

2D Frame Analysis

Structural analysis of beams, trusses and frames under static, dynamic, linear and non-linear loads


EngiSSol
Engineering Software Solutions



Contents

GENERAL.....	5
DISCLAIMER	7
SYSTEM REQUIREMENTS	10
PROGRAM FEATURES	11
Analysis.....	11
Pre-processing	11
Post-processing	12
EDITIONS COMPARISON	13
MAIN SCREEN LAYOUT	15
MAIN MENU DESCRIPTION.....	16
File Menu.....	16
Edit Menu	18
View Menu	20
Project Data Menu	21
Elements Menu	22
Draw Menu.....	24
Analysis Menu	25
Settings Menu.....	26
Help Menu	27
TOOLBAR ITEMS DESCRIPTION.....	28
Bottom toolbar	28
Vertical toolbar.....	29
Model page horizontal toolbar.....	30
Model page vertical toolbar	33
Results page toolbar.....	34
DESCRIPTION OF THE FORMS.....	36



Units	36
Preferences.....	36
Reset to factory default settings 	37
Project information	37
Material definition.....	38
Material from library	39
Cross section definition	40
Section from library.....	41
Frame element	42
Node	46
Static load case	47
Non linear static load case	48
Response spectrum case	49
Standard response spectrum function	50
User defined response spectrum function.....	51
Response spectrum function from file	52
Time history case.....	53
Sine time history function	54
Cosine time history function	55
Ramp time history function.....	56
Saw tooth time history function.....	57
Triangular time history function.....	58
User defined time history function	59
Time history function from file.....	60
Load combination.....	61
Calculation parameters	62
Frame results	64



2D FRAME ANALYSIS

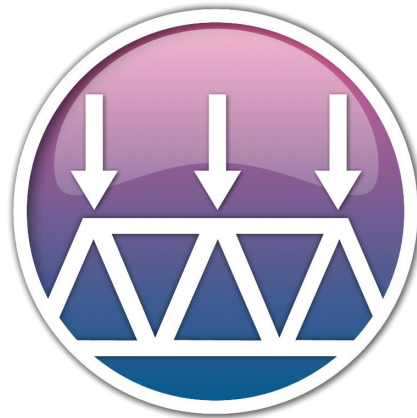
Structural analysis of beams, trusses and frames under static, dynamic, linear and non-linear loads

Node results	65
Results locations.....	66
Results tables.....	67
Report settings	68



User manual

GENERAL



2D Frame Analysis

Structural analysis of beams, trusses and frames under static, dynamic, linear and non-linear loads

EngiSSol
Engineering Software Solutions

Thank you for using ENGISSOL® products. We can assure you that you will be provided with a robust, powerful and reliable application of high standards and guaranteed accurate results, regarding the analysis of any kind of 2D structures (generic trusses, frames, beams etc.). EngiSSol's know-how with respect to engineering application development is incomparable, a fact that is proven by thousands of satisfied end-users around the world.



2D FRAME ANALYSIS

Structural analysis of beams, trusses and frames under static, dynamic, linear and non-linear loads

2D Frame Analysis™ performs structural analysis on beams, trusses, frames under static, dynamic, linear and non-linear loads. It uses the powerful and well-known analysis core “ENGISSOL – FEM Library” which has received many awards because of its flexibility in handling large-scale linear and non-linear arithmetic problems. Among the program’s capabilities are the calculation and illustration of internal forces diagrams, the deformed shape, the dynamic modes and the comprehensive representation of the results on an automatically created report.

2D Frame Analysis™ covers efficiently the design engineer's everyday needs due to its simplicity and the wide range of supported cases. Besides, it has been tested thoroughly to meet specific standards and is considered to be the best tool for the analysis of beam, frame and trusses on the market.

At ENGISSOL we are always looking for feedback on how to improve our products. You can contact us at engissol@engissol.com.



DISCLAIMER

IMPORTANT- READ CAREFULLY: This ENGINEERING SOFTWARE SOLUTIONS ("ENGISSOL") End-User License Agreement ("EULA") is a legal agreement between you, user of software applications, ("End User") and ENGISSOL for all ENGISSOL products, source code (where provided), demos, printed materials, and "online" or electronic documentation ("SOFTWARE PRODUCT(S)").

ENGISSOL grants to you a nonexclusive license to use the SOFTWARE PRODUCT(S) for the sole purposes of engineering and design calculations. By using the SOFTWARE PRODUCT(S), you agree to be bound by the terms of this EULA. If you do not agree to any part of the terms of this EULA, DO NOT USE, EVALUATE, OR REPLICATE IN ANY MANNER, ANY PART, FILE OR PORTION OF THE SOFTWARE PRODUCT(S).

All SOFTWARE PRODUCT(S) is licensed, not sold. By purchasing any of the SOFTWARE PRODUCT(S), you instantly acquire a license for the SOFTWARE PRODUCT(S) from ENGISSOL. The licenses do not expire unless explicitly stated otherwise. Each license of any SOFTWARE PRODUCT(S) can only be used on a single computer.

If the SOFTWARE PRODUCT(S) you have obtained is marked as a "**DEMO**" you may use one copy of the SOFTWARE PRODUCT(S) for testing purposes ("Evaluation Period"). DURING THE EVALUATION PERIOD YOU ARE NOT ENTITLED TO USE THE DEMO FOR COMMERCIAL PURPOSES.

ENFORCEMENT OF INTELLECTUAL PROPERTY RIGHTS. If the licensed right of use for this SOFTWARE PRODUCT(S) is purchased by you with any intent to reverse engineer, decompile, create derivative works, and the exploitation or unauthorized transfer of, any ENGISSOL intellectual property and trade secrets, to include any exposed methods or source code where provided, no licensed right of use shall exist, and any PRODUCT(s) created as a result shall be judged illegal by definition of all applicable law. Any sale or resale of intellectual property or created derivatives so obtained will be prosecuted to the fullest extent of all local, European and international law.

All ENGISSOL files remain ENGISSOL's exclusive property and YOU MAY NOT DISTRIBUTE ANY OF THEM.

1. GRANT OF LICENSE

This EULA, if legally executed as defined herein, licenses and so grants the single End User the following rights:

2. LIMITATIONS ON REVERSE ENGINEERING, DECOMPIATION, AND DISASSEMBLY

You may not reverse engineer, decompile, create derivative works, translate, or disassemble the SOFTWARE PRODUCT(S), and only to the extent that such activity is expressly permitted by applicable law notwithstanding this limitation. You agree to take all reasonable, legal and appropriate measures to prohibit the illegal dissemination of the SOFTWARE PRODUCT(S) or any of its constituent parts.



3. RENTAL

You may not rent, lease, or lend the SOFTWARE PRODUCT(S).

6. UPDATES

You must be properly licensed to use the SOFTWARE PRODUCT(S) identified by ENGISSOL as being eligible for the update in order to use the SOFTWARE PRODUCT(S). An update of SOFTWARE PRODUCT(S) replaces the SOFTWARE PRODUCT(S) that formed the basis for your eligibility for the update, and together constitutes a single PRODUCT(S). You may use the resulting updated PRODUCT(S) only in accordance with all the terms of this EULA.

