects All objects All objects All objects O/36 Bass O/46					A	B																TOP	1
& Austiliary objects 0/1	- 90	13)-			T	1				_			1									-0.3	1.0
	- 0'2												-										1 12.0
	-																						
	-0'01																						10,0
	h.												-										
	-8																			11 0 (1		-	8,0
Geometry & Groups ,	L.	12)-				1	- 11	- 74	- 77	- 14	- 14		100							1.1.1		-(-12	6.0
ie Value Un 🐣	-												-										° _
	-9																			-			¢-
	- 2																			1			-
	2,0																			-			2,0
	La	11.)-				-			1.4	14			-	1	- 16							+ -(-L1	8-
	-	4	4			1	10							a.									-

6. Close the **Polyline-Contour** dialog box, the defined contour should look as shown below:

Figure 36 - Defined contour of panel view.

7. Press icon (right side of the Robot screen) or select **Geometry** > **Panels** from the menu to open the **Panel** dialog box.

INFO

A surface in Robot is called a "panel". So if you want to create a slab or wall, you must define an appropriate panel.

8. In the **Panel** dialog box set parameters: **Panel** in **Contour type** field, **Internal point** (Creation with field), **none** (Reinforcement field), **TH12_CONCR** (Thickness field) and **Shell** (Model field) - see picture below:

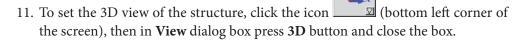
🛰 Panel		
Number:	44	
Contour type —	O Opening	
Creation with Internal poin	ł	
		(m)
📃 Only the	current selection	
O Object list		,
O Surface elen	nents (FE) list	ʻ
Properties		
Reinforcement:	none	✓ …
Material:	CONCR	
Thickness:	TH12_CONCR	✓ …
Model:	Shell	•
Add	Close	Help

Figure 37 - Panel definition settings.

NOTE:

In the **Reinforcement** option you can define a reinforcement type. However this option is unavailable in Robot Free, in the Slab design module, in the full commercial version of the program, you can perform a complete design process of an RC slab (including calculation of provided reinforcement and deflection verification with reinforcement and cracking taken into consideration).

- 9. To assign the recently defined panel to the selected contour set the cursor in the **Internal Point** field then move it to the graphical viewer and click on a point located within the area of the slab.
- 10. Close the Panel dialog box.



- 12. Press A, i icons at the bottom of the screen to turn on appropriate display of supports and sections shapes.
- 13. The defined structure with slabs should look as shown below:

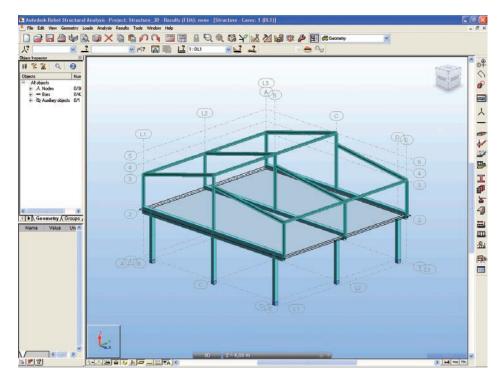


Figure 38 - Structure with slab view.

Offset definition

In this step we define offset between axis slab and RC beams:

- 1. Press icon at the bottom of the screen to open **Display** dialog box.
- 2. Select the **Panels/FE** option (left panel) and tick **Panel thickness** on (right panel) to turn displaying thickness of panels on.

Display		
Template: Standard	v 🕽 🔁	
Favorites Model	Name	
Nodes	Panel description	
Bars Devolution	Panel contours	
Panels / FE Mark with colors	🛨 Filling the interior	✓ #
Loads	Panel thickness	🛛
View (Open-GL version) View (version without Open	Characteristic points of panels	L kì 🛨
Structure	Contour components	✓ #
	Numbers and labels of edges	
	Reinforcement direction	- +
	Direction of load distribution	- +
< >	+ Finite elements	V 🔁
	Emitters	
Symbol size: 30	Display attributes on selected objects	ly for
Help	OK Cancel	Apply

Figure 39 - Switch thickness of panel display on.

3. On the structure drawing, which appears on the screen we can see that neutral axis of the RC beams and RC slab are on the same level with no eccentricity by default.

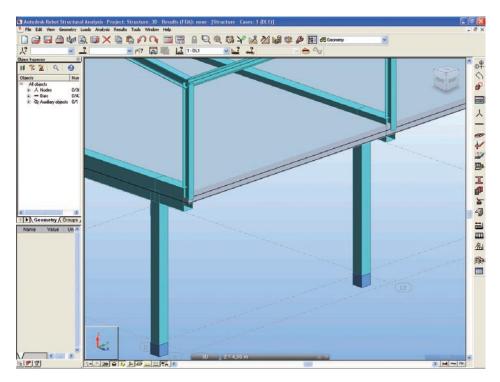


Figure 40 - Default view of RC beams and RC slab on the same level with no eccentricity.

- 4. Select Geometry > Additional Attributes > Offsets... from the menu then press
 New offset definition button to open the New Offset dialog box.
- 5. In the New Offset dialog box set parameters: Offset_1 in Label field, -45,24 [cm] (this is half of RC slab height plus half of RC beams height) in the Beginning UZ: and End UZ: fields and tick Global (Coordinate system field) on see picture below:

New Of	fset									
Absolute										
Label:	Offs	set_1								
Values-	Values Beginning: End:									
UX:	0,00	0,00	(cm)							
UY:	0,00	0,00	(cm)							
UZ:	-45,24	-45,24	(cm)							
Coordina	Coordinate system									
OLoc										
O Loo	🔘 Local - translated coordinate system									
Add	Close	e 🗌 🗌	Help							

Figure 41 - New offset (between beams and slab) definition.

- 6. Add and close the New Offset dialog box.
- 7. Press icon (left,top corner of the Robot screen) to open the **Selection** dialog box.

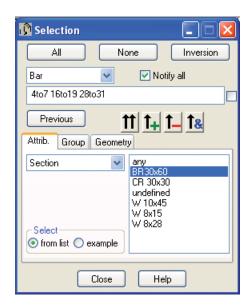


Figure 42 - Selection of RC beams by filters.

- 8. Indicate **Section** as selection criterion and click **B R30x60** in the listbox above.
- 9. Press the **1** button to enter defined selection to the edit field and mark red concrete beams, to which offsets will be attributed. **Close** the box.
- 10. In the **Offsets** dialog box place the cursor in the **Current Selection** field numbers of selected beams will appear.
- 11. Press Apply and Close to finish offset definition.
- 12. To display defined offset beams press icon at the bottom of the screen to open **Display** dialog box.
- 13. Select the **Favorites** option (left panel) and tick **Offsets** on (right panel), then press **Apply** and **OK** result should look as shown below:

Display		
Template: Standard	v 🕽 🕞 🛯] 🗙 🔀
Favorites Model	Name	
Nodes	Node numbers	□ 1
Bars	🛨 Bar description	<u> </u>
Panels / FE Mark with colors	 Panel description 	
Loads	Supports	
View (Open-GL version) View (version without Open	Section - shape	🗹 🌰
Structure	🛨 Local systems	
	+ Releases	1011
	Offsets	
	+ Load symbols	
< >	Load values	
	F Structural axes	
Symbol size: 30	Display attributes on selected objects	ly for
Help	OK Cancel	Apply

Figure 43 - Switch offsets display on.

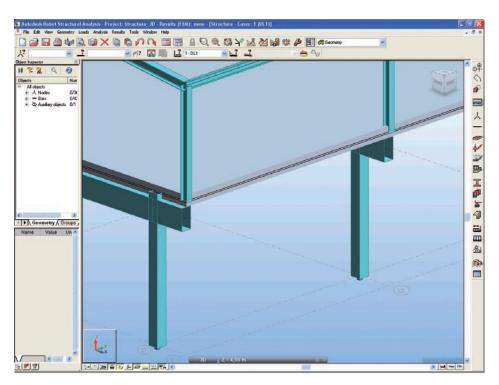


Figure 44 - View of RC beams and RC slab with offset defined.

Definition of a wall

In this step, we learn how to define a wall, first we define a contour then we assign physical properties to it:

- To set a new work plane press the icon provided in the bottom left corner of the screen View dialog box will be open.
- 2. Press **2D** and **YZ** buttons and select **"Structure axis D" 14,00 (m)**. Once these options are selected the structure is set on the YZ plane at the recently defined X coordinate (e.g. X=40); only structure components from this plane are displayed.

View	×
2D 2D/3D 3D XY YZ XZ	Close
14,00 🗸 🗸 🗸	Help
"Structure axis A" -1,00 (m) "Structure axis B" 0,00 (m) "Structure axis C" 7,00 (m)	
"Structure axis D" 14,00 (m) "Structure axis F" 15.00 (m)	

Figure 45 - New work plane definition (in axis D).