If the SOFTWARE PRODUCT(S) contains new critical features and therefore constitutes a major update, additional costs may apply in relevance and in proportion to the new features.

Any supplemental SOFTWARE PRODUCT(S) provided to you as part of the Support Services shall be considered part of the SOFTWARE PRODUCT(S) and subject to the terms and conditions of this EULA.

8. COPYRIGHT

All title and copyrights in and to the SOFTWARE PRODUCT(S) (including but not limited to any copy written documentation, demos, incorporated into any copies of the SOFTWARE PRODUCT(S)) are owned by ENGISSOL. The SOFTWARE PRODUCT(S) is protected by copyright laws and international treaty provisions and therefore, you must treat the SOFTWARE PRODUCT(S) like any other copyrighted material except that you may install the SOFTWARE PRODUCT(S) as described in this EULA.

9. DISCLAIMER OF WARRANTY

ENGISSOL expressly disclaims any warranty for the SOFTWARE PRODUCT(S). THE SOFTWARE PRODUCT(S) AND ANY RELATED DOCUMENTATION IS PROVIDED "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NONINFRINGEMENT.

10. LIMITATIONS ON LIABILITY

To the maximum extent permitted by applicable law, in no event shall ENGISSOL be liable for any special, incidental, indirect, or consequential damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other pecuniary loss) arising out of the use of or inability to use the SOFTWARE PRODUCT(S) or the provision of or failure to provide Support Services, even if ENGISSOL has been advised of the possibility of such damages.

End User understands that the SOFTWARE PRODUCT(S) may produce inaccurate results because of a failure or fault by End User to properly use and or deploy the SOFTWARE PRODUCT(S). End User assumes full and sole responsibility for any use of the SOFTWARE



PRODUCT(S), and bears the entire risk for failures or faults within the SOFTWARE PRODUCT(S). You agree that regardless the cause of failure or fault or the form of any claim, YOUR REMEDY AND ENGISSOL'S OBLIGATION SHALL BE GOVERNED BY THIS AGREEMENT AND THAT IS THE FAULT'S RESOLUTION. IN CASE OF INEFFECTIVENESS OF THE ABOVE ACTION ENGISSOL'S LIABILITY SHALL IN NO EVENT EXCEED THE PRICE PAID TO ENGISSOL FOR THE SOFTWARE PRODUCT(S). This Limited Warranty is void if failure of the SOFTWARE PRODUCT(S) has resulted from accident, abuse, alteration, unauthorized use or misapplication of the SOFTWARE PRODUCT(S).

11. TERMINATION

Without prejudice to any other rights or remedies, ENGISSOL will terminate this EULA upon your failure to comply with all the terms and conditions of this EULA. In such events, you must destroy all copies of the SOFTWARE PRODUCT(S) and all of its parts including any related documentation, and must remove ANY and ALL use of such technology immediately from any applications using technology contained in the SOFTWARE PRODUCT(S) developed by you, whether in native, altered or compiled state.

12. MISCELLANEOUS

This EULA may only be modified in writing signed by an authorized officer of ENGISSOL. If any provision of this EULA is found void or unenforceable, the remainder will remain valid and enforceable according to its terms.

ENGISSOL reserves all rights not specifically granted in this EULA.

End User acknowledges that he or she has read this Agreement, understands it, and agrees to be bound by its terms and conditions.

Should you have any questions concerning this EULA, or if you desire to contact ENGISSOL for any reason, please contact us at engissol@engissol.com.



SYSTEM REQUIREMENTS

2D Frame Analysis ® is a 32- and 64-bit compatible application.

Your computer shall meet these minimum requirements so that 2D Frame Analysis can run properly.

Processor	
Minimum	400 MHz
RAM	
Minimum	96 MB
Disk space (minimum)	
32-bit	280 MB
64-bit	610 MB
Operating system	
Windows 10 (all versions) x86 and x64	
Windows 7 (all versions) x86 and x64	
Windows Vista (all versions) x86 and x64	
Windows XP (all versions) x86 and x64	
Windows Millennium Edition	
Microsoft Windows 98 Second Edition	
Microsoft Windows 98	



PROGRAM FEATURES

2D Frame Analysis is a powerful application that can perform a wide range of calculations on 2D structures (beams, trusses, frames etc.). The structure can be drawn directly using the versatile featured user interface. The user can also import standard steel sections from a complete shape library according to all major codes (AISC, Australian-New Zealand, BS, Chinese, European, Indian, Aluminum etc.) There are no limitations regarding the geometry of the structure, materials or loads, since the program can handle any arbitrary 2D structure under static, dynamic, linear and non-linear loads. Among its capabilities, 2D Frame Analysis can calculate and graphically illustrate deformations, internal forces, dynamic modes and other analysis results. A large material library is also available according to almost all concrete, steel, timber, aluminum etc. material specifications. Apart from these, the user can define custom material and cross section data for use in the structure that is modelled.

The program features can be found below comprehensively:

Analysis

- ✚ Use of highly flexible, general, finite element method
- ✚ Parallel processing algorithms for optimal analysis performance
- ✚ All boundary supports (roller, hinge, fixed etc.)
- ✚ Spring supports (translational and rotational)
- ✚ All types of member end-releases, including internal partial spring stiffness for semi-rigid end releases
- ✚ Timoshenko elements for the consideration of shear deformations
- ✚ End length offsets for the simulation of member connections
- ✚ Linear static analysis of all types of structures (trusses, beams, frames, etc.)
- ✚ Incremental non-linear static analysis (second order effects) of all types of structures (trusses, beams, frames, etc.) with many stiffness matrix update options
- ✚ Modal analysis for the estimation of dynamic modes, periods, frequencies, etc. considering the mass source (loads, self-mass etc.) specified by the user
- ✚ Both lumped and consistent mass matrix
- ✚ Time History analysis of all types of structures (trusses, beams, frames, etc.)
- ✚ Response Spectrum analysis of all types of structures (trusses, beams, frames, etc.)
- ✚ Unlimited count of load cases
- ✚ Unlimited count of load combinations using many combination options (Add, Envelope, Absolute, SRSS)

Pre-processing

- ✚ All major systems of units are supported (Metric, US, User defined)
- ✚ Top quality graphics rendering
- ✚ Full GUI including zoom, pan, grid, snap options
- ✚ Every user action can be done graphically at real time
- ✚ Library with all major steel section shapes



- ✚ Library with all major concrete and reinforcement grades, as well as other materials such as cold formed/hot rolled steel etc.
- ✚ Definition of custom materials and cross sections
- ✚ Application of nodal loads
- ✚ Application of element loads
- ✚ Unlimited span loads
- ✚ Uniform and trapezoidal loads
- ✚ Thermal loads
- ✚ Local, Global and Projected global coordinate systems for the application of loads

Post-processing

- ✚ Comprehensive and versatile reporting of the results
- ✚ Automatic creation of high-quality, editable calculation report that can be exported to Word (doc), Excel (xls) and Adobe PDF (pdf) formats.
- ✚ Graphical representation of the support reactions
- ✚ Graphical representation of the deformed shape of the structure
- ✚ Creation of fully customizable graphs of internal forces (Axial force, Shear force, Bending moment) for each load case and load combination
- ✚ Graphical representation of the dynamic modes of the structure
- ✚ Representation of the results in tabular form that is exportable to Excel



EDITIONS COMPARISON

2D Frame Analysis ships in 3 different editions, the Truss Edition, the Static Edition and the Dynamic Edition. The features of the editions are represented below.

Feature	Truss Edition	Static Edition	Dynamic Edition
Use of optimized frame finite elements	✓	✓	✓
Parallel processing algorithms for optimal analysis performance	✓	✓	✓
All boundary supports available (roller, hinge, fixed, etc.)	✓	✓	✓
Spring supports	✓	✓	✓
End releases including partial spring stiffness	✓	✓	✓
Both Timoshenko elements (for shear deformation effects) and classical Euler-Bernoulli elements	✓	✓	✓
End length offsets for the simulation of member connections	✓	✓	✓
Linear static analysis	✓	✓	✓
Incremental non-linear static analysis (second order effects) of all types of structures (trusses, beams, frames, etc.) with many	✓	✓	✓
Modal analysis for the estimation of dynamic modes, periods, frequencies, etc. considering the mass source (loads, self-mass			✓
Both lumped and consistent mass matrix			✓
Time History analysis			✓
Response Spectrum analysis			✓
Unlimited count of load cases	✓	✓	✓
Unlimited count of load combinations using many combination options (Add, Envelope, Absolute, SRSS)	✓	✓	✓
All major systems of units are supported (Metric, US, User defined)	✓	✓	✓
Top quality graphics rendering	✓	✓	✓
Full GUI including zoom, pan, grid, snap options	✓	✓	✓
Library with all major steel section shapes	✓	✓	✓
Library with all major concrete and reinforcement grades, as well as other materials such as cold formed/hot rolled steel etc.	✓	✓	✓



2D FRAME ANALYSIS

Structural analysis of beams, trusses and frames under static, dynamic, linear and non-linear loads

Definition of custom materials and cross sections	✓	✓	✓
Application of nodal loads	✓	✓	✓
Application of element loads	✓	✓	✓
Unlimited span loads	✓	✓	✓
Uniform and trapezoidal loads	✓	✓	✓
Thermal loads	✓	✓	✓
Local, Global and Projected global coordinate systems for the application of loads	✓	✓	✓
Automatic creation of high-quality, editable calculation report that can be exported to Word (doc), Excel (xls) and Adobe PDF	✓	✓	✓
Graphical representation of support reactions, deformed shape, axial force, shear force and bending moment diagrams	✓	✓	✓
Graphical representation of the dynamic modes of the structure	✓	✓	✓
Representation of the results in tabular form that is exportable to Excel	✓	✓	✓
Only pinned connection supported	✓		

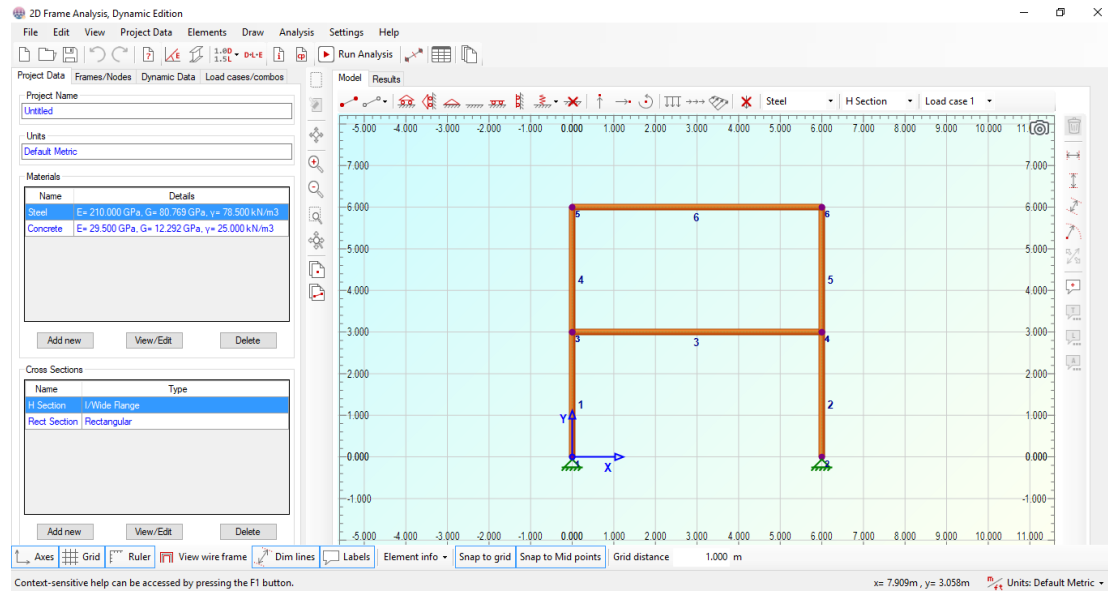


2D FRAME ANALYSIS

Structural analysis of beams, trusses and frames under static, dynamic, linear and non-linear loads

MAIN SCREEN LAYOUT

The main screen layout 2D Frame Analysis is illustrated below.





MAIN MENU DESCRIPTION

File Menu

New

Creates a new empty file.

New from template

Creates a new file from the available templates

New from existing

Creates a new file adopting all properties, except geometry, of another specified file.

Open

Opens a file from disk.

Open sample project

Shows a dialog with all sample projects that can be opened.

Import Text file

Imports a text file.

Save

“Save” will save the current file under the same name in the same location. It saves the changes over the original file that has been opened.

Save As

“Save As” will always bring up a window allowing the user to change the file name and its location in disk. Assuming you make a change to name or location, the original file will still be intact in its original location after the “Save As”.

Export to Text file

Exports the model to a text file. Results are not exported.

Capture picture

The user can take a picture of what is on the drawing screen by selecting this command under the File. A dialog will open where the user can specify the location and format (jpg, bmp, gif) of the image.

Recent projects



All recent projects that have been opened or saved lately will appear here. They can be directly opened by clicking on them without using the “Open” command.

Exit

Closes the application and asks for saving in case of unsaved data in current file.



Edit Menu

Edit -> Select submenu

All

Selects all elements and nodes.

By Material

Selects the elements that have been assigned with the specified material.

By Cross Section

Selects the elements that have been assigned with the specified cross section.

All Dimension Lines

Selects all existing dimension lines.

All Labels

Selects all existing dimension labels.

Boxed selection

The user can draw a rectangular window using two points. If the window is defined rightwards (from left to right), all objects which lie entirely within it, will be selected. If a window is defined leftwards (from right to left), only objects which lie entirely within the window or cross the window border, are selected.