- 3. Close the View dialog box.
- 4. Press A, V icons at the bottom of the screen to turn display of supports symbol and sections shape off.
- 5. Press icon (right side of the Robot screen) or select **Geometry > Objects > Polyline-contour** from the menu to open the **Polyline-Contour** dialog box.
- 6. Indicate required points of the contour in the graphic viewer by cursor as follow:
 - (14; 0; 0)
 - (14; 2; 0)
 - (14; 2; 3)
 - (14; 4; 3)
 - (14; 4; 0)
 - (14; 10; 0)
 - (14; 10; 3)
 - (14; 12; 3)
 - (14; 12; 0)
 - (14; 14; 0)
 - (14; 14; 4)
 - (14; 0; 4)
- 7. Press Apply and close the Polyline-Contour dialog box.

- B -20 -H 656 ۷ es? 0 0.1 14,0 0,5 5 0/3 0/4 0/2 4 2(4) 112 3 ٢ AN BH BK+ - 2 2 : 0.2 Geometry Groups 副日島 Value Unit -20 龠 2.0 E # 2 34
- 8. Defined contour should look as shown below:

Figure 46 - Contour of defined panel view.

icon (right side of the Robot screen) or select Geometry/Panels from 9. Press the menu to open the Panel dialog box.

10. In the Panel dialog box set parameters: Panel in Contour type field, Internal point (Creation with field), none (Reinforcement:) and Shell (Model field).

🕿 Panel		
Number:	46	
 Panel 	🔘 Opening	
Creation with Internal poin	ł	
		(m)
📃 Only the	current selection	
O Object list		
O Surface eler	nents (FE) list	
Properties		
Reinforcement:	none	✓ …
Material:	CONCR	
Thickness:	TH12_CONCR	✓ …
Model:	Shell	✓ …
Add	Close	Help

Figure 47 - Panel definition settings.

11. To define thickness of the panel press ____ button (right to the **Thickness** field), then in the **Th** = field type **30** (cm), in Label: field name **TH30,00** will be automatically updated, **Add** and **close** the **New Thickness** dialog box (see below).

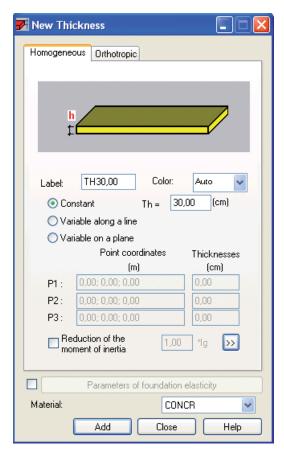


Figure 48 - New panel thickness definition.

- 12. To apply chosen panel properties to the recently defined contour, in the **Panel** dialog box, set the cursor in the **Internal point** field, then move the cursor to the graphical viewer and indicate a point within the boundaries of the panel contour.
- 13. Close the box.

Definition of wall support

In this step, we create a line support underneath the wall.

- 1. Press icon (right side of the Robot screen) to open the **Supports** dialog box.
- 2. Go to the Linear tab then select the Fixed support type.

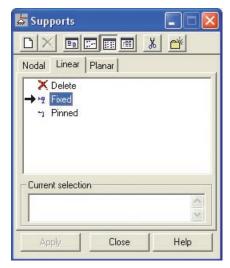
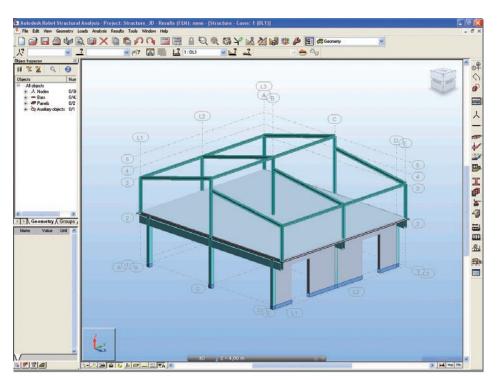


Figure 49 - Linear support of the wall definition.

- 3. Switch to the graphical viewer, point at the bottom slab edges and click once (when it is highlighted) on the edge.
- 4. Close the box.
- 5. To set 3D view of structure click the icon (bottom left corner of the screen), then in **View** dialog box press **3D** button and close the box.
- 6. Press A, i icons at the bottom of the screen to turn on display of supports symbol and sections shapes.



7. Defined structure with slab and wall should look as shown below:

Figure 50 - View of entire structure (with slab and wall defined).

Meshing parameters definition

In this step, we learn how to mesh the panels, since Robot is finite element solution. Robot has extremely powerful meshing algorithms to cope with virtually any shape of structure or mesh requirement.

NOTE:

The meshing of the plates and shells are made automatically (with standard parameters), after starting the calculation process. However it is often desirable to mesh the structure manually - often to obtain better quality meshing we should to modify the standard settings.

Options of FE Mesh Generation then Meshing Options – click these icons or select the Analysis > Calculation model > Meshing Options option from the menu to open the Meshing Options dialog box.

utodes	k Robot Structural Analysis Professional
1	No panels selected for this operation. Do you want to perform these operations for all panels

- 2. Select Complex mesh generation (Delaunay) to choose meshing method.
- 3. Type **0,5** [m] to define size of finite elements (as shown below):

🖺 Meshing Opti	ons ? 🔀
Meshing methods	Method parameters
- Available meshin	ig methods
🔵 Simple mesh	generation (Coons)
 Complex mes 	h generation (Delaunay)
🔘 Automatic se	lection of a meshing method
Mesh generation	
 Automatic 	O User
 Element size 	
0,5	(m)
Mest	n of volumetric elements
Fine	Coarse
Additional m	eshing of solid surface
	Advanced options
ОК	Cancel Help

Figure 51 - Meshing method and parametrs definition.

- 4. Click **OK** to close **Meshing Options** dialog box.
- 5. From Options of FE Mesh Generation toolbar click Generation icon or select the Analysis > Calculation model > Generation option to generate the mesh.
- 6. Close the **Options of FE Mesh Generation** toolbar.
- 7. Meshed slab and wall should look as shown below:

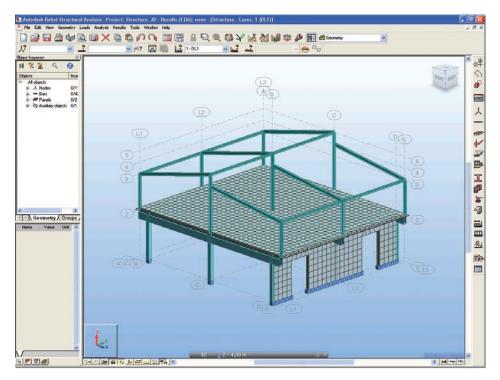


Figure 52 - Structure with meshed panels view.

INFO

Meshing parameters may be defined separately for each panel. This is s times useful if the Engineer is interested in results in a set area and can therefore mesh this area more densely than other areas.

Definition of slab loads

In this step, we define additional loads to be applied to the slab.

- 1. Press icon (right side of the Robot screen) to open the Load Types dialog box.
- 2. Choose in the **Nature** field live, then **New** to define a live load with a standard name **LL3**.
- 3. To define loads for LL3 case select this load case in the List of defined cases field.
- 4. Close the Load Types box.
- 5. Press icon (right side of the Robot screen) to open the Load Definition dialog box.
- 6. On the **Surface** tab press icon to open **Uniform Planar Load** dialog box, then type value of load −10 (kN/m2) in the pZ field, click **Add** and **close** the box.
- 7. In the **Load Definition** dialog box set the cursor in the **Apply to** field, then indicate panel (click when it is highlighted).
- 8. Press Apply and close Load definition dialog box.

STRUCTURE ANALYSIS

Here we start the analysis process, but firstly we will define load case combinations:

 Select Loads > Manual Combinations... option from the menu to open the Combination Definition > Modification dialog box. Leave Combination name and Combination type as set default.

🖺 Combination Defi	nition/Modific 🔀
Combination number:	8
Combination <u>n</u> ame:	COMB1
Combination type:	ULS 💌
Seismic combination ty	- Norman and Anna and Anna
€ <u>C</u> QC € S <u>R</u> SS	C 25 <u>M</u> C <u>1</u> 0%
Natu <u>r</u> e:	dead 💌
🔲 Quadratic combina	tion
<u> </u>	ose Help

Figure 53 - Definition of combination type.

2. Click **OK** button to open **Combinations** dialog box.

🖹 Combinat	tions				
<u>C</u> o	mbination: 8 : COM	B1 : ULS	•]	
Case list:			List of ca	ses in con	nbination:
Nature: All	-		Factor	No.	Case name
1 D 2 D 3 LL 4 LL 5 W	_2 /IND1	<u>> </u>			
Eactor:	auto tor definition		<		>
New	1 (elete	Apply	Close	Help

Figure 54 - Definition of cases in combination.