Clear selection

Deselects all objects. This action can be invoked with the Esc button as well.

Delete

Deletes all selected objects.

Change Dimension Line Location

Changes the offset of a dimension line (straight or radial). The user shall first click on this command and afterwards click on a dimension line in order to modify its offset.

Modify label Text

Changes the text of a label. The user shall first click on this command and afterwards click on a label in order to modify its text.



Modify Location of text label

Changes the location of the text of a label. The user shall first click on this command and afterwards click on a label in order to modify its text location.

Modify Arrow location of label

Changes the location of the arrow of a label. The user shall first click on this command and afterwards click on a label in order to modify its arrow location.

Undo

This command allows the user to undo a number of recent actions in a file.

Redo

This command allows the user to redo a number of recent actions in a file, which have been undone previously.



View Menu

Pan

Panning allows you to quickly move around the drawing area at the same magnification you currently have set.

Zoom In/Out

The Zoom In/Out commands allow the user to change the display size of a drawing on the computer screen.

Zoom Window

The Zoom Window command zooms to the rectangle defined by the user in the drawing screen.

Zoom Extends

The Zoom Extends command zooms to the edges of the drawing.



Project Data Menu

Project information

This command displays a form where the project information (Project Name, Engineer Name, Date etc.) is available.

Add new Material

By clicking this item, a form opens where a new material can be specified or imported from the library.

Add new Cross section

By clicking this item, a form opens where a new cross section can be specified or imported from the library.

Add new Response Spectrum function

By clicking this item, a new response spectrum function can be defined. (Available in the Dynamic Edition only)

Add new Time History function

By clicking this item, a new time history function can be defined. (Available in the Dynamic Edition only)

Add new Load Case

By clicking this item, a new load case can be defined.

Add new Load Case

By clicking this item, a new load combination can be defined.



Elements Menu

Elements-> Add new Frame Element submenu

Fixed-Fixed

Adds a new frame element with rigid ends (no releases). The element is inserted graphically by the user.

Fixed-Pinned

Adds a new frame element with rigid starting end and pinned ending end. The element is inserted graphically by the user.

Pinned-Fixed

Adds a new frame element with pinned starting end and rigid ending end. The element is inserted graphically by the user.

Pinned-Pinned

Adds a new frame element with pinned both starting and ending ends. The element is inserted graphically by the user. This option is automatically applied to all new elements in the Truss Edition of the program.

Add new Frame Element manually

Adds a new frame element by providing its coordinates. The user can specify all related properties of the element.

Assign Material to Selected Frames

This command is only available when at least one element has been selected. Using these options, the user can massively assign a new material to all selected elements.

Assign Section to Selected Frames

This command is only available when at least one element has been selected. Using these options, the user can massively assign a new cross section to all selected elements.

Copy/paste node assignments

Using this option, the user can transfer the chosen properties of a source node including material, section, section placement, end length offsets, releases (only available in Truss Edition) and loads to another destination frame.

Copy/paste frame assignments



Using this option, the user can transfer the chosen properties of a source frame including supports, Masses (only in Dynamic Edition) to another destination node.

Properties of selected frame or node

This command is only available when only one element or node has been selected. “Properties” command displays the geometric data, material properties etc. of the selected entity.



Draw Menu

Linear Horizontal Linear Dimension Line

Using this command, the user can draw a straight horizontal dimension line.

Linear Vertical Linear Dimension Line

Using this command, the user can draw a straight vertical dimension line.

Linear Dimension Line

Using this command, the user can draw a straight dimension line.

Radial Dimension Line

Using this command, the user can draw a radial dimension line.

Label

Using this command, the user can insert a description label.



Analysis Menu

Calculation Parameters

This command opens a form where the Calculation Parameters can be viewed or modified.

Run Analysis

Performs the structural analysis considering the specified load cases

Specify Results Location

Opens a form where the user can specify the location of each frame element to be included in the calculation report. The user can override the default locations and add additional ones.

Result tables

Opens a form where all analysis results are represented in tabular form and are exportable to Excel for further processing.

Calculation report

Creates the calculation report according to the properties specified by the user.



Settings Menu

Set Units

This command shows the “Units definition” form, where the user can choose between a metric, US or fully customized set of units for current project.

Preferences

This opens the “Preferences” form, where the view options including colors and fonts/sizes can be viewed or modified.

Reset to factory default settings

This option restores all user-defined preferences (colors, fonts, view options etc.) to their default values.



Help Menu

User Manual

This command displays the Help file of 2D Frame Analysis.

Online worked examples

This command displays a document with same basic, step-by-step, worked examples for 2D Frame Analysis.

Check for updates

Connects to the internet and searches for an updated version of 2D Frame Analysis.

Go to website

Opens your internet browser and displays the webpage of 2D Frame Analysis.

Find on facebook

Opens your internet browser and displays the page of 2D Frame Analysis on Facebook.

Follow on twitter

Opens your internet browser and displays the page of 2D Frame Analysis on Twitter.

YouTube channel

Opens your internet browser and displays the YouTube channel of 2D Frame Analysis, where demonstrative videos, samples and other useful information can be viewed.

License type

Specifies the license type (standalone or network).

Remove License

Removes all licensing information and reverts the program back to demo.

About

Displays the “About” screen of the application, where installed version and your machine ID can be found.



TOOLBAR ITEMS DESCRIPTION

Bottom toolbar



Toggle button that specifies if axes are visible.



Toggle button that specifies if grid is visible.



Toggle button that specifies if ruler is visible.



Toggle button that specifies if the elements of the structure are displayed wired.



Toggle button that specifies if dimension lines are visible.



Toggle button that specifies if labels are visible.

Element info

Specifies the element information to be displayed (Materials, Sections, Supports Frame number, Loads, Local axes, End releases, Rigid offsets).

Snap to grid

Toggle button that specifies if snap to grid function is enabled.

Snap to Mid points

Toggle button that specifies if snap to element middle points is enabled.

Grid distance

Specifies the grid distance.



Vertical toolbar

Boxed Selection

The Zoom Window command zooms to the rectangle defined by the user in the drawing screen.

Clear Selection

Clear current selection.

Pan

Panning allows you to quickly move around the drawing area at the same magnification you currently have set.

Zoom In/Out

The Zoom In/Out commands allow the user to change the display size of a drawing on the computer screen.

Zoom Window

The Zoom Window command zooms to the rectangle defined by the user in the drawing screen.

Zoom Extends

The Zoom Extends command zooms to the edges of the drawing.

Copy/paste node assignments

Using this option, the user can transfer the chosen properties of a source node including material, section, section placement, end length offsets, releases (only available in Truss Edition) and loads to another destination frame.

Copy/paste frame assignments

Using this option, the user can transfer the chosen properties of a source frame including supports, Masses (only in Dynamic Edition) to another destination node.



Model page horizontal toolbar

Add frame element

Adds a new frame element with rigid ends (no releases). The element is inserted graphically by the user.

Add new Frame Element manually (provide coordinates)

Adds a new frame element by providing its coordinates. The user can specify all related properties of the element.

Fixed-Pinned

Adds a new frame element with rigid starting end and pinned ending end. The element is inserted graphically by the user.

Pinned-Fixed

Adds a new frame element with pinned starting end and rigid ending end. The element is inserted graphically by the user.

Pinned-Fixed

Adds a new frame element with pinned both starting and ending ends. The element is inserted graphically by the user. This option is automatically applied to all new elements in the Truss Edition of the program.

Delete frame element

Deletes a frame element by clicking on it.

Add horizontal roller

Assigns a horizontal roller to the specified node.

Add vertical roller

Assigns a vertical roller to the specified node.

Add hinge

Assigns a hinge to the specified node.

Add fixed support

Assigns a fixed support to the specified node.



Add horizontal fixed roller support

Assigns a horizontal fixed roller to the specified node.

Add vertical fixed roller support

Assigns a vertical fixed roller to the specified node.

x spring

Assigns a horizontal spring to the specified node. The spring stiffness is provided in the corresponding form.

y spring

Assigns a vertical spring to the specified node. The spring stiffness is provided in the corresponding form.

Rot. spring

Assigns a rotational spring to the specified node. The spring stiffness is provided in the corresponding form.

Clear supports

Clear the assigned supports of the specified node.

Apply vertical nodal load

Assigns a vertical load to the specified node for the selected load case.

Apply horizontal nodal load

Assigns a horizontal load to the specified node for the selected load case.

Apply nodal moment

Assigns a moment to the specified node for the selected load case.

Apply vertical uniform frame load

Assigns a vertical uniform load to the specified frame element for the selected load case.

Apply horizontal uniform frame load

Assigns a horizontal uniform load to the specified frame element for the selected load case.



Apply custom load on frame

Assigns a user-specified frame load for the selected load case.

Clear nodal or frame loads

Clears the loads of the specified node or frame element.

Active material

Specifies the material that will be assigned to new frame elements.

Active section

Specifies the cross section that will be assigned to new frame elements.

Active load case

Specifies the active load case.



Model page vertical toolbar

Delete 

Deletes all selected objects.

Linear Horizontal Linear Dimension Line 

Using this command, the user can draw a straight horizontal dimension line.

Linear Vertical Linear Dimension Line 

Using this command, the user can draw a straight vertical dimension line.

Linear Dimension Line 

Using this command, the user can draw a straight dimension line.

Radial Dimension Line 

Using this command, the user can draw a radial dimension line.

Change Dimension Line Location 

Changes the offset of a dimension line (straight or radial). The user shall first click on this command and afterwards click on a dimension line in order to modify its offset.

Label 

Using this command, the user can insert a description label.

Modify label Text 

Changes the text of a label. The user shall first click on this command and afterwards click on a label in order to modify its text.

Modify Location of text label 

Changes the location of the text of a label. The user shall first click on this command and afterwards click on a label in order to modify its text location.

Modify Arrow location of label 

Changes the location of the arrow of a label. The user shall first click on this command and afterwards click on a label in order to modify its arrow location.



Results page toolbar

Reactions

Displays the reactions for the specified load case or load combination.

Deformed Shape

Displays the deformed shape for the specified load case or load combination.

Axial Force Diagram

Displays the axial force diagram for the specified load case or load combination.

Shear Force Diagram

Displays the shear force diagram for the specified load case or load combination.

Bending Moment Diagram

Displays the axial bending moment diagram for the specified load case or load combination.

Dynamic Modes

Displays the dynamic modes of the structure. This option is active when at least one response spectrum or time history load case is available. This function is available in the Dynamic Edition only.

Load case/combo

Specifies the current load case or load combination for results reporting.

Show values

Specifies if force/moment values are displayed for the axial, shear force and bending moment diagrams.

Show min/max values

Specifies if Minimum, Maximum or Minimum and Maximum values are displayed. This option is available when displaying results for a load combination of type "Envelope". Moreover, this option is available if displaying results of a load combination that includes a response spectrum case.

Scale

The diagrams scaling can be controlled by this control.

Step



Specifies the current step to display results for. It is only available if reading results from a non-linear static load case.

Mode

Specifies the dynamic mode to display. This option is active when at least one response spectrum or time history load case is available. This function is available in the Dynamic Edition only.



DESCRIPTION OF THE FORMS

Units

How to open

1. By selecting Settings -> Set Units options from the main window menu bar.
2. By clicking on the “Units” status bar button in the lower right corner of the main screen and then selecting “Details”

Description

The system of units for all input and output data can be defined using this form. The user can choose between the predefined Metric and US (Imperial) systems or even provide his own units for each item, such as force, length, area etc.

Preferences

How to open

By selecting Settings -> Preferences from the main window menu bar.

Description

Misc. tab

The user can specify the positive diagram location (above or below frame element). It defines how internal force diagrams appear.

Moreover, the user can specify if the program automatically checks for updates when it starts.

“Reset to default values” button applies all default settings for the data listed in the “View options” form.

Fonts and sizes

In this tab, the user can define the displayed font families and corresponding font sizes.

Colors tab

In this tab, the user can modify the colors of the drawing area (background, grid, ruler, axes etc.), the colors of the geometric objects (polygons, points, reinforcement bars etc.), color preferences for dimensions/labels as well as for the information pane on the left of the main screen. Moreover, the user is able to see/modify the colors of the displayed resulting screens.




Reset to factory default settings

This option restores all user-defined preferences (colors, fonts, view options etc.) to their default values.

Project information

How to open

1. By selecting Project Data -> Project information from the main window menu bar
2. By clicking on the corresponding toolbar button  of the main window


Description

Project, company as well as model information can be provided using this form. All relative data will be included in the calculation report.



Material definition

How to open

1. By selecting Project Data -> Add new Material from the main window menu bar
2. By clicking on the corresponding toolbar button  of the main window

Description

The available materials are defined in this form.

Name

The name for the material.

Elasticity modulus

Specifies the elasticity modulus of the material.

Shear modulus

Specifies the shear modulus of the material.

Specific weight

Specifies the specific weight of the material. It is used to calculate the self-weight load as well as the self-mass in case of dynamic analysis.

Temperature coefficient of resistance

Specifies the temperature coefficient of resistance of the material.

Import from library

Open the library form of materials in order to import a predefined material.



Material from library

How to open

By clicking the “Import from library” button from the Material definition form.

Description

The user can import a predefined material from a wide variety of predefined material libraries.

Material type

The library to choose a material from. Available lists: Australian NZ, ASTM, CSA, BS, European04, European, UNI, GOST, JIS, JIS (Civil), GB, GB (Civil), JTG04, TB05, KS01, KS, IS, CNS560, CNS.


Available material

The available materials list. A material can be selected and imported to the project by clicking the “Insert material” button.