3. In this case we use automatic definition of factors (option auto in Factor field)

so in the Combinations dialog box press	button to transfer all defined load
cases (left panel) to the list of cases in combin	nation (right panel):

Combinations				
Combination: 8:CO	MB1 : ULS	1	·	
Case list:		List of ca	ases in cor	mbination:
Nature: All		Factor	No.	Case name
No. Case name		1.20	1	DL1
Nu. Case name		1.20	2	DL2
		1.60	3	LL1
		1.60	4	LL2
	1	1.60	5	WIND1
	<	0.50	6	SN1
< >	<<	1.60	7	LL3
Eactor: auto				
Factor definition		<		>
<u>N</u> ew <u>C</u> hange	D <u>e</u> lete		Close	Help

Figure 55 - List of cases in combination with automatically assigned factors.

- 4. **Apply** and **close** the box.
- 5. Defined load combination **COMB1** was added to the list of defined load cases:

Autodesk Robot St File Edit View Ge		al Anaty Loads A				_		its (Fi	:M): no	one -	Stru	cture	- Ca	ses: 1	(DL1)	11	
					5	Q			6	Q	•	<u>RQ</u> QA	Y	8		6	墩
以	-	?		• 6	§?	A		12	1 : DL1			-	?	2			
Dbject Inspector	X								1 : DL1 2 : DL2 3 : LL1								
Objects	Nun								4: LL2 5: WIN	D1							
🖅 🙏 Nodes	0/6!								6:SN1 7:LL3								
🛨 – Bars 🛨 – ┛ Panels	0/4:						Ų2		8 : COM Simple I		D.						
	0/2						1		Combin		N	at.					

Figure 56 - List of defined load cases updated by load combination COMB1.

6. **Calculations** – click this icon or select the **Analysis** > **Calculations** option from the menu to start the calculation process.

7. Once the calculations are completed the information: **Results (FEM): Available** should be displayed at the top of the screen.

INFO

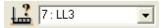
A key strength of Robot is the possibility to define a wide range of analysis types (linear static, non-linear geometry and material, buckling, modal analysis, harmonic analysis, seismic analysis, time history analysis etc.) If the user wishes to see these possibilities, he/she may look in the **Analysis** > **Analysis Types** > **Change analysis** type pull down menu. However, for this simple example, we will just assume the default linear static type of analysis.

RESULTS PREVIEW

Panels results in map form

In this step, we learn how to display calculation results on panels as "maps" for selected load cases:

1. In the list of defined load cases choose the load case for which the results will be displayed:



- 2. Select **Results > Maps...** from the menu to open **Maps** dialog box.
- 3. In the **Detailed** tab check **Displacements u,w** for **z** direction box.
- 4. Tick **With FE mesh** on (bottom part of the box):

😂 Maps						×
Parameters S Detailed	cale De Principa		natio 	n C Com		
	Layer : mi	ddle				
	Direction	х			[
1 P.I.		xx	уу	ху	z	
Stresses - s			Г	Г		
Membrane force	es - N		Г	Г		
Moments - M		Γ.	Г	Г		
Shear stresses -	t	Γ.	Г			
Shear forces - G	į	Γ	Г			
Displacements -	u,w		Г			
Rotations - R		Ε	Г			
Soil reactions - I	<				E	
		_	_			
smoothing within a	a panel				-	
 Isolines Maps 		-		norm. FE m	alization esh	
C Values ☑ Open new wir	idow with :			desci played		
Apply	Close		1		lelp	1

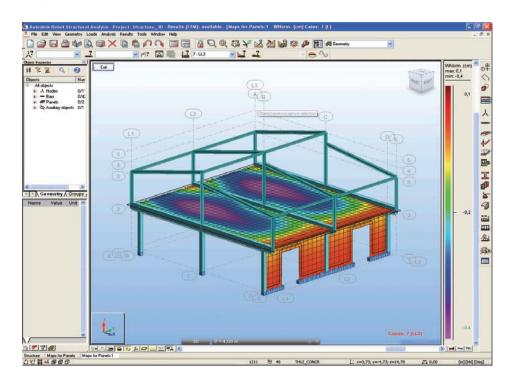
Figure 57 - Parameters of maps on panels definition.

INFO

To display scale of colors, on the **Detailed** tab tick the "Open new window with scale" displayed box . On the **Scale** tab you can change parameters of map presentation (color palette, scale type), in this example, choose **Color palette: 256 colors**.

5. Click **Apply** and **close** the **Maps** dialog box then enlarge the graphical viewer (by

stretching) and click **Zoom All** icon to maximize model view.



6. Map of deflection of the plate should look as shown below:

Figure 58 - Panels displacements in z direction in map form.

7. Press **Exit** to close additional window. To switch off the map display, go back to the maps dialogue box, remove the tick and apply

NOTE:

In the similar way, maps that show other values available from the **Detailed** dialog box can also be displayed.

Deformation of the structure

In this step, we learn how to display deformation of the structure.

- 1. Press 🗾 icon at the bottom of the screen to turn displaying sections shape off.
- 2. Select **Results > Diagrams for Bars...** from the menu to open **Diagrams** dialog box.

🔁 Diagrams 📃 🗆 🔀
NTM Deformation Stresses Reactions
Deformation
Exact deformation for bars
Deformation in structure scale
Diagram scale for 1 (cm)
0,1 (cm)
Animation Number of frames: 10
Number of frames/second: 8 🗢
Start
All None Normalize
Diagram size: + 🕂
Open a new window Constant scale
Apply Close Help

3. On the **Deformation** tab tick **Deformation** box on, then press **Normalize** button (to auto scale). Deformation of the structure should look as shown below:

Figure 59 - Presentation parameters of structure deformation.

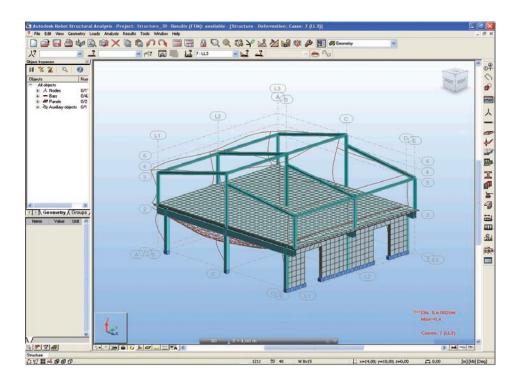


Figure 60 -Deformation of the structure.

NOTE:

In a similar way, diagrams that exhibit other values available from the **NTM** dialog box can be viewed (see below):

🔀 Diagrams					
NTM Deformation Str	esses Reactions				
Diagram scale for 1 (cm)					
Fx Force	(kN)				
Fy Force	(kN)				
Fz Force	(kN)				
Mx Moment	(kN*m)				
My Moment	(kN*m)				
Mz Moment	(kN*m)				
Elastic ground reaction					
📕 📃 Ky Reaction	(kN/m)				
📕 🗌 Kz Reaction	(kN/m)				
All None Normalize					
Diagram size: + 🕂					
Open a new window Constant scale					
Apply Clo	se Help				



4. Tick off **Deformation** and **Apply**, then **close** the **Diagrams** dialog box.

Results on panels in tabular form

In this step, you learn how to display calculation results as tables:

1. Select Results > Panel and Shell Results from the menu to open FE Results dia-

log box, or click **Tables** icon (toolbar on the right side of the screen) and tick **Results for Plates and Shells** option on, then **OK** to close the box.



Figure 62 - Selection of data to tabular presentation.

- 2. Click the right mouse button and select **Table Columns** option to open **Results for Plates and Shells** dialog box.
- 3. In the **Detailed** tab tick the following options in:
 - Membrane forces N: in direction xx
 - Shear forces Q: in direction xx
 - **Displacements u,w:** in direction **z** (as shown below):

Autodesk Robot Structural Analysis - Getting Started Guide

Results for finite e	lements		
Results for finite e Detailed Principal Cor Layer : Lay Directi Stresses - s Membrane forces - N Moments - M Shear stresses - t Shear forces - Q Displacements - u,w Rotations - R Soil reactions - K All	mplex Paramete ver middle		OK Cancel Help Filters Extremes Cases
FE Results For the active table, colo r will be added		this tab -	sting ones

Figure 63 - Table content composition.

4. Click the **Direction X** button and in the opened **Selection of Direction** dialog box tick the **Automatic** option in:

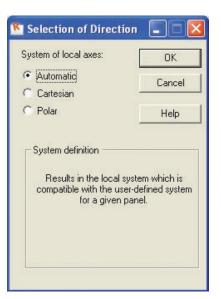


Figure 64 - Definition of the X direction of the local co-ordinate system of panels.

INFO

Selection of Direction window allows the user to define the main direction of the co-ordinate system (more precisely, to define the X-axis), which will be utilized by the user during surface FE result presentation. In this case we using **Automatic** option so the local element system will be automatically adopted according to the system defined for each of the panels.