Cross section definition

How to open

1. By selecting Project Data -> Add new Cross section from the main window menu bar
2. By clicking on the corresponding toolbar button  of the main window

Description

The available cross sections are defined in this form.

Section name

The name for the cross section.

Section type

Using this control, the user can specify the type of the cross section (Rectangular, Circular, I wide flange, Channel, Tee, Angle, Box and Pipe).

Import from library

A predefined cross section can be inserted using this button.

Shear coefficient

The Timoshenko shear coefficient can be specified using this control. The shear coefficient is considered if the option "Consider shear deformation effect" on the Calculation parameters form is enabled. Shear coefficient depends on the section shape and is normally assumed equal to $5/6$ for rectangular sections.

Section dimensions

The sectional dimensions are specified in the corresponding fields.

Section properties

The automatically calculated sectional properties (Area, inertia moments and section moduli) are represented in these fields.

Preview

The chosen section is displayed in this area.



Section from library

How to open

By clicking the “Import from library” button from the Cross section definition form.

Description

The user can import a predefined cross section from a wide variety of predefined section libraries.

Section type

The name for the cross section.

Section type

Using this control, the user can specify the type of the imported cross section (I wide flange, Channel, Tee, Angle, Box and Pipe).

Library list

The library to choose a section from. Available lists: Aluminum shapes, Aluminum shapes 6061-T6, AISC shapes v3, AISC shapes v13, AISC13, AISCLRFD1, AISCLRFD2, AISCLRFD3, Australian and New Zealand steel shapes v8, British Standards steel shapes, British Standards steel shapes 2006, Chinese steel shapes, European steel shapes, Indian steel shapes.

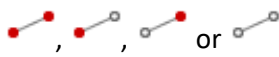

Section

The cross section to import.



Frame element

How to open

1. By selecting Elements -> Add new Frame Element manually from the main window menu bar. A new frame element is created using this option.
2. By selecting Elements -> Properties of selected frame or node from the main window menu bar. An existing frame element can be viewed/modified using these buttons.
3. By clicking one of the corresponding toolbar buttons  or  of the Model page horizontal toolbar. An existing frame element can be viewed/modified using these buttons.
4. By right-clicking on a frame element when on Model page.

Description

The user can create or view/edit a frame element using this form.

Geometry tab

The starting and ending nodes coordinates are displayed.

Properties tab

Material

Specifies the material assigned to the frame element

Cross section

Specifies the cross section assigned to the frame element

Section placement

Specifies how the cross section of the frame element is inserted. If choosing the “Strong axis” option, the cross section is inserted so that the inertia data about the major axis of the section are considered. The strong axis of the cross section is perpendicular to screen.

Similarly, if choosing the “Weak axis” option, the cross section is inserted so that the inertia data about the minor axis of the section are considered. The weak axis of the cross section is perpendicular to screen.

End length offset

Specifies the length of starting and ending offset lengths for the frame element. The stiffness of end length offsets is controlled by the “Rigid end length offset stiffness multiplier” option in the Calculation parameters form.



Consider connection with joints and other frame elements along element length

Specifies if joints within the frame span or intersection with other frame elements are considered as connections. When this option is enabled, an internal node during the analysis is added at these points so that a rigid connection takes place.

Releases tab

Specifies the end releases of the degrees of freedom at the starting and ending node of the frame element. Both full and partial spring releases are supported. In case of partial springs, the corresponding translational/rotational stiffness values can be entered.

Loads tab

Load case

Specifies the load case to display loads for.

Span point loads

Using this control, concentrated span loads (horizontal, vertical forces and/or moments) can be applied.

Coordinate system

The coordinate system of current span loads is specified. Span loads can be provided in local or global coordinate system.

Horizontal load

Horizontal load value

Vertical load

Vertical load value

Moment

Moment value

Distance from starting node

Specifies the location of the span loads. If option "Absolute" for location definition is selected, the distance is provided in absolute units, for example m, ft, etc. If option "Relative" for location definition is selected, the distance is provided relatively to the frame length, for example 0.50 means at the middle of the frame element.

Temperature load

Thermal difference



Specifies the temperature difference between environment and central frame line, in absolute temperature units. This load mainly causes axial tension/compression.

Thermal gradient

Thermal gradient is defined as the ratio of the temperature difference between two points, each located above and below the frame element. Thermal gradient has the units of temperature divided by length. This load mainly causes bending.

Uniform load

Coordinate system

The coordinate system of uniform loads is specified. Uniform loads can be provided in local, global or projected global coordinate system. If choosing global projected coordinate system, loads are applied at global coordinate system and they are afterwards projected to the frame element. Snow loads on slopped roofs are usually entered at projected global system.

Load

The uniform load value in units force per length.

Direction

Specifies the load direction (horizontal or vertical).

Trapezoidal load

Coordinate system

The coordinate system of uniform loads is specified. Uniform loads can be provided in local, global or projected global coordinate system. If choosing global projected coordinate system, loads are applied at global coordinate system and they are afterwards projected to the frame element. Snow loads on slopped roofs are usually entered at projected global system.

Distance 1 from starting node

Specifies the starting location of the trapezoidal load. If option “Absolute” for location definition is selected, the distance is provided in absolute units, for example m, ft, etc. If option “Relative” for location definition is selected, the distance is provided relatively to the frame length, for example 0.50 means at the middle of the frame element.

Load 1

The uniform load value at the starting location of the trapezoidal load in units force per length.

Distance 2 from starting node



Specifies the ending location of the trapezoidal load. If option “Absolute” for location definition is selected, the distance is provided in absolute units, for example m, ft, etc. If option “Relative” for location definition is selected, the distance is provided relatively to the frame length, for example 0.50 means at the middle of the frame element.

Load 2

The uniform load value at the ending location of the trapezoidal load in units force per length.

Direction

Specifies the load direction (horizontal or vertical).



Node

How to open

1. By selecting Elements -> Properties of selected frame or node from the main window menu bar. An existing frame element can be viewed/modified using these buttons.
2. By right-clicking on a node when on Model page.

Description

The user can view/edit a node using this form.

Geometry tab

Coordinates

The nodal coordinates are displayed.

Additional Masses

The additional nodal masses can be specified using this control. This option is available in the Dynamic Edition only.

Supports tab

Specifies the support types of the node. All combination of degrees of freedom restraints are supported. In case of spring supports, the corresponding translational/rotational spring stiffness values can be provided.

Loads tab

Load case

Specifies the load case to display loads for.


Nodal load

Using this control, nodal loads (Horizontal, Vertical forces and Bending moments at global coordinate system can be applied.



Static load case

How to open

1. By clicking the “Add new” button from the Load cases field of the Load cases/combo tab page on the left of the main page. Using this option, a static load case can be added.
2. By clicking the “Details” button from the Load cases field of the Load cases/combo tab page on the left of the main page. Using this option, a static load case can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Load Case menu item.
4. By clicking on the corresponding toolbar button  of the main window

Description

A linear static analysis case is represented in this form.

Name

The name of the load case.

Include self weight

Specifies if self weight is considered for this load case. If this option is enabled, self weight is automatically added to the remaining loads specified by the user.


Self weight multiplier

Specifies the self weight factor to be considered. This option is available if “Include self weight” option is selected.



Non linear static load case

How to open

1. By clicking the “Add new” button from the Load cases field of the Load cases/combos tab page on the left of the main page. Using this option, a non linear static load case can be added.
2. By clicking the “Details” button from the Load cases field of the Load cases/combos tab page on the left of the main page. Using this option, a non linear static load case can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Load Case menu item.
4. By clicking on the corresponding toolbar button  of the main window

Description

A non linear static analysis case is represented in this form.


Data

The analysis cases, that are considered in the non linear load case, are specified here. A factor can also be specified for each participating load. Only linear static load cases can take part in a non-linear static case.



Response spectrum case

How to open

1. By clicking the “Add new” button from the Load cases field of the Load cases/combos tab page on the left of the main page. Using this option, a response spectrum load case can be added.
2. By clicking the “Details” button from the Load cases field of the Load cases/combos tab page on the left of the main page. Using this option, a response spectrum load case can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Load Case menu item.
4. By clicking on the corresponding toolbar button  of the main window

Description

A response spectrum analysis case is represented in this form.

Direction

Specifies if the response spectrum case will be applied at the X (horizontal) or Y (vertical) global coordinate.

Modal combination method

Specifies how the calculated modes will be combined in order to get the dynamic response of the structure. Available methods: CQC (Complete-Quadratic-Combination) and SRSS (Square Root of the Sum of the Squares).

Response Spectrum Function

Specifies the response spectrum function to use for this load case.



Standard response spectrum function

How to open

1. By clicking the “Add new” button from the Response Spectra field of the Dynamic Data tab page on the left of the main page. Using this option, a response spectrum can be added.
2. By clicking the “Details” button from the Load cases/combo tab page on the left of the main page. Using this option, a response spectrum can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Response Spectrum function menu item.

Name

The name of the response spectrum function.

Starting acceleration

Specifies the acceleration at the start of the function.

Period at point 2

Specifies the period at the second function point.

Acceleration at plateau

Specifies the acceleration at the top of the function

Period at point 3

Specifies the period at the third function point.

Ending period

Specifies the period at the end of the function.

Ending acceleration

Specifies the acceleration at the end of the function.

Response spectrum function

Displays the response spectrum data.

Response spectrum graph

Displays the response spectrum function plot.



User defined response spectrum function

How to open

1. By clicking the “Add new” button from the Response Spectra field of the Dynamic Data tab page on the left of the main page. Using this option, a response spectrum can be added.
2. By clicking the “Details” button from the Load cases/combo tab page on the left of the main page. Using this option, a response spectrum can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Response Spectrum function menu item.

Name

The name of the response spectrum function.

Period

The period of a point of the response spectrum function to add.

Acceleration

The acceleration of a point of the response spectrum function to add.

Add/Edit/Delete

The points of the diagram can be added, modified or deleted using these buttons.



Response spectrum function from file

How to open

1. By clicking the “Add new” button from the Response Spectra field of the Dynamic Data tab page on the left of the main page. Using this option, a response spectrum can be added.
2. By clicking the “Details” button from the Load cases/combo tab page on the left of the main page. Using this option, a response spectrum can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Response Spectrum function menu item.

Name

The name of the response spectrum function.

Select file

Select the file that includes the response spectrum function.

Response spectrum function

Displays the response spectrum data.


Response spectrum graph

Displays the response spectrum function plot.



Time history case

How to open

1. By clicking the “Add new” button from the Load cases field of the Load cases/combos tab page on the left of the main page. Using this option, a time history load case can be added.
2. By clicking the “Details” button from the Load cases field of the Load cases/combos tab page on the left of the main page. Using this option, a time history load case can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Load Case menu item.
4. By clicking on the corresponding toolbar button  of the main window

Description

A time history analysis case is represented in this form.

Direction

Specifies if the time history case will be applied at the X (horizontal) or Y (vertical) global coordinate.

Modal combination method

Specifies how the calculated modes will be combined in order to get the dynamic response of the structure. Available methods: CQC (Complete-Quadratic-Combination) and SRSS (Square Root of the Sum of the Squares).

Time History Function

Specifies the time history function to use for this load case.

Steps

Specifies the number of steps to consider.

dt

Specifies the time interval for each step.



Sine time history function

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Amplitude

Specifies the amplitude of the time history function.

Period

Specifies the period of the time history function.

Number of cycles

Specifies the number of cycles of the time history function.

Steps per cycle

Specifies the steps per cycle of the time history function.

Time history function

Displays the time history data.

Time history graph

Displays the time history function plot.



Cosine time history function

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Amplitude

Specifies the amplitude of the time history function.

Period

Specifies the period of the time history function.

Number of cycles

Specifies the number of cycles of the time history function.

Steps per cycle

Specifies the steps per cycle of the time history function.

Time history function

Displays the time history data.

Time history graph

Displays the time history function plot.



Ramp time history function

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Ramp time

Specifies the time at the ramp.

Amplitude

Specifies the amplitude of the time history function.

Maximum time

Specifies the maximum time of the time history function.

Time history function

Displays the time history data.

Time history graph

Displays the time history function plot.



Saw tooth time history function

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Ramp time

Specifies the time at the ramp.

Period

Specifies the period of the time history function.

Number of cycles

Specifies the number of cycles of the time history function.

Time history function

Displays the time history data.

Time history graph

Displays the time history function plot.



Triangular time history function

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Amplitude

Specifies the amplitude of the time history function.

Period

Specifies the period of the time history function.

Number of cycles

Specifies the number of cycles of the time history function.

Time history function

Displays the time history data.

Time history graph

Displays the time history function plot.



User defined time history function

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Time

The time of a point of the time history function to add.

Acceleration

The acceleration of a point of the time history function to add.

Add/Edit/Delete

The points of the diagram can be added, modified or deleted using these buttons.

Time history function

Displays the time history data.

Time history graph

Displays the time history function plot.



Time history function from file

How to open

1. By clicking the “Add new” button from the Time History functions of the Dynamic Data tab page on the left of the main page. Using this option, a time history function can be added.
2. By clicking the “Details” button from the Time History functions of the Dynamic Data tab page. Using this option, a time history function can be viewed/modified.
3. By choosing the corresponding item from the Project Data->Add new Time History function menu item.

Name

The name of the time history function.

Select file

Select the file that includes the time history function.

Time history function

Displays the time history data.


Time history graph

Displays the time history function plot.



Load combination

How to open

1. By clicking the “Add new” button from the Load combinations field of the Load cases/combo tab page on the left of the main page. Using this option, a load combination can be added.
2. By clicking the “Details” button from the Load combinations field of the Load cases/combo tab page on the left of the main page. Using this option, a load combination can be viewed/modified.
3. By clicking the Project Data->Add new Load Combination function menu item.
4. By clicking on the corresponding toolbar button  of the main window

Name

The name of the load combination.