- 5. Press **OK** to close **Selection of Direction** dialog box then **OK** to close the **Results for finite elements** box.
- 6. In the list of defined load cases choose the **8:COMB1** for which the results will be displayed, the results for selected values in tabular form is shown below:

🚅 🖩 🌰 🕼				(🕼 大 🕅	S 22 19 18		Geometry	~
110791 790 794 79 🤘 👱	11044.47 🛩 🔊		8: COH81	 A A				
Napector 📰	Panel Hode Case	MXX (klimina)	MYY (klimina)	MXY (kilmim)	IDX dillino	0000 dkillimä	Wilorm. (cm)	
F & K W	44/ 2/ 8.(C)	23,74	44,59	2,33	305,32	-12,96	-0,1	
s Nur	44 4 8 (C)	60,59	64,09	-7,35	1162,38	-0,52	-0,1	
d objects	44 6/ 8 (C)	4,07	-15,17	-12,43	242,90	11,04	-0,0	
A Nodes 0/1"	44 7/ 8.(C)	2,74	20,21	0,83	67,22	10,98	-0,0	
- Bars 0/4:	44 8 8 (C)	2,35	-21,23 105,02	-10,32	216,23 376,97	-0,50	0,1	
& Aunitary objects 0/1	44 14 8 (C) 44 14 8 (C)	123,41	221,20	0,00	1104,21	4,29	-0,2	
of runny topon of t	44 18 8 (C)	10,92	-9,90	0,05	226,10	16,60	-0,0	
1	44 19 8 (C)	24,15	155,54	0,00	42,92	15,36	-0,0	
	44 24 8 (C)	3,29	-2,70	0,02	173,20	-4,97	0,1	
	44 24 8 (C)	23,74	44,59	-2,33	305,32	-12,06	-0,1	
	44/ 28/ 8.(C)	60,59	64,09	7,35	1162,37	-0,52	-0,1	
1	44 34 8 (C)	4,07	-15,17	12,50	242,44	11,03	-0,0	
	44 31/ B.(C)	2,74	20,21	-0,83	67,22	10,98	-0,0	
E Contraction International In	44/ 32/ 8 (C)	2,98	-21,16	10,34	218,72 35,61	-0,51	0,1	
	44/ 37/ 8 (C) 44/ 38/ 8 (C)	7,49	-14,15	0,89	33,33	-4,61	-0,2	
1	44 39 B (C)	-0.94	-43.08	-0,21	-28.35	-7,45	-0,4	
1	44 44 8 (C)	0,68	-41,44	0,49	-16,11	-1,99	-0,3	
	44/ 41/ 8 (C)	-5,69	-63,80	0,06	-33,16	-7,66	-0,5	
	44/ 42/ 8 (C)	-2,21	-63,78	0,17	-16,01	-4,96	-0,5	
	44 43 B (C)	-8,59	-77,67	1,21	-23,68	-8,28	-0,6	
-	44 44 B (C)	-4,00	-78,55	1,21	-10,76	-7,20	-0,6	
	44 45 8.(C)	-10,23	-04,93	2,97	-14,63	-0,63	-0,7	
	44/ 46/ 8 (C) 44/ 47/ 8 (C)	-5,01	-86,31 -05,73	3,06	-6,29 -9,11	-8,36 -0,00	-0,6 -0,7	
Geometry Groups	44 47 8 (C)	-5,41	-87,32	5,32	-3,69	-0,00	-0,7	
e Value Unit 🔿 🗖	44 49 8 (C)	-10.68	-80,14	7,16	-7,43	-8,73	-0,7	
	44 59 8 (C)	-5,29	-01,67	7,66	-2,90	-0.69	-0,7	
1	44 51/ 8 (C)	-0,56	-68,15	9,13	-9,39	-8,41	-9,7	
1	44 52 B (C)	-4,62	-69,37	9,70	-3,75	-7,06	-0,6	
1	44/ 53/ 8 (C)	-7,34	-49,68	10,67	-14,73	-7,83	-0,6	
1	44 54 8 (C)	-3,27	-50,26	11,33	-6,20	-6,21	-0,5	
-	44 55 8 (C)	-3,65	-24,54	11,40	-22,60	-7,11	-0,5	
	44 56 8 (C) 44 57 8 (C)	-0,99 2,17	-24,06 7,56	11,83	-0,90	-3,59 -6,12	-0,4 -0,4	
	44 57 8 (C) 44 58 8 (C)	2,70	9,82	10,67	-13.67	-0,12	-0,4	
	44 59 8 (C)	11.56	47,42	9,35	-21,24	-6.59	-0,3	
	44 64 8 (C)	0,61	\$1,06	6,60	-10,25	6,27	-0,2	
	44/ 61/ 8 (C)	26,61	96,14	5,59	38,20	-4,63	-0,2	
	44/ 62/ B (C)	19,66	105,07	0,33	45,83	26,33	-0,1	
	44/ 63/ 8 (C)	19,66	105,07	-0,33	46,83	26,33	-0,1	
	48 64 8 (C)	26,61	96,14	-5,50	38,20	-4,43	-0,2	
	44 65 B (C)	11,56	47,42	-9,35	-21,24	-6,59	-0,3	
	44 66 8 (C)	8,61	\$1,86	-6,68	-10,25	6,27	-0,2	
	44/ 67/ 8 (C) 44/ 68/ 8 (C)	2,17	7,58	-11,13	-29,53	-6,12	-0,4	
	44 69 8 (C)	-3,65	-24,54	-11,40	-22,68	-7,11	-0,5	
× -	44 79 8 (C)	-0,99	-24,06	+11,83	-9.90	-3,59	-0,4	
17/41	I Values (Envelope)			10.67	127	7.03	0.6	

Figure 65 - Example of data presentation in tabular form.

INFO

There are few tabs on the bottom of the window to display additional data such as **Envelope**, **Global Extremes** and info about **Panels**.

- 7. In the table, press right mouse button, then "convert to Excel" to extract all data to a spreadsheet
- 8. Close the **FE Results** table.

INTEGRATION OF AUTODESK ROBOT STRUCTURAL ANALYSIS WITH REVIT® STRUCTURE

This example of a five storey concrete building shows the importance of using a capable analysis solution as part of the BIM process. Such a model cannot be described as "architecturally demanding", yet the analysis model produced from Revit® Structure poses some challenges for a lot of structural analysis solutions - including non-rectangular openings, curved slab edges and cores. However, Robot is ideally suited to such structures and will directly calculate the Revit® Structure generated data, without the need to "hack" the geometry as would be necessary to satisfy the limitations of some other structural software, thereby compromising the overall BIM integrity.

Synopsis:

The purpose of this example is to show the ease of transfer of data from Revit[®] Structure to Robot and also to show how changes in the model made in Robot can be reflected again in Revit[®] Structure, thereby maintaining the integrity of the overall Building Information Model. In particular, this example looks at the import of floors and walls from Revit[®] Structure, in the form of finite element surfaces in Robot. In addition to showing the user how to interact with and display basic data in Robot, this example also highlights the display of FE results and making a basic FE mesh, plus amending the mesh to suit the Engineer's preferences

For this example, it is assumed that Autodesk Robot Structural Analysis 2010, plus the Extensions For Revit (Integration with Robot Structural Analysis) are installed on the same PC.

EXPORT REVIT MODEL TO ROBOT

Opening project in Revit® Structure

In this step, we open and examine the model in Revit[®] Structure:

1. Press icon to run the Revit[®] Structure program then open the Revit[®] Structure project: **Revit2Robot building example.rvt**:

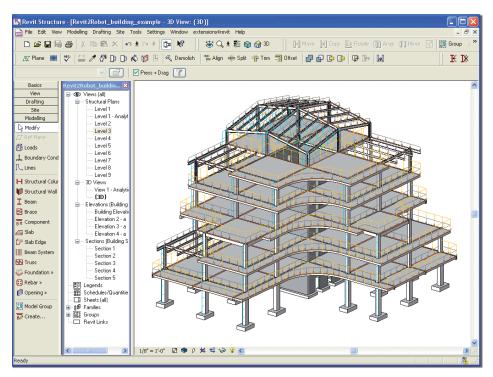


Figure 1 - Model of building in Revit Structure

- 2. Rotate it to show the type of elements it contains:
 - Sections:
 - steel (according to US AISC and also UK Steel section databases)
 - concrete beams and columns
 - Slabs, walls
 - Foundations:
 - isolated footings in Robot these will be treated as a fixed support,
 - continuous around wall base in Robot these
 - will be treated as a line support
 - Loads (Line and Area Load)

Sending data to Robot

In this step, we learn how to export the Revit model to Robot:

NOTE:

A useful feature is that only "selected" structure data in Revit® Structure is transferred to Robot – this allows only part of the model to be transferred to Robot (it could be useful if the user wants to select only part of a structure to analyze).

However, in this example, we want Robot to analyze all of the Revit® Structure data so it is important to ensure that either no data is selected or alternatively all of the structure is selected.

1. Go to the extensions4revit menu and select the Extensions Manager option:



Figure 2 - Extensions4revit menu

NOTE: Revit[®] Structure: **Modify** option should be selected in the **Basics** tab of the **Design Bar**.

2. Extensions Manager dialog box will appear:

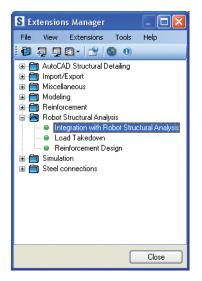


Figure 3 - Extensions Manager dialog box

- 3. To open **Integration with Robot Structural Analysis** double click the appropriate option (see above).
- 4. In dialog box shown below leave **Send model to Autodesk Robot Structural Analysis** option active (it is chosen by default) and click OK.



Figure 4 -Integration with Robot Structural Analysis (send model option)



5. During the transfer process the following splash screen is displayed:

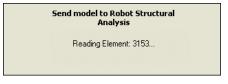


Figure 5 -Send model to Robot splash screen

6. Next, **Integration with Robot Structural Analysis – Send Options** dialog box will appear:

S Integration with Robot Structural Analysis - Send Options
Basic Options Additional Options
Scope and correction
 Send entire Revit project (ignore current selection)
 Send only current selection
Execute model correction in Robot
Specify the case that contains self-weight
 ● DL1
O Ignore self-weight
Bar end releases
O not use Revit settings
O Use Revit settings
Change Pinned-Pinned to Fixed-Fixed
Help About OK Cancel

Figure 6 -Integration with Robot Structural Analysis - Send Options dialog box

7. Leave default settings as shown above and click OK.

Integration with Robot Structural Analysis – Send Options dialog box allows you to configure export parameters

Basic Opitons tab:

Scope and correction group – enables the user to:

- Send entire Revit project (ignore current selection) sends whole model (this option is selected by default – it prevents exporting of an accidentally selected element or elements.
- Send current selection only sends only selected elements
- Execute model correction in Robot decides whether to run "drawing model correction" in Robot automatically, which will serve to join nodes to gether that fall within a certain tolerance

Specify the case that contains self-weight - option allows for defining which load case will be considered in RSA as a self-weight case

- If the first option is chosen, then User is able to specify Revit load case, which will be used in RSA for definition of self-weight. All loads defined in this case remain unchanged – self-weight will be added.
- It is possible to ignore definition of self-weight with using Revit load cases – by choosing second option.

Bar end releases group - allows the user to select the way of handling end releases:

- Do not use Revit settings (assign end releases in Robot) end releases are manually defined in Robot.
- Use Revit settings recognizes and assigns bar end releases defined in Revit[®] Structure
- Change Pinned-Pinned to Fixed-Fixed all Pinned-Pinned end releases defined in Revit[®] Structure are changed to Fixed-Fixed. This is often set to avoid mechanisms in the structural model

INFO

Integration with Robot Structural Analysis – Send Options dialog box allows you to configure export parameters

Additional Opitons tab:

Materials group - allows you to select the way of handling materials:

- Use Robot default materials for each element's material type (steel, concrete, timber) Robot will assign material from Robot database with standard properties
- Define new materials in Robot creates new materials in Robot with such physical properties as defined in Revit[®] Structure
- Select material of the best matching parameters assigns material properties from Robot data base with the nearest values to those defined in Revit[®] Structure

Curtain walls - option allows for setting sending parameters for Curtain walls. Panels created in RSA have a property (triangular method of load distribution in panel calculation model is set), which allows load distribution to other elements.

- Analytical model only (no system panels, no mullions) creates one panel in RSA without interior elements (system panels and mullions)
 – differences in singular panels are neglected
- Analytical model and mullions (no system panels) creates one panel in RSA (no division to system panels) but also mullions are transferred – differences in singular panels are neglected
- System panels and mullions (detailed model) sends all interior elements – system panels and mullions are created in RSA – differences in singular panels are considered

Transfer (optionally) - set of additional options of sending

- Use drawing model offsets as analytical an offset parameter is defined for bar elements in RSA according to Revit's definition
- Reinforcement projects (beams, columns, spread footings) defined reinforcement in listed elements is transferred to concrete design modules in RSA
- Steel connections option not available in current version

8. The Send model to Robot splash screen is displayed once again. Information about Robot launching appears. A progress information shows each step of the transfer process (such as reading of necessary Revit^{*} Structure data, creation of Robot project, export of elements and creation of corresponding Robot structural elements).

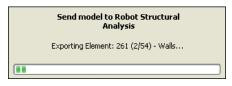


Figure 7 -Send model to Robot splash screen

NOTE:

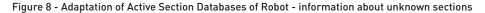
Occasionally sections may be defined in Revit[®] Structure that are not defined in Robot. The **Section Database Organizer** (see below) allows the user to add such databases into Robot without disturbing the transfer of data between programs.

 Adaptation of Active Section Databases of Robot dialog box may appear and display (upper, left corner of the box) information about Unknown sections (in this case there is one unknown section 178x102x19UB from the British section database).

NOTE:

Such a message may or may not occur on individual PC's depending on the language, preferences and databases that are used in Robot.

Ada	ptation o	of Active Section Databases of Robot				
Revil	on labels in Ro	I command was not able to create some bobt. You can select any available Ind add it to active section databases.				
AISC	·	Add to active databases				
	: (American hot section databa	rolled shapes (Jan 2006))				
	Database	Database name and description				
1	AISC	AISC (American hot rolled shapes (Jan 2006))				
	AISI	AISI (American cold formed shapes)				
	ARBU	Arbed_USA (ARBED - American section ranges)				
4	RUSER	UZYTKOWNIK (Baza danych z profilami uzytkownika)				
4						
<u>Help</u>		Apply Cancel				



10. In the **Available section databases** pull-down menu select **UKST** (British section database).

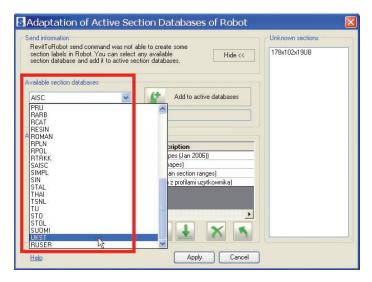


Figure 9 - Adaptation of Active Section Databases of Robot - available section databases pull-down menu

11. Click to **Add** to active databases button, **UKST** database will appear in the **Active section databases** window.

Send i Revi secti	information itToRobot send ion labels in Ro	of Active Section Databases of Robot Image: Control of Con
UKS	T (British hot ro	Add to active databases
1	Database UKST	Database name and description UKST (British hot rolled section)
3	AISC 3 AISI 4 ARBU 5 RUSER	Aroc (American not foiled shapes) Arbit (American cold formed shapes) Arbed_USA (ARBED - American section ranges) UZYTKOWNIK (Baza danych z profilami uzykownika)
	2	Apply Cancel

Figure 10 - Adaptation of Active Section Databases of Robot - information about added section database

- 12. Press Apply to close the window and continue exporting process.
- 13. After data transfer, we can see an events report, to do this click Yes.



Figure 11 - Events report after exporting data selection.

14. List of messages (and eventual warnings) will appear (see below):

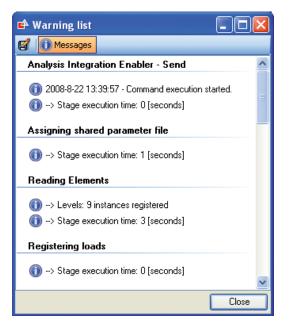


Figure 12 -Warning and messages list

15. Close the Warning list to finish sending data to Robot process.

STRUCTURE ANALYSIS IN ROBOT

NOTE: Change units from imperial to metric: Tools/Job Preferences/Units and Formats/Metric. You can check active units in right, bottom corner of the screen. In this example should be the display: [m] [kN] [Deg].

Exported Revit model of structure should appear in Robot as shown below:

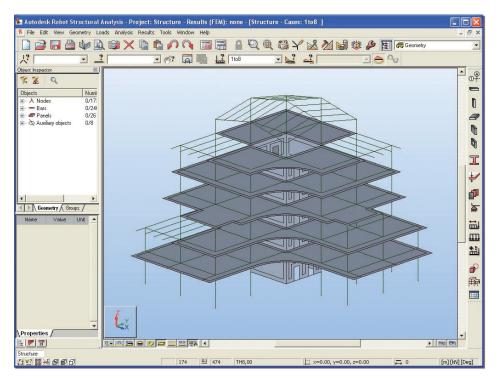
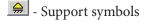
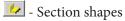


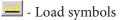
Figure 13 - Exported model of structure in Robot

Displaying items on the screen

In this step, we learn how to display additional attributes on the screen. By selecting icons of the toolbar located under the graphic field display a few more items can be displayed:







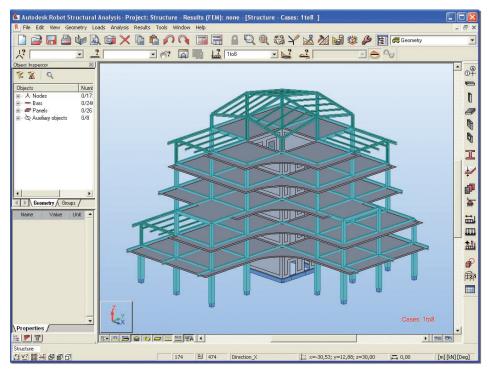


Figure 14 -Model of structure in Robot with additional display attributes

Presentation of load cases transferred from Revit[®] Structure

In this step, we display the load cases defined in Revit[®] Structure:

1. From the list of defined load cases choose **2: LL1**

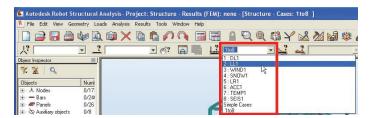


Figure 15 -List of defined load cases

2. Loads belonging to the case **2:LL1** case are shown in the graphic viewer (in the same way display next cases: **3: WIND1** and **4: SNOW1**):

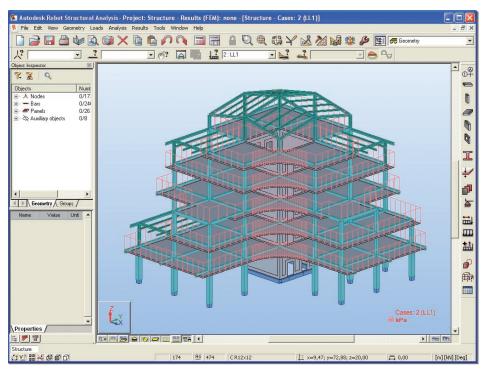


Figure 16 -2:LL1 Load case

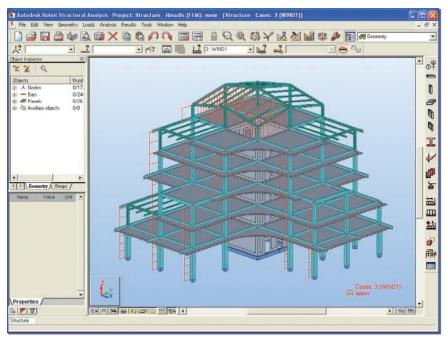


Figure 17 -3:WIND1 Load case

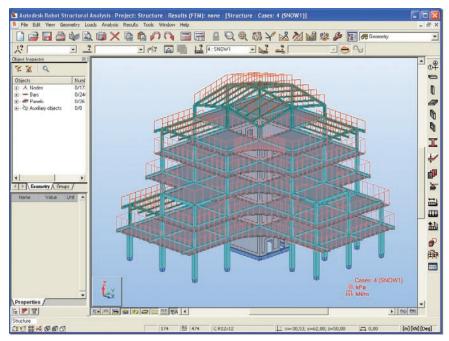


Figure 18 -4:SNOW1 Load case

- 3. To display numerical values of loads click 🕮 icon (bottom left corner of the screen).
- 4. Display the **View** submenu.
- 5. By means of 3D Orbit option from the **View** toolbar rotate the model to

show all the details: Profiles, Panels, Supports, Loads.

- 6. Click one more time the icons: 🔜 🕮 to turn loads display off.
- 7. Click one more time the icons: 🚔 🐓 to turn support and sections display off.

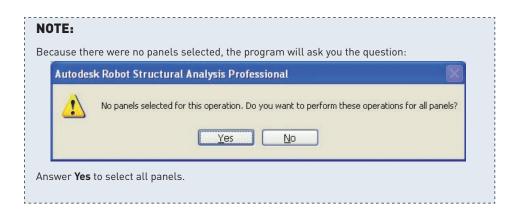
Meshing parameters definition

In this step, we learn how to mesh the panels.

INFO

The meshing of the plates and shells are made automatically (with standard parameters). A key strength of Robot is the ability to rapidly automesh virtually any shape of surface and to edit and refine the mesh as required.

Options of FE Mesh Generation then Meshing Options – click these icons or select the Analysis > Calculation model > Meshing Options option from the menu to open the Meshing Options dialog box.



- 2. Select Complex mesh generation (Delaunay) to choose meshing method.
- 3. Type **0,61** [m] to define size of finite elements (as shown below):

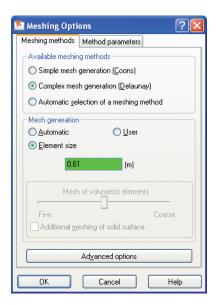


Figure 19 - Meshing method selection

- 4. Click OK to close Meshing Options dialog box.
- From Options of FE Mesh Generation toolbar click Generations icon or select the Analysis > Calculation model > Generation option to generate the FE mesh.

- 6. Close the **Options of FE Mesh Generation** toolbar.
- 7. Meshed slabs and core walls should appear as shown below:

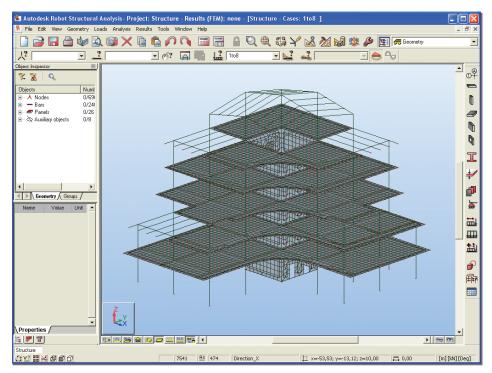


Figure 20 - Example of meshed structure

8. Press 🏝 icon at the bottom of the screen to hide the finite element mesh.



Calculations

Here we start the analysis process, but firstly we will tell Robot to make automatic code combinations (from any one of the list of Codes in **Job Preferences**):

 Select Loads > Automatic Combinations... option from the menu to open the Load Case Code Combinations dialog box. Select the Full automatic combinations option. The program will now automatically assign a number of combinations to find the most onerous load combination.

🖺 Load Case Code Combinations 🛛 🛛 🔀					
Combinations according to code:					
◯ None / Delete					
 Full automatic combinations 					
 Simplified automatic combinations 					
Manual combinations - generate					
Estimated number of combinations: 79					
Automatic generation of all combinations. Combination types (ULS, SLS) are grouped into composed cases whose components include subsequent combinations. Combination envelope cases (+/-) are also generated. It works only for linear cases and requires launching calculations.					
OK Cancel Help More >					

Figure 21 -Code combinations dialog box

- 2. Click OK button automatic calculation of code combinations will be done.
- 3. **Calculations** click this icon or select the **Analysis** > **Calculations** option from the menu to start the calculation process.
- 4. Once the calculations are completed the information: **Results (FEM): Available** should be displayed at the top of the screen.

INFO

A key strength of Robot is the possibility to define a wide range of analysis types (linear static, non-linear geometry and material, buckling, modal analysis, harmonic analysis, seismic analysis, time history analysis etc.) If the user wishes to see these possibilities, he/she may look in the **Analysis > Analysis Types > Change analysis type** pull down menu.

However, for this simple example, we will just assume the default linear static type of analysis.

Results preview - displaying panel results in map form

In this step, we learn how to display calculation results on panels as contour maps for selected load cases:

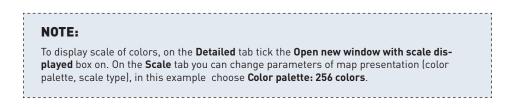
1. In the list of defined load cases choose the load case for which the results will be displayed:



- 2. Select **Results > Maps...** from the menu to open **Maps** dialog box.
- 3. In the **Detailed** tab check **Displacements u,w** for **z** direction box in.
- 4. Tick **With FE mesh** on (bottom part of the box):

🚭 Maps							
Parameters 9 Detailed	Scale Defor Principal	mation Cros Comple					
	Layer : middle						
	Direction X						
Stresses - s Membrane foro Moments - M Shear stresses Shear forces -	-t []	ГГ	z				
Displacements	a (19	E I					
Rotations - R Soil reactions -							
smoothing within a panel							
 G Isolines Maps C Values ☑ Open new w 	N N	With normaliz With FE mest With descript e displayed	1				
Apply	Close	Help					

Figure 22 - Maps for surface finite elements (panels) definition



Click Apply and Close the Maps dialog box then enlarge the graphical viewer (by 5.

Zoom All icon to maximize the model view. stretching) and click

6. Maps for selected value should look as shown below:

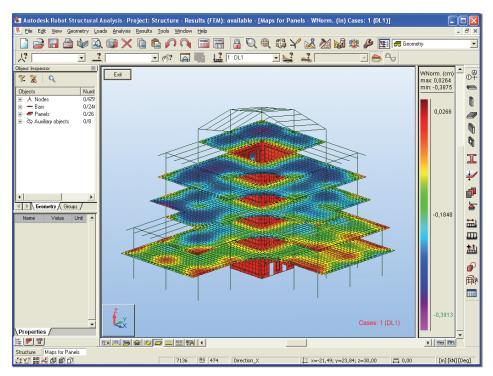


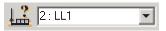
Figure 23 -Example of displacements presentation in map form.

- 7. Tick off the **Displacement z** check-box and click **Apply** to switch the maps off.
- 8. Close the **Maps** dialog box.
- 9. Click **Exit** button (upper, left corner of the Robot screen) to close the window with the scale.