Type

The type of combination between load cases and load combinations that participate in this load combination. Supported types are:

Add: The program performs a linear addition of the response of each participating load case or load combination in this load combination.

Absolute: Performs a linear addition of the absolute values of the response of each participating load case or load combination in this load combination, in order to get the maximum values of the response. Minimum values are obtained by multiplying maximum values by -1.

Envelope: A max/min Envelope of the included load cases or load combinations is evaluated for each frame output segment and node. The participating load cases or load combinations that give the maximum and minimum components are used for this combo. Therefore, the load Combo holds two values (minimum and maximum) for each output segment and node.

SRSS: To get the maximum values, the program considers the square root of the sum of the squares of the larger absolute values for each of the included load cases or load combinations. Minimum values are obtained by multiplying maximum values by -1.


Analysis cases

The participating load cases or load combinations and their factors are reported on the corresponding control.



Calculation parameters

How to open

1. By selecting Analysis -> Calculation Parameters from the main window menu bar.
2. By clicking on the corresponding toolbar button  of the main window.

Description

Using this form, the user can define the parameters that are considered during the structural analysis of the model.

General tab

Consider shear deformation effect

Using this control, the user can specify if shear deformation effects are to be considered in the analysis. When this option is enabled, Timoshenko beam theory is used. Timoshenko beam theory is recommended for structures with small ratios of length per beam height.

Rigid end length offset stiffness multiplier

Specifies that stiffness multiplier of the end length offset length (if any) at starting and ending frame element locations. This value is unitless and is applied to the initial frame element stiffness.

Non Linear Analysis tab

The Modified Newton-Raphson algorithm is used in order to carry out the static non-linear analysis. The related parameters are represented below.

Load steps

Specifies the number of load steps for the non-linear loads.

Iterations per load step

Specifies the number of iterations within a load step for the non-linear loads.

Convergence tolerance

Specifies the stiffness tolerance that is considered for convergence.

Update method

Specifies how the stiffness matrix is updated during the non-linear analysis. Available options are: After each iteration inside a load step, After each load step and Keep constant.

Dynamic Analysis tab

The parameters of dynamic response spectrum, time history and modal analysis are specified in this control. This option is only available in the Dynamic Edition.



Mass source

Specifies the mass source for the dynamic analysis.

From self weight only: Dynamic mass is assumed to come from Self weight only.

From self weight plus specified loads: Dynamic mass is assumed to come from Self weight plus specified load cases. The corresponding load cases control is enabled if this option is selected.

From specified loads only: Dynamic mass is assumed to come from specified load cases only. The corresponding load cases control is enabled if this option is selected.

Number of modes to find

Specifies the number of dynamic modes to calculate. This number cannot be greater than the total degrees of freedom of the structure.

Acceleration of gravity

Specifies the gravity acceleration to consider in order to convert loads to masses.

Lumped mass matrix

If selected, the specified mass matrix is lumped (only diagonal terms). Otherwise, a consistent mass matrix is produced. It is strongly recommended to use a lumped mass



Frame results

How to open

By right clicking on a frame element when in Results page

Description

The results of a single frame element are reported in this form.

Frame element

The label of the frame element is displayed.

Length

The length of the frame element is displayed.

Location

The location to get results at. Location is provided in length units. Internal forces and displacements are reported for the specified location. The given location cannot lie within any end offset lengths, if defined.

Load case/combo

The load case or load combination to get results for.

Results type

Specifies if Minimum, Maximum or Minimum and Maximum values are displayed. This option is available when displaying results for a load combination of type "Envelope". Moreover, this option is available if displaying results of a load combination that includes a response spectrum case.

Step

Specifies the current step to display results for. It is only available if reading results from a non-linear static load case.

Displacements/Rotations

The translational displacements and rotations at the local coordinate system of the frame element are represented.

Internal forces

The internal forces at the local coordinate system of the frame element are represented.

Internal forces representation

The minimum and maximum values and corresponding locations are reported. These locations are also represented graphically by clicking on each of them.



Node results

How to open

By right clicking on a node when in Results page

Description

The results of a single node are reported in this form.

Node

The label of the node is displayed.

x, y

The nodal coordinates are displayed.

Load case/combo

The load case or load combination to get results for.

Results type

Specifies if Minimum, Maximum or Minimum and Maximum values are displayed. This option is available when displaying results for a load combination of type “Envelope”. Moreover, this option is available if displaying results of a load combination that includes a response spectrum case.

Step

Specifies the current step to display results for. It is only available if reading results from a non-linear static load case.

Displacements/Rotations

The translational displacements and rotations at the global coordinate system are represented.


Reactions

The reactions of the constrained degrees of freedom (if any), at the global coordinate system are represented.



Results locations

How to open

1. By selecting Analysis -> Specify Results Locations from the main window menu bar.
2. By clicking on the corresponding toolbar button  of the main window

Description

The locations of the frame elements that will be included in the calculation report are defined using this form. Warning: Results locations are cleared and replaced by default values each time the structural analysis is carried out. They shall be set by the user just before the creation of the calculation report.

Element

The element to view or modify its results locations.

New location to add

Using this control, a new location for results reporting can be added.

Delete selected location

Using this button, the selected results location is deleted.

Reset for element

Using this button, the selected results location for the selected frame element are cleared and reverted to their defaults.


Reset for all elements

Using this button, the selected results location for all frame elements are cleared and reverted to their defaults.



Results tables

How to open

1. By selecting Analysis -> Results tables from the main window menu bar.
2. By clicking on the corresponding toolbar button  of the main window

Description

The analysis results of the whole structure are represented in this form in tabular format. The values can be exported to Excel for further processing.

Results

Specifies the results to be displayed. Available options are:

Support reactions: Displays the nodal reactions

Total reactions: Displays the total reaction of the whole structure

Node displacements/rotations: Displays the nodal displacements/rotations

Frame forces: Displays the frame elements internal forces

Frame deformation: Displays the frame elements deformations

Modal analysis results: Displays the calculated dynamic modes and the related modal analysis data. This option is available in the Dynamic Edition only and is active if at least one response spectrum or time history analysis has been included in the structure.

Load case/combination

Specifies the load case or load combination to display results for.

Output type

Specifies if Minimum, Maximum or Minimum and Maximum values are displayed. This option is available when displaying results for a load combination of type "Envelope". Moreover, this option is available if displaying results of a load combination that includes a response spectrum case.


Step

Specifies the current step to display results for. It is only available if reading results from a non-linear static load case.



Report settings

How to open

1. By selecting Analysis -> Calculation report from the main window menu bar.
2. By clicking on the corresponding toolbar button  of the main window.

Description

In this form, the user can define the data which will be represented in the calculation report.

Data to include in calculation report

All relative project items can be selected to be included in the report.

The report will only contain items that have already be used in the project. Thus, materials and sections that have not been assigned to frame elements will not be included.

Select load cases/combos

The user can select which of the load cases/combinations will be represented on the calculation report.