Results preview - displaying results on bars in diagram form

In this step, we learn how to display calculation results on bars as diagrams for selected load cases:

1. In the list of defined load cases choose the load case for which the results will be displayed:



- 2. Press 🗲 icon at the bottom of the screen to turn off display of panel interiors.
- 3. Select Results > Diagrams for Bars option from the menu to open the Diagrams dialog box.
- 4. Select **MY Moment** (this shows the major axis bending moment on the beams) as shown below:

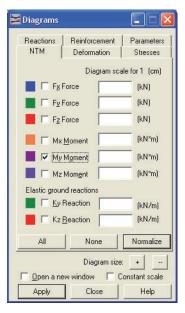


Figure 24 -Diagrams on bars definition (force selection)

5. On the **Parameters** tab tick **differentiated** option (on **Positive and negative values** field) on (see below) and return to **NTM** tab:

🞽 Diagrams						
NTM Reactions	Deformation Reinforcement	Stresses Parameters				
Diagram de		<u>A</u>				
● <u>n</u> one	C labels C					
⊻alues:	Local extrem	185 <u>▼</u>				
- Positive an	d negative values					
<u>undifferentiated</u> • differentiated						
Filling • fence	in filled					
□ Open a new window □ Constant scale						
Apply	Close	Help				

Figure 25 -Diagrams on bars definition (parameters of diagrams displaying)

6. On the **NTM** tab press **Normalize** button (to autoscale) to display bending moment diagrams for bars:

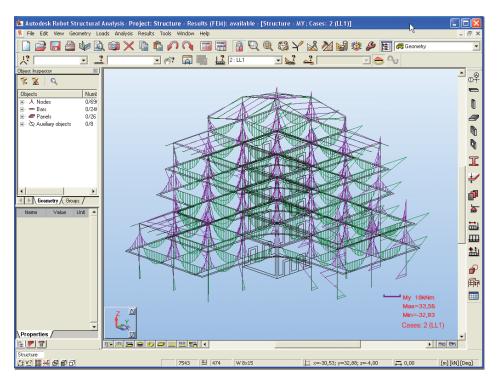


Figure 26 - Example of bending moment presentation in diagram form

- Tick off My Moment and apply to remove the diagram. 7.
- Close the dialog box. 8.
- 9. Press 🚔, 📂, 롣 icons at the bottom of the screen to turn on display of the supports, section shapes and panel interior.

MODIFICATION OF THE STRUCTURE IN ROBOT

Replacing sections

In this step, we learn how to replace section shapes:

- 1. Press icon on top toolbar to open the **Selection** dialog box.
- 2. Indicate Section as selection criterion and click 178x102x19UB.
- 3. Press the **1** button to enter defined selection to the edit field and mark red steel beams (see picture below):

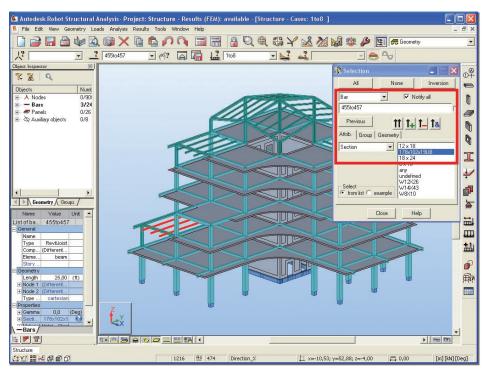


Figure 27 -Selection of beams by filters.

- 4. Go to the toolbar on the right side of the screen and click icon **Bar Sections** icon.
- 5. In Sections dialog box select W 8x10 section:

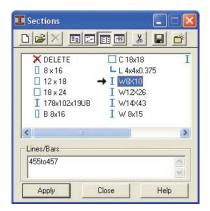
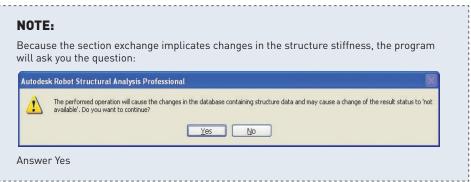


Figure 28 -Section selection.

Apply and Close the box. This changes all the selected sections to W 8x10 6.



In the same way we can replace concrete columns 18x24 by sections C18x18.

- 7. Press icon to open the **Selection** dialog box.
- Indicate Sections as selection criterion and click 18x24. 8.

9. Press the **b**utton to enter defined selection to the edit field and mark red concrete columns:

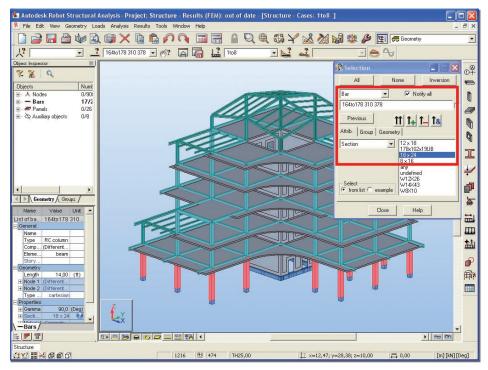


Figure 29 -Selection of columns by filters.

10. Click Close.

11. In the Sections dialog box choose C18x18 section as shown below:

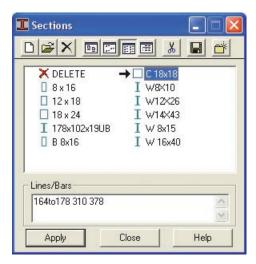


Figure 30 -Section selection.

- 12. Click Apply in Sections dialog box to give the new section type to bars
- 13. Close the Sections dialog box.

Deleting bars

In this step, we learn how to delete elements:

1. Indicate bars (just click bar when is highlighted) as shown below. To make a multi selection, click while holding down the ctrl button:

Autodesk Robot Structural Analysis - Getting Started Guide

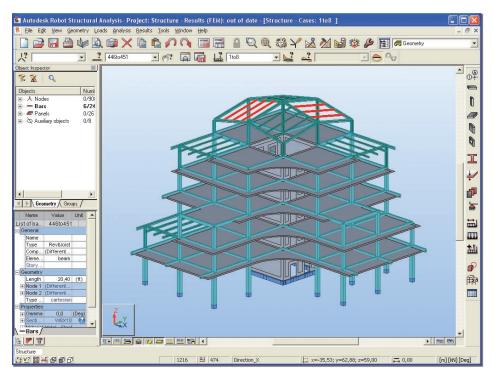


Figure 31 -Selection of beams by coursor.

Delete – press this icon or select the Edit > Delete option from the menu to delete selected elements.

Adding new elements

In this step, we learn how to create new elements (in this case we will add cross bracing):

- 1. Press the icon 🗾 to turn sections display off.
- 2. Press right mouse button on the graphics screen, then in the context menu select **Window** option:



3. Select by dragging window the view of the highest storey as shown below:

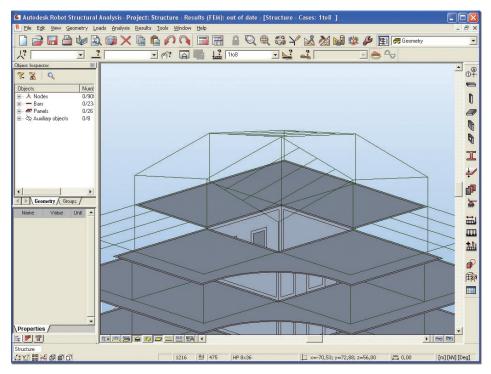


Figure 32 - Top storey view.

4. Select the **Geometry** > **Bars** option from the menu to open the **Bars** dialog box.

5. Set **Bar type:** as **Simple bar** (whatever is selected is not important for analysis, but affects the design parameters for subsequent member design, such a buckling length, position of restraint etc)

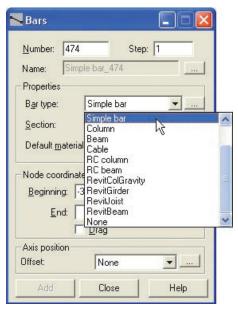


Figure 33 -Bar type definition.

- 6. Define Section: as L4x4x0.375.
- 7. If the **L4x4x0.375** section is not available on the list, you should click the (...) button located beside the **Section** field.
- 8. In New section dialog box, in Section selection field for Database: select AISC, for Family: select L then in the Section: set L4x4x0.375:

New Section	
Standard Parametric Tapere Label: L 4x4x0.375 Color: Auto	Image: Special Ax, Iy, Iz Image: Special Ax, Iy, Iz </th
<u>G</u> amma angle: 0	Section type: Steel
Add Close	Help

Figure 34 - New section definition

9. Click **Add** and **Close** the box.

INFO

There are many extra options that may be entered for fabricated members, tapering sections and also for beams that the user wants to define as able to exhibit plasticity.

- 10. Click the following points (their numbers are shown in the picture below) to draw new bracing:
 - 137 126 to create 1st bar
 - 124 125 to create 2nd bar
 - 126 136 to create 3rd bar
 - 127 128 to create 4th bar

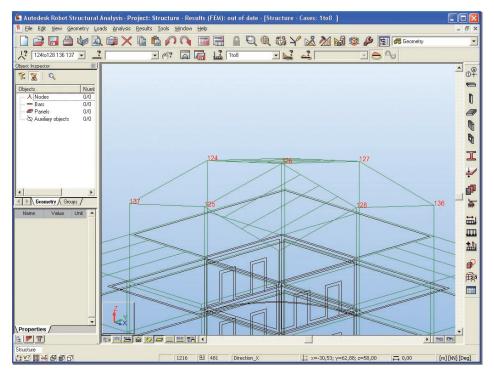


Figure 35 -Nodes numbers necessary to bracing definition.

- 11. Close the Bars dialog box.
- 12. Press the icon 🗾 to turn sections display on.
- 13. Bracings should look as shown below:

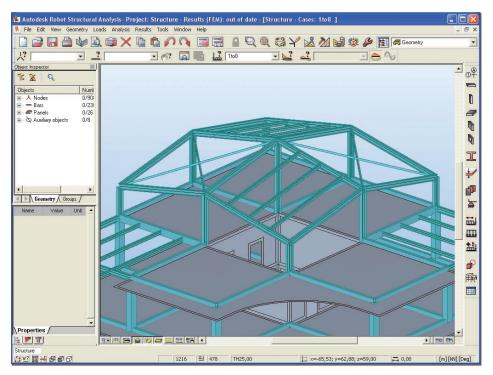


Figure 36 -View of defined bracings.

UPDATE REVIT MODEL FROM ROBOT

Updating Revit[®] Structure project

In this step, we learn how to update model data after analysis and changes in Robot:

- 1. Go back to Revit[®] Structure.
- 2. Choose the extensions4revit menu and select the Extensions Manager option:

Autodesk Robot Structural Analysis - Getting Started Guide

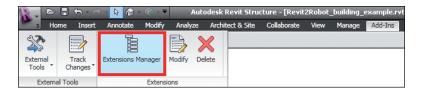


Figure 37 - Extensions4revit menu

3. Extensions Manager dialog box will appear (see below):



Figure 38 - Extensions Manager dialog box.

- 4. To open **Integration with Robot Structural Analysis** double click appropriate option (see above)
- 5. In dialog box shown below click **Update model from Autodesk Robot Structural Analysis** option and click OK:

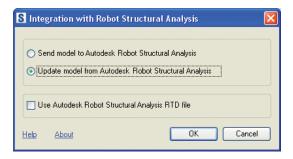


Figure 39 -Integration with Robot Structural Analysis (update model option).

6. Next, **Integration with Robot Structural Analysis – Update Options** dialog box will appear:

S	Integration with Robot Structural Analysis - Update Options 🛛 🔀					
	Scope - consider current selection Update the whole project (ignore current selection) 					
	O Update only the structure part selected in Robot					
	O Update only the structure part selected in Revit Structure					
	Select modified elements in Revit Structure					
Transfer (optionally)						
	Results (reactions and internal forces)					
	Reinforcement projects (beams, columns, spread footings)					
	Steel connections					
1	Help About OK Cancel					

Figure 40 -Integration with Robot Structural Analysis - Update Options dialog box

7. Leave default settings as shown above and click OK.

Autodesk Robot Structural Analysis - Getting Started Guide

INFO

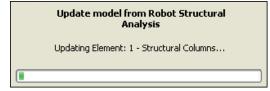
Integration with Robot Structural Analysis – Update Options dialog box allows you to configure update parameters

Scope – consider current selection – possibility of update whole structure or selected elements

- Update the whole project (ignore current selection) all elements will be taken into account in update process
- Update only the structure part selected in Robot only selected elements in RSA will be updated
- Update only the structure part selected in Revit Structure only selected elements in Revit will be updated
- Select modified elements in Revit Structure if some elements are new or changed then they will be selected in Revit after update process

Transfer (optionally)

- Results (reactions and internal forces) allows to transfer internal forces for Revit load cases in elements (option available only if the structure is calculated in RSA).
- Reinforcement projects (beams, columns, spread footings) defined reinforcement in listed elements is transferred from concrete design modules in RSA to Revit
- Steel connections option not available in current version
- 8. During sending process the following splash screen is displayed:



INFO

It is often the case that the user can generate sections in Robot that are not in the Revit® Structure database. Simple feature allows the user to add this database information without interrupting or abandoning the transfer of data.

9. Autodesk Robot Structural Analysis To Revit Update Log dialog box may appear:

1	🕼 Autodesk Robot Structural Analysis To Revit Update Log							×			
ſ	Sections	s not found in Revit	Sections succesfully updated					1			
		Section Revit Family		1	Robot Bars List			Replace With			
	1	L 4x4x0.375	Structural Framing		474	475 4	76 477		new bar	\$	
ł	To dupar	nicallu undate/create	e Revit elements select								
	and load	all necessary family	files. Additionally you ossible replacements for	12	2						
	section th	hat was not found in	current Revit project								
	M	ulti-select and load	Bevit Familu Files			Help			Apply	Cancel	-
			ricvici anily riles		1	Telp		<u> </u>		Cancer	

Figure 40 - Unknown section during updating process.

- 10. Click Multi-select and load Revit Family Files... button (bottom, left corner of the box).
- 11. In the opened dialog box select L-Angle.rfa family file (default localization: C:\ Documents and Settings\All Users\Application Data\Autodesk\RST 2008\Imperial Library\Structural\Framing\Steel) to load unavailable section family:

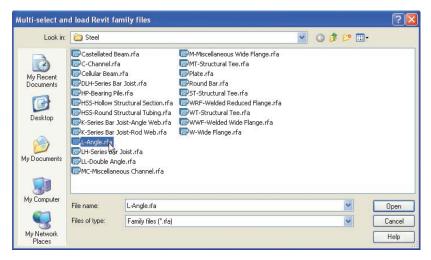


Figure 41 -Selection of missing Revit family file.

- 12. Press **Open** to open the database and this will close the window and continue the updating process.
- 13. The **Update model from Robot** splash screen is displayed once again. A progress information shows each step of the transfer process.
- 14. Autodesk Robot Structural Analysis To Revit Update Log dialog box will appear once again showing information about successfully updated sections:

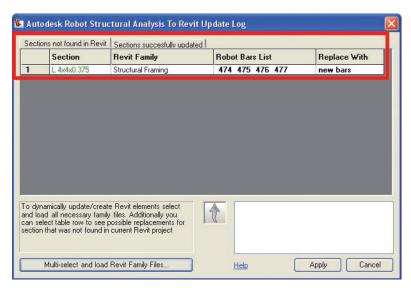


Figure 42 -Sections succesfully updated list.

- 15. Click **Apply** to close the box.
- 16. After finish sending data you can see an events report, to do this click Yes.

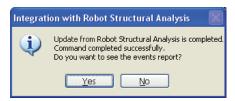


Figure 43 - Events report after updating data selection.

17. List of messages (and also warnings, if any will appear - see below):

Autodesk Robot Structural Analysis - Getting Started Guide

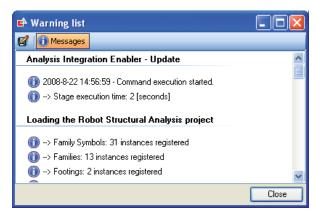


Figure 44 -Events report.

- 18. Close the box.
- 19. Updated model of structure should look as shown below:

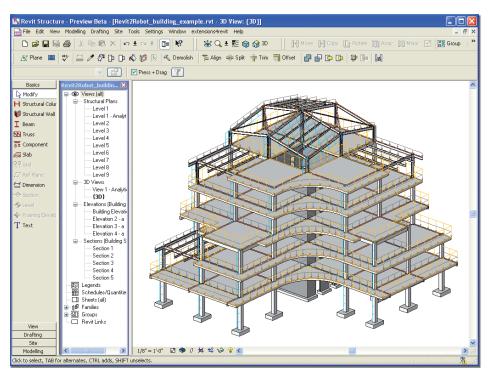


Figure 45 - Updated model of building in Revit Structure.

Model changes presentation

In this step, we can check and display changes in the model:

1. Zoom to view upper storey as shown below, notice that lateral beams **W 8x10** has been deleted and new bracings added (as changed in Robot):

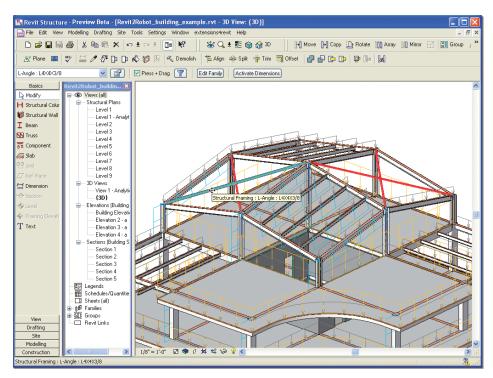


Figure 46 -View of added bracings.

- 2. Go with the mouse cursor to the one of the new beams and select it to check the section (L4x4x3/8 should be displayed).
- 3. Zoom to view second storey on the left side of the building as shown below: