User’s Manual - Analysis3D v 3.01
Cuylaerts Engineering

Index

INDEX 2

INTRODUCTION 4

1. GENERAL OVERVIEW - GETTING STARTED 5

2. THE FILE MENU 7

3. THE STRUCTURE MENU 8

3.2. TRUSS OR FRAME 8

3.3. JOINTS 8

3.4. MEMBERS 9

3.5. COORDINATE SYSTEMS 11

3.6. SUPPORT JOINTS 13

3.6.1. ENTERING SUPPORT JOINTS FROM THE STRUCTURE MENU 14

3.6.2. ENTERING SUPPORT JOINTS INTERACTIVE 15

3.7. CROSS SECTIONAL DATA 15

3.8. MATERIAL DATA 18

3.9. UNITS AND REGION 18

4. THE LOADS MENU 20

4.1. JOINT LOADS 20

4.2. MEMBER LOAD 20

4.3. WIND LOAD 21

4.4. SNOW LOAD 29

4.5. SEISMIC LOADS 35

4.6. LOAD COMBINATIONS 47

4.7. CALCULATE OR NEGLECT THE SELF WEIGHT 48

4.8. TEMPERATURE 48

5. THE DRAWING MENU 49

5.1. DRAWING SHORTCUT KEYS 49

6. CALCULATE 51
7. RESULTS 53
7.1. JOINT DISPLACEMENTS 53
7.2. MEMBER FORCES 53
7.3. MEMBER STRESSES 53
7.4. SWAY AND NATURAL FREQUENCY 54
7.5. BUCKLING 55
7.6. REACTIONS 56
7.7. DETAILED MEMBER FORCES 56
7.8. DETAILED MEMBER STRESSES 57
7.9. MEMBER DESIGN 59
7.10. DETAILED MEMBER DESIGN 59
7.11. AISC-ASD 60
7.12. AISC-LRFD 61
7.13. BS 5950 76
7.14. CISC 94 76
7.15. EUROCODE 3 76
8. THE PRINT MENU 84
9. END-USER LICENSE AGREEMENT ("AGREEMENT") 85

Introduction

Welcome to the group of users of *Analysis3D*.
With *Analysis3D* you have purchased one of the most user-friendly analysis tools for steel constructions you can get.
We hope this manual will guide you in a clear way through the fascinating world of Finite Element Analysis and Structural Analysis.

Ever since 1992 Cuylaerts Engineering has been producing structural engineering software with special focus on a user-friendly interface. Due to this *Analysis3D* is being used in over 60 countries worldwide.
To assist you, there is already a large library of different sections included (Australia and New Zealand, China GB, Euro, India BIS, Japan JIS, Russia GOST, UK British Steel, US AISC, and Timber).

The software performs complex 3D analysis, such as analysis for flexure, torsion, axial, shear, and deflection. Users can input their desired loads and support conditions, choose or define any structural sections, produce 3D visualization and analysis text reports.
Analysis3D also includes **Structural Steel Design** that proposes the best section for each member. Each cross-section is tested for Plastic Tension or Compression, Bending moment (major and minor axis), Plastic Shear resistance, Reduced plastic shear resistance for the case of combined Torsion and Shear, Reduced moment resistance for the case of combined Bending and Shear, Reduced plastic moment resistance for the case of combined Bending and Axial force or bi-axial bending, Compression buckling,
Lateral Torsional buckling, Combined bending and axial compression (Chen and Atsuta) and Maximum deflection.

Nonlinear or 2nd order Calculation takes into account the deformed structure and incrementally calculates the total effect on the structure.

Analysis3D has become what it is now, thanks to the comments and remarks of our users. We believe we can always improve *Analysis3D*, so we will be grateful to receive your remarks and observations.

https://www.cuylaerts.net/
Please send your questions and remarks to frank@cuylaerts.net

1. General Overview - Getting started

Analysis3D calculates two- and three-dimensional Trusses and Frames. These structures consist of **long slender Members** that are supposed to behave **linear elastic**.

Below you can find a step-by-step approach to get started:
A: start from the Structure menu

1) Select the units and your region to determine which library of cross sections to use
2) Select the dimensions and type of structure: 2D or 3D, Truss (with hinges on the joints) or Frame
3) Add your joints and their coordinates
4) Connect the joints with members (beams) and specify the material and cross section of each beam
5) Specify all the support joints to give your structure the required stability

B: using the Loads menu

1) Specify the joint loads and/or member loads acting on the structure
2) Specify if the own weight of the structure needs to be calculated or not
3) Add your wind Loads

C: Calculate the structure and check the Results

Before entering a construction, be sure to determine whether the structure is a Frame or a Truss and whether the structure has 2 or 3 dimensions. You can also select your preferred Units and Region for the Cross Sections Library.

First enter the Joint coordinates using the Global Coordinate system.
Members can only be entered when the Joints are defined.

Joints, Members and all other structure data can be entered using the Menu Structure or interactively, by means of the mouse.

Entering the Data interactively has the following advantage:
1. The result of each minor change is visualized instantly and can be checked.
2. Selecting a Joint or a Member with the mouse is much handier than a search through the corresponding lists.
3. Support Joints and Joint Loads can be entered together with the corresponding Joint.
4. Member Loads can be entered together with the corresponding Member.

When all Structure data (Joints, Members, Support Joints,…) are entered, you can define the loads on the structure (Joint Loads, Member Loads, Wind Load,..)

A number of drawing buttons help you simplify the drawing control.

When the construction is fully described, you can Calculate the structure and view the Results.

The Joint Displacements are shown automatically. Members that fail the requirements for Member Stress or Buckling are indicated in red. The course of Axial Forces, Shear Forces, Torsion, Bending Moments and Deflection can be viewed with the Result Menu option Detail Member Forces.
All compressed members are tested for Buckling.

By using Member Design, you can improve and optimize the structure. *Analysis3D* will check the strength of each member and propose the minimal Cross Section for the loads applied.

Analysis for Windows calculates two- and three-dimensional Trusses and Frames. These structures consist of **long slender Members** that are supposed to behave **linear elastic**.

Before entering a construction, be sure to determine whether the structure is a Frame or a Truss and whether the structure has 2 or 3 dimensions. You can enter your data in International or US Units.

First enter the Joint coordinates. Members can only be entered when the Joints are given.

Joints, Members and all other structure data can be entered with the Menu Structure or interactively, by means of the mouse.

Entering the Data interactively has the following advantage:
1. The result of each minor change is visualized instantly and can be checked.
2. Selecting a Joint or a Member with the mouse is much handier than a search through the corresponding lists.
3. Support Joints and Joint Loads can be entered together with the corresponding Joint.
4. Member Loads can be entered together with the corresponding Member.

A number of drawing shortcuts simplify the Graphics control.

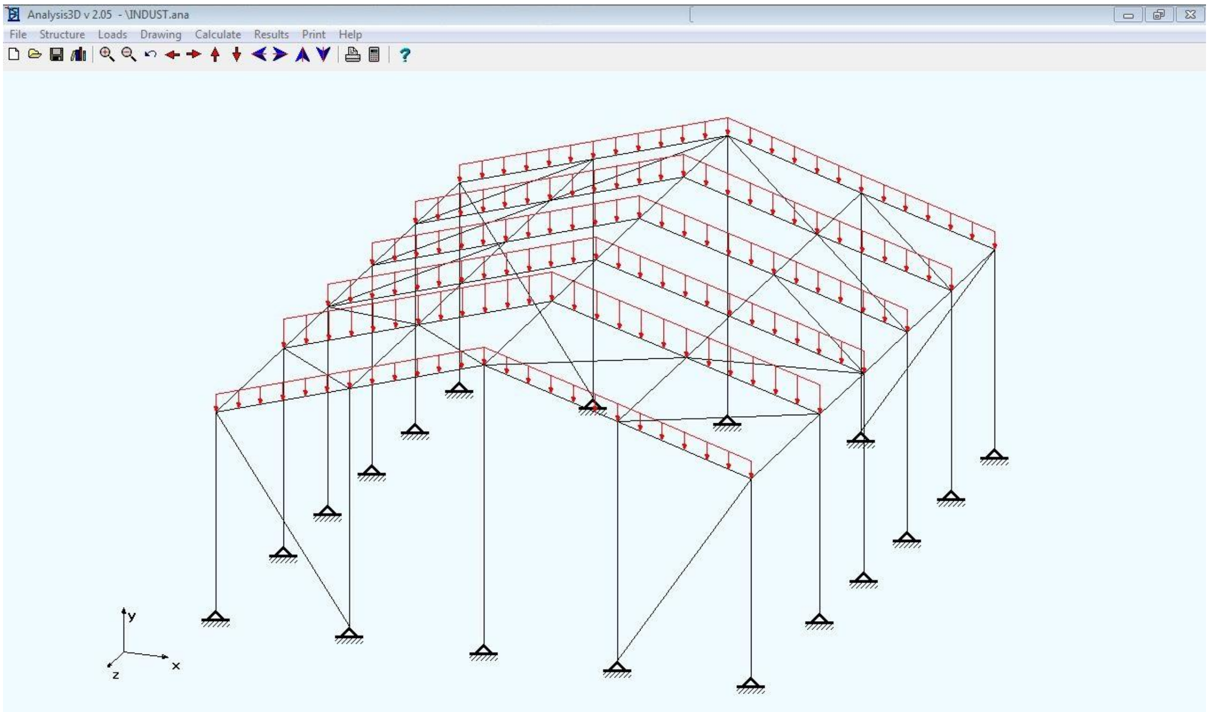
After defining Joints and Members, you can add Support Joints, Joint Loads and Member Loads.

When the construction is fully described, you can calculate the structure and view the Results.

The Joint Displacements are shown automatically. Members that fail the requirements for Member Stress or Buckling are indicated in red. The course of Axial Forces, Shear Forces, Torsion, Bending Moments and Deflection can be viewed with the Result Menu option Detailed Member Forces.

All compressed members are tested for Buckling.

By using **Member Design**, you can improve and optimize the structure. *Analysis3D* will check the strength of each member and propose the minimal Cross Section for the loads applied.



2. The File Menu

The Files containing the structure data are saved by default with the extension 'ANA', but other extensions are possible.

New erases all structure data in order to enter a new construction from scratch.

Open loads an existing construction from disk into memory. The old structure data in memory will be overwritten and lost.

Save writes the structure data to disk with the given filename.

Combine loads the structure data from a given file into memory. The loaded data will be combined with the old structure in memory.

DXF-Files can be read and saved by this option. Thus, drawings from AutoCad or other drawing tools can be processed and edited. Reading and writing of DXF-files is restricted to straight lines. The name of the layer in the drawing will be used as the descriptor for the section used for each member.

Text File in/out retrieves or saves the structure in a text file with extension 'TXT'. When the structure has been calculated, the results will be added in the text file. In this way the results can be used for further processing or importing in other software like Excel. The layout is identical as the printer layout.

Library contains all Cross Section and Material data. The Library also contains your preferred Language, the color settings for drawing, your preferred Units used, the Region preference for the Cross Sections and safety factors for Buckling. At startup *Analysis3D* loads by default the Library file Analysis.lbr containing all settings that were last saved.

Exit stops *Analysis3D*. If some data have been changed since the last saving, the user will be prompted for a confirmation, after which the data can still be saved.

3. The Structure menu

3.1. Two or three-dimensional structures

With the Structure Menu option '2 or 3-Dimensional' you can change the dimension of the construction. By selecting the menu option, the structure toggles between 2 and 3 dimensions.

The small coordinate system in the bottom left corner also indicates the actual dimensions selected.

When an existing 2-dimensional structure is converted into 3 dimensions, the resulting construction will be unstable due to a lack of restraints in the Z-direction. An adjustment of the support joints or joint restrictions is required.

3.2. Truss or Frame

With the Structure Menu option Truss or Frame the type of connections between the members is established. By clicking this option, the structure toggles between a Truss and a Frame.

When a structure only consists of members that are exclusively subjected to axial Forces, then the structure is called a Truss. When one or more Members are subjected to shear Forces or bending moments, then the structure should be considered as a Frame.

In case that the construction consists of both types of members, start with a Frame and indicate those members that act as a Truss. Use **Hinges** on both sides of the member for that purpose.

A circle in the origin of the small coordinate system in the bottom left corner indicates that the structure is a Truss. The circle symbolizes the Hinge.

3.3. Joints

A Joint is defined by a Joint Number and the corresponding coordinates given in American or International Units. The units are specified by the Structure Menu option Units. In International Units, Joint coordinates can be entered in meter or mm as indicated by the check box. The Global coordinate system for any structure is indicated by the small coordinate system in the bottom left corner of the screen.

The Joints can be entered either with the menu or interactively with the mouse.

3.3.1. Entering Joints from the Structure menu.

To add, change or remove Joints in the Joint list simply select the Joint option from the Structure menu.

Adding new Joints: Enter the Joint Number and his coordinates to the left of the ADD button. When entering Joints in a 2-dimensional construction, Z-coordinates will not be accepted.

To add the Joint data to the list simply press ENTER in the last edit box, or press the ADD button. After each new added Joint, the Joint Number will automatically increase by one.

Removing Joints: Select the Joint(s) you want to remove and press the REMOVE button. The program requires that all Joint Number follow without a gap. The gap that is caused by the removal of the Joints will automatically be filled when the window is closed. All Joints after the gap will be moved forward.

Changing Joints: This can be done in 3 ways:
1) Type the Joint Number and press ENTER.
2) Double Click on the Joint Number with the mouse.
3) Select the Joint Number and click the Edit button.

In both cases the Joint will be removed from the list to the edit boxes, where you can edit the joint data. Don't forget to add the Joint to the list after editing. Editing the Joints directly in the list is not allowed.

The **COPY** button enables you to copy or move the selected Joints over a given distance in all directions.

3.3.2. Moving and Copying Joints

The **selected Joints** can be moved or copied over a given distance in the X, Y or Z direction with the COPY button from the Joint window.

All the intermediate Members will be copied also. This simplifies entering a large construction with identical substructures.

If the selected Joints only have to be moved instead of copied, select the MOVE button in the Copy Joints window.

3.3.3. Entering Joints interactively

To **add a new Joint**, double click the left mouse button on the screen. A new window will appear to enter the new Joint coordinates.

To **change** an existing Joint, double click with the left mouse button on the Joint you want to change. A new window will appear to allow editing of the selected joint and the selected Joint number will turn red.

To deselect a Joint, press the right mouse button.

From this window you can change the Joint coordinates directly. Use the arrow buttons to change the joint number you want to change. The **+button** enables you to **add** a new Joint. Using - will **delete** the current Joint.

The tabs SUPPORT and LOAD allow you change the Support Joints and Joint Loads of the selected Joint. This method allows you to view each change on the screen as soon as it is entered.

3.4. Members

A Member is defined by a begin Joint and an end Joint. The begin Joint is the Joint with the smallest x-coordinate according the local coordinate system.

It is not necessary to apply this rule, since *Analysis3D* will automatically check begin and end Joint and makes these changes for you.

The names of the Cross Sections need to correspond to the ones defined in the library of Cross Sections.

The Material name must correspond to the Material name in the Material Data list.

The Members of a Frame can have Hinges and can be rotated over a certain Angle around its local x-axis.

Members can be entered either with the menu or interactively with the mouse.

3.4.1. Entering Members from the Structure Menu.

To add, change or remove Members in the Member list simply select the Members option from the Structure menu.

Adding new Members: Enter the Member Number begin and end Joint, the name of the Cross Section and in case of a Frame: the Hinges and the rotation Angle.

To add the Member data to the list simply press ENTER in the last edit box, or press the ADD button. After each new added Member, the Member Number will automatically increase by one.

Removing Members**:** Select the Member(s) and press the REMOVE button. The program requires that all Member Number follow without a gap. The gap that is caused by the removal of Members will automatically be filled when the window is closed. The Member numbers after the gap will be moved forward.

Changing Members: This can be done in two ways:
1) Type the Member Number and press ENTER.
2) Select the Member with the mouse and press the right mouse button.

In both cases the Member will be moved from the list to the edit boxes, where you can edit the Member data. Don't forget to add the Member to the list after editing. Editing the Members directly in the list is not allowed.

The Cross **Section** button lets you change the Cross Section of **all** Members with a given Section. For instance, you can change all Members with Cross Section IPE200 into IPE240.

The **Material** button lets you change the material name of **all** Members of a given material. For instance, you can change all Members with Material FE360 to FE450.

With the **Hinge** button you can change the Hinges of the selected Members.

3.4.2. Entering Members interactively.

To enter a **new Member** select begin and end Joint Number with the left mouse button. A selected Joint Number turns Red. Use the right mouse button to deselect a Joint.

The new Member window will enable you to add the other properties of the Member. *Analysis3D* assumes that the new Member has the same properties as the last entered Member, but this can be overwritten.

The names of defined Cross Sections and Material Data can be selected from the drop-down list.

To **change** an existing Member select begin and end Joint Number with the left mouse button. A window in the top right of the screen will enable you to change the properties of the Member. At the same time the Local Coordinate System of the selected Member is displayed.

Once a Member has been selected, it can be **removed**, together with the Member Loads, by pressing the **-** button. To add a member, press the **+ button**.

To enter the Member Loads of the selected Member press the LOAD tab.

This method lets you view each change or new Member on the screen instantly.

3.4.3. Hinges

A Frame can contain one or more Members that have a Hinge on one or both sides. A Hinge indicates that the member is exclusively subjected to axial Forces on the side of the Hinge.

The following symbols are used to indicate the Hinges:

- 0 : The Member has no Hinges.
- B : The Member has one Hinge at the **Begin** Joint.
- E : The Member has one Hinge at the **End** Joint.
- 2 : The Member has two Hinges at the begin and end Joint.

A Truss by definition consists of members with Hinges on both sides. Hence Hinges cannot be defined in case of a Truss.

3.4.4. The Rotation Angle of a Member

A Member can be rotated around his local x-axis over a certain angle during construction. For example, an I-beam can be used turned over 90° to bear large transverse Forces. An exact representation of the angle of rotation is important since the Moment of Inertia of the Member has to be recalculated. The angle is positive according the right-hand rotation rule along the positive local x-axis.

The resulting Rotation Angle can be represented graphically by selecting the Member with the mouse. The Local Coordinate System of the selected Member will be displayed. These coordinates will help you to determine the positive local x-axis. The local y-axis always coincides with the major Cross Section Moment of Inertia Ix.

Since Members in a Truss are exclusively subjected to tension and compression, it has no influence whether the Members are rotated or not.

3.5. Coordinate Systems

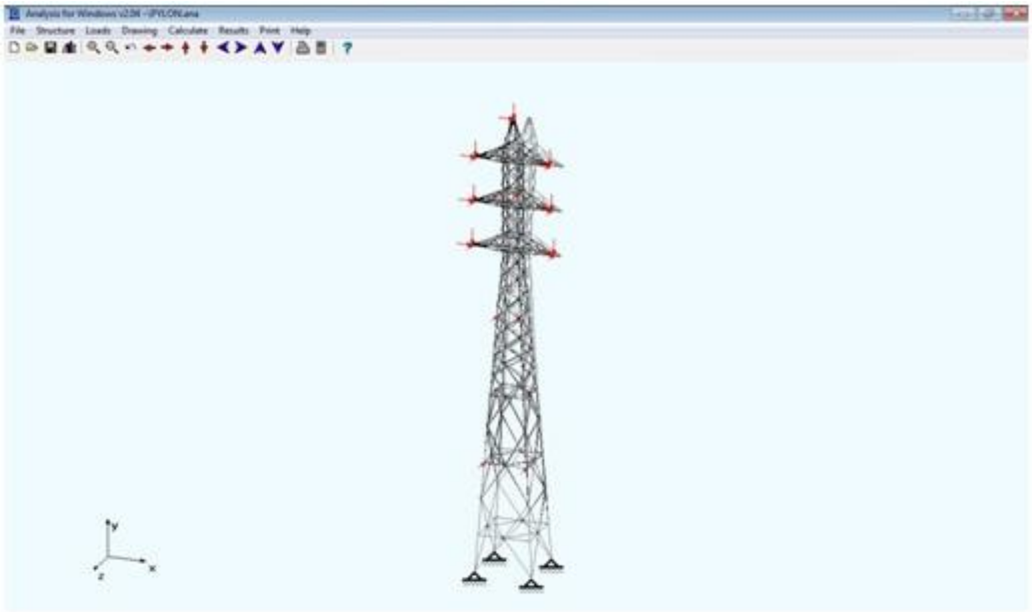
3.5.1. The Global Coordinate System

All Joints, Support Joints, Joint Loads, Joint Displacements and Reactions are referenced to the Global Coordinate System.

The Global Coordinate System consists of a set of right-hand orthogonal axes X, Y and Z which are oriented as shown in the left bottom corner of the screen.

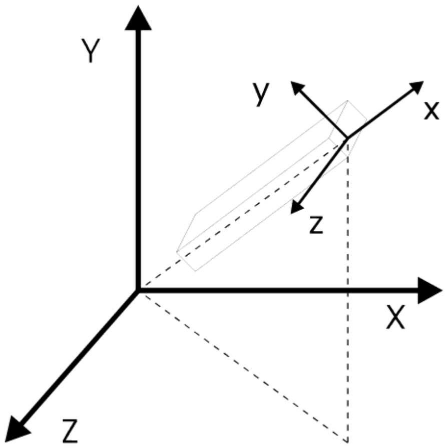
The representation of the Global Coordinate System will follow the point of view. It will also indicate whether the Structure is a Truss or a Frame, and whether the structure has two or three dimensions.

The positive global Y-axis represents the vertical height of a structure. This means that the own-weight forces will work in the negative Y direction.



3.5.2. The Local Coordinate System

IN THEORY



The **local x-axis** runs parallel with the Member through the middle of the Cross Section.

The Joint with the smallest global X coordinate is the begin Joint. This means that the local x-axis always points from the begin Joint to the end Joint of a Member.

In case that the global X coordinates are equal, the Joint with the smallest Y coordinate is the begin Joint. When both X and Y coordinates are equal, the smallest Z coordinate indicates the begin Joint.

The **local z-axis** will always be oriented parallel to the global XZ plane.

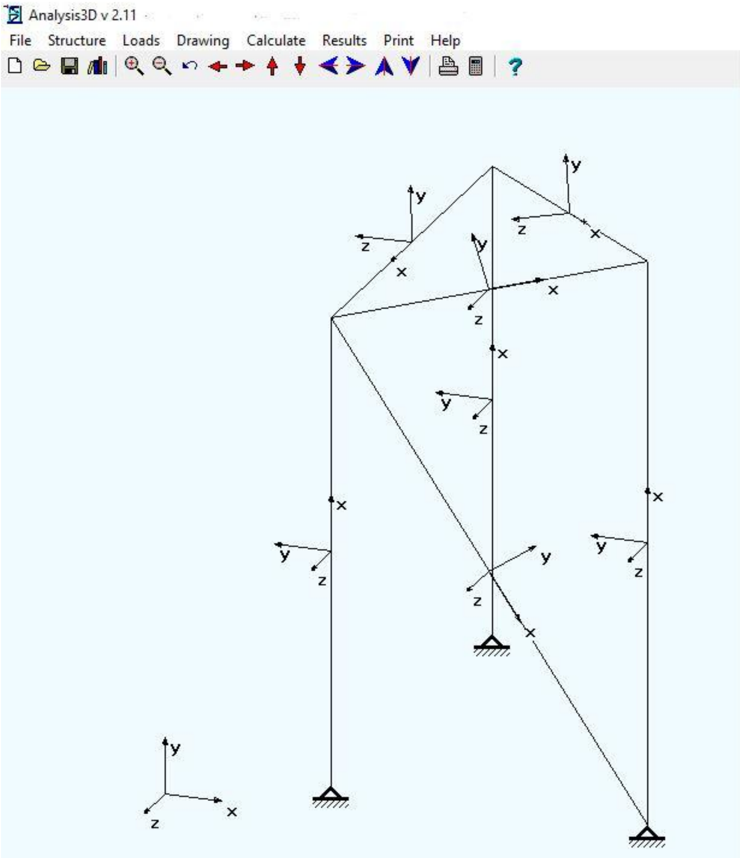
The **local y-axis** will be oriented so that its positive direction projects onto the positive global Y-axis.

In the special case that the local x axis is parallel to the global Y axis (e.g., a vertical Member), the local z-axis will have the additional restriction that it will be parallel to the global Z axis and extend in the positive Z direction.

IN PRACTICE

Put into practice, *Analysis3D* will take care that the Member Data coincides with the local coordinate system.

For your information the local coordinate system of the Member is shown additionally on the screen when a Member is selected with the mouse (by selecting both joint numbers of that member).



3.6. Support Joints

A Support Joint is defined by the Joint Number and the restrictions per axis.

To indicate the Restraints, we use the following agreement:

> When there is **no Restraint** in the given direction, the corresponding direction will be left blank using a **space**.
> **R** : any movement is **Restrained** in the given direction. Displacements will stay zero.
> **D** : the support experiences a forced **Displacement** in the given direction. The magnitude of the displacement it given by Dx, Dy or Dz.
> **S** : a **Spring supports** the joint in the given direction. Dx, Dy or Dz indicate the Spring constant.

Rx, **Ry** and **Rz** indicate a translation Restraint in the Global X-, Y- and Z-direction. When for instance Rx = R, translation is restricted in the Global X-direction. As a result, a Reaction Force in the X-direction will act upon the structure from the Support Joint.

Mx, **My** and **Mz** indicate a rotation Restraint round the Global X-, Y- and Z-direction. When for instance Mx = R, rotation is restricted round the Global X-axis. As a result, a Reaction Moment round the X-axis will act upon the structure from the Support Joint.

Since the Members in a Truss are connected with Hinges, Trusses cannot have rotation Restraints.

The Joints in a two-dimensional Frame can only have translation Restraints in the Global X- and Y-direction and a rotation Restraint round the Z-axis.

Dx, Dy and **Dz** indicate the magnitude of the forced displacements or spring constants. The meaning of the value depends on the contents of the corresponding translation Restraint.

Forced displacements can be entered in mm or inches depending on the Units option from the Structure Menu. Accordingly, Spring constants can be given in kN/cm or Kips/inch.

Examples:


Pinned support:
Rx = R   Ry = R   Mz =


Horizontal roller:
Rx =   Ry = R   Mz =


Fixed support:
Rx = R   Ry = R   Mz = R

The top left of the Support windows displays the most commonly used types of supports. Click on one of these images to choose your type of support.

Since the drawing of the support can only show the Restrictions in two dimensions, the drawing will change according to the point of view in a three-dimensional structure.

Support Joints can be entered both with the Menu and with the mouse.

3.6.1. Entering Support Joints from the Structure Menu

To add, change or remove Support Joints in the Restraints list simply select the Support Joint option from the Structure menu.

Adding new Support Joints: Enter the Joint Number and the restrictions for each axis. Frames can also have rotation Restraints besides the translation Restraints.

To add the Support Joints to the list simply press ENTER in the last edit box, or press the ADD button. After each new added Support, the Support Joints Number will automatically increase by one.

Removing Restraints: Select the Support(s) and press the DELETE button.

Changing Restraints: This can be done in 3 ways:

1) Type the Support Number and press ENTER.
2) Double click the Joint number with the left mouse button.
3) Select the Joint number and press the Edit button.

In both cases the Support Joint will be removed from the list to the edit boxes, where you can edit the Restraint data. Don't forget to add the Support Joints to the list after editing. Editing the Supports directly in the list is not allowed.

3.6.2. Entering Support Joints interactive

To add, change or remove Support Joints first select the Joint by double clicking the left mouse button. In the lower right corner of the screen appears a window to enter the Joint coordinates. The Support Joints can be edited by selecting the SUPPORT tab.

To clear a Support Joint clear all restraints. Pressing the - button will also delete the Joint displayed.

This method allows you to view each change on the screen as soon as it is entered.

3.7. Cross Sectional Data

Analysis3D has an extensive library of standard Member Cross Sections. The list can be easily updated.

The Units and Region option from the Structure Menu allows you to select a different Library of Cross Sections. Also, the units are specified by the same Structure Menu selection.

You can also download or save a different file with Cross Sections using the File menu from the top left of the Cross Section window. *Analysis3D* offers a wide selection of Libraries from different regions to choose from. E.g., Euro, US AISC, UK British Steel, Russia GOST, China GB, India BIS, Japan JIS, Australia and New Zealand and Timber.

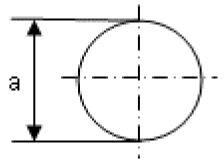
Each Cross Section has a unique **Section Name**:

Shape: Identifies the section out of the following choices: I-section, U-section or channel, L-section, T-section or half I-section, Rectangular box, Circular tube, Full section.
Fabric: How was the section made: Rolled or Welded

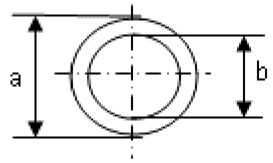
Ax : the Cross Sectional Area in [cm2] or [inch2]
h : total height of the section in [mm] or [inch]
d : total width of the section in [mm] or [inch]
tw : web thickness in [mm] or [inch]
tf : flange thickness in [mm] or [inch]
r : radius between flange and web in [mm] or [inch]

Ix : the Cross Section Moment of Inertia about the x axis in [cm4] or [inch4]
(as a result of a Load along the local y-axis)
Iy : the Cross Section Moment of Inertia about the y axis [cm4] or [inch4]
(as a result of a Load along the local z-axis)

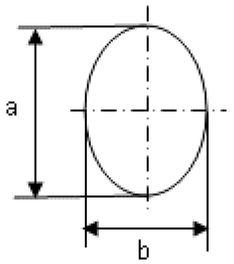
It : the Cross Section Torsional Constant [cm^4] or [$inch^4$]
(as a result of a Moment round the local x-axis)
For a round section the Torsional Constant is equal to the polar Moment of Inertia.



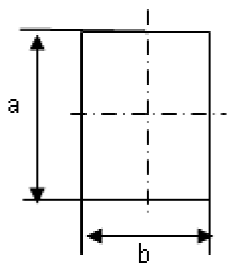
$I_t = \pi/32\ a^4$



$I_t = \pi/32\ a^4\ (1-(b/a)^4)$

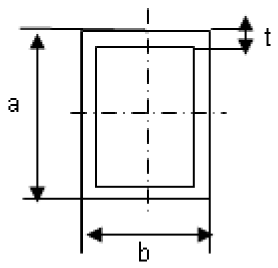


$I_t = \pi/16\ a^3\ b^3\ /\ (a^2+b^2)$

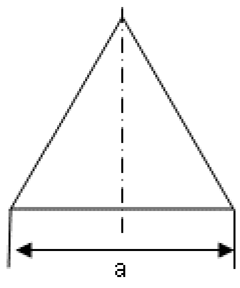


$I_t = \mu\ a\ b^3$

a/b	μ
1	0.141
1.5	0.196
2	0.229
3	0.263
4	0.281
6	0.298
8	0.307
10	0.312
∞	0.333



$I_t = 2\ b^2\ a^2\ \ t\ /\ (b+a)$



$I_t = a^4\ /\ 46.2$

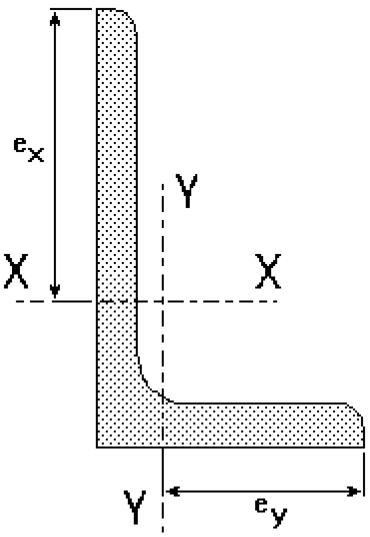
Wx, Wy : Elastic Section Modulus in [cm3] or [inch3]

Zx, Zy : Plastic Section Modulus in [cm3] or [inch3]

ex : the distance from the neutral line of the Member to the outside of the Member along the main axis of that Member. In the case of asymmetrical Cross Sections this will usually be the longest distance.

ey : the distance from the neutral line of the Member to the outside of the Member along the secondary axis of that Member.

For both ex and ey it is important to use the longest distance to the outside of the member as shown in the figure below.



et : for closed sections et is the greater of ex or ey. In the case of open sections et is the thickness of the thickest part of the section. This usually is the maximum flange thickness of the beam cross section.

et for full rectangular beams: µ b (see Torsional Constant above)

a/b	µ
1	0.675
1.5	0.852
2	0.928
3	0.977
4	0.990
6	0.997
8	0.999
>=10	1.000

et for full open sections: maximum thickness (of flange or body)

M : the **Mass** of the Member per meter [kg/m] or [kg/ft]

3.8. Material Data

Each Material is defined by:

Material: a unique name for the Material

E : the Modulus of Elasticity in [kN/cm²] or [Kips/inch²]

G : the Shear Modulus in [kN/cm²] or [Kips/inch²]
G = E / 2(1+v) with: v = Poisson constant;
for metals v = ±1/3

Re : the Limit Stress of the material in [N/mm²] or [Kips/inch²] = 0.7 Yield Point or Tensile Strength.

The **Linear Coefficient of Expansion** in 10^{-6} 1/K.

The units are specified by the Structure Menu option Units.

3.9. Units and Region

All data can be expressed in International SI units, in US Units or Continental Units. You can select your preferred Units from the drop-down list.

It is possible to enter the Joint coordinates in International SI Units, the Forces in American Units and view the results in both International and American Units. Toggling between Units can introduce a minor deviation due to rounding off.

Below are the most commonly used Units with the corresponding conversion factors.

	SI Units	**US Units**	**Continental Units**
Joint Coordinates:	meter [m]	foot [ft]	meter [m]
Forces:	kNewton [kN]	thousands of pounds [Kips]	kilogram [kg]
Moments:	[kNm]	[Kip-ft]	[kgm]
Distributed Load:	[kN/m]	[Kip/ft]	[kg/m]
Joint Displacements:	[mm]	Inch [in]	[mm]
Stresses:	[N/mm²]	[Kips/in²]	[kgf/mm²]

1 foot = 0.3048 meter
1 inch = 25.4 mm
1 pound [lbf] = 4.44822 Newton
1 kgf = 9.80665 Newton
1 N/mm² = 1MPa

Region

You can select from the drop-down list the Region for which the Cross-Sectional Data will be selected.
The following Regions are available with Lists of Cross Sections:

- Australia New Zealand
- China GB
- Euro
- India BIS
- Japan JIS

- Russia GOST
- UK British Steel
- US AISC (available both in metric and in imperial units)
- Other Sections
- Timber

On request, this list can be easily expanded or amended to fit your needs.

Standard used

This selection allows you to choose which International or Local Standard will be used for optimizing the structure with the Member Design or Detailed Design options from the Result Menu.
Analysis3D offers the following Standards to choose from:

- AISC ASD
- AISC LRFD
- BS 5950
- CISC 94
- Eurocode 3

Your selection of Units and Region needs to be stored in the Library file for this to remain active when you restart *Analysis3D* as your default selection.

4. The Loads Menu

4.1. Joint Loads

A Joint Load is defined by the Joint Number and the Forces and Moments acting on that Joint.

You can use up to 9 different **Load Cases**. Each Load Case is defined by the Load Case number from 1 to 9. The importance of each Load Case is determined with the Load Combinations option from the structure menu.

The Forces in kN or Kips are parallel with the Global coordinate System as drawn in the left bottom of the screen. Thus, a positive Force Fy points up. By default, Forces are represented in red.

Moments in kNm or Kips-ft are positive according the right-hand rule round the respective global coordinate axis. In two-dimensional Structures the Moments are drawn as an arc, in the case of three dimensions the moments are drawn as an arrow, by default painted purple.

The units are specified by the Structure Menu option Units and Region.

Since the Members in a Truss are connected with Hinges, Trusses cannot have Moments as Joint Loads.

The Joints in a two-dimensional Frame can only have Forces in the Global X- and Y-direction and a Moment round the Z-axis.

Joint Loads can be entered from the Structure Menu or interactively.

To enter Joint Loads interactively, first select the desired Joint and press the LOAD tab.

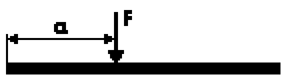
4.2. Member Load

A Member Load is defined by the Member Number and the Forces in kN or Kips and Moments in kNm or Kip-ft acting on that Member.

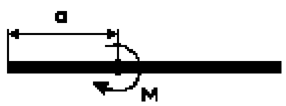
You can use up to 9 different **Load Cases**. Each Load Case is defined by the Load Case number from 1 to 9. The importance of each Load Case is determined with the **Load Combinations** option from the structure menu.

There are five different basic **Load Types** for Member Loads. For simplicity these five Load Types get an arbitrary Load Type Number as indicated below. But you can also simply click at the bottom of the screen on the Load Type you want to use and Analysis3D will select the corresponding Load Type number for you:

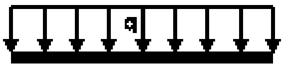
1 : A **Point Load** F in [kN] or [Kips] along the x-, y- or z-axis, at a distance a in [m] or [ft] from the begin Joint.



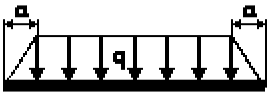
2 : A **Moment** M in [kNm] or [Kip-ft] round the x-, y- or z-axis, at a distance a in [m] or [ft] from the start Joint.



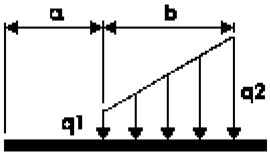
3 : A **Distributed Load** q in [kN/m] or [Kips/ft] along the x-, y- or z-axis.



4 : A **Symmetrical Trapezoidal Distributed Load** q in [kN/m] or [Kips/ft] along the x-, y- or z-axis with transition a in [m] or [ft].



5 : a **Non Symmetrical Trapezoidal Distributed Load** along the x-, y- or z-axis that starts with a Load Q1 in [kN/m] or [Kips/ft] at a distance a in [m] or [ft] from the begin Joint and that is b [m] or [ft] long. Thus, the end Load Q2 in [kN/m] or [Kips/ft] is on a distance a+b from the begin Joint.



L/G: Forces and Moments can be entered according the **G**lobal or the **L**ocal coordinate System. Entering L or G in the L/G field makes this choice. Forces that are parallel or perpendicular to a Member with a slope can easily be entered with the Local Coordinate System.

Member Loads can be entered from the Structure Menu or interactive with the mouse.

To enter Member Loads **interactively**, first select the Member with the mouse, and press the LOAD tab. Select the Load case and Load type by using the arrow buttons. To delete a load, all forces have to be cleared.

4.3. Wind Load

Wind Load is calculated according to Eurocode 1991-1-4 or according to ASCE 7-16. Analysis3D also provides a General option not restricted by a national code.

Wind Load according to Eurocode 1991-1-4 (EN 1991-1-4:2005+A1 April 2010)

For Wind Load calculations there are basically 2 options:
- for an open structure only the wind load on the beams is calculated
- for a closed structure the calculation assumes the walls are closed and the wind acts on the surface of the entire closed area.

Wind Load calculation according to Eurocode 1991-1-4 starts from the fundamental value of the basic wind velocity Vb0. This is the characteristic 10 minutes mean wind velocity, irrespective of wind direction and time of year, at 10 m above ground level, in open country terrain. It corresponds to a mean return period of 50 years.

The Basic wind velocity Vb = Cdir . Cseason . Cprob . Calt . Vb0.

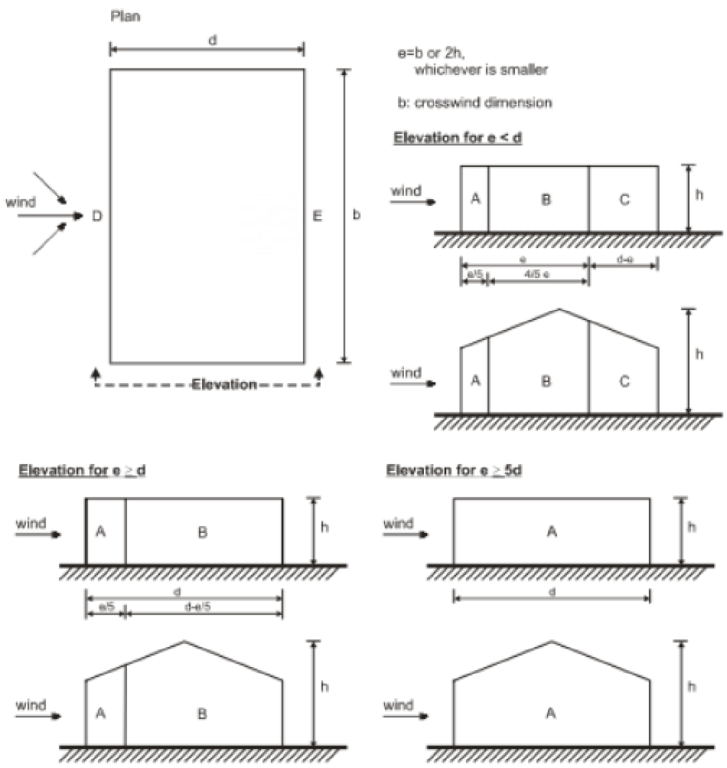
Cprob is calculated with the input given using the formula: $$c_{prob} = \left(\frac{1 - K \cdot \ln(-\ln(1-p))}{1 - K \cdot \ln(-\ln(0{,}98))} \right)^n$$

The mean wind velocity Vm(z) at height z above the terrain is calculated as: Vm(z) = Cr(z).Co(z).Vb

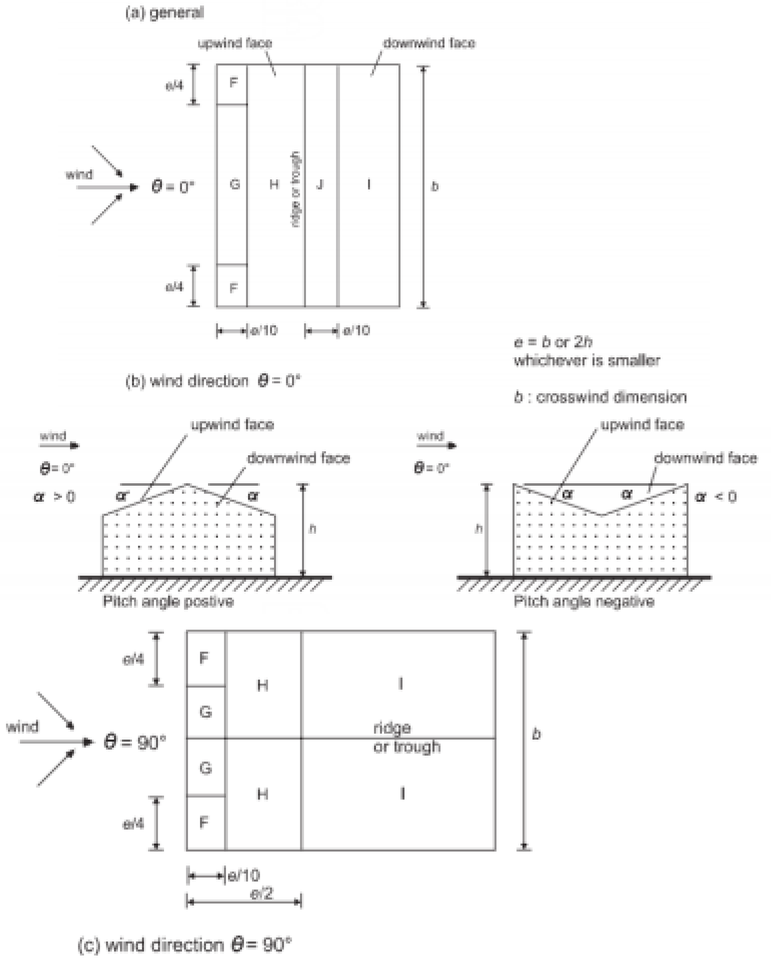
The roughness factor Cr(z) is calculated based in the Terrain Category specified and the formula
Cr(z)= kr. ln(z/z0)

Peak velocity pressure is given by qp(z) = [1+7Iv(z)] ½ .ρ. $Vm^2(z)$
Where Iv(z) is the turbulence intensity which allows to take into account the contribution from short term fluctuations Iv(z) = k1 / [Co(z). ln(z/z0)].
The wind pressure acting on the external surface: We = qp(z).Cp
Where Cp is the pressure coefficient as indicated in the overview per zone.

For walls:



For duopitch roofs:



Please note that Eurocode requires to investigate the option with positive and with negative pressure coefficients. For duopitch roofs, 4 combinations are required for investigation, consisting on positive and negative pressure coefficients on each side of the roof.
Finally the forces on the structure are calculated using $F_{we} = C_sC_d.\sum W_e.A_{ref}$.

All calculated values are shown in grey cells, the values that require input are left white. However; most of the non-calculated cells are completed with default values already.

The button Calculate determines the pressure coefficients as per Eurocode. However, the user has the freedom to overwrite any number before applying the Wind Loads to the structure. When pressing the button Apply Wind Load, *Analysis3D* will use the numbers displayed for the final calculation.

To apply these forces, the user can select the Load Case the Wind Loads should be attributed to and they will be added to the list of Member Loads.

With the option Load Combination from the main menu, the user can choose the relative weight of the Wind Load for the combined calculation.

Wind Load according to ASCE 7-16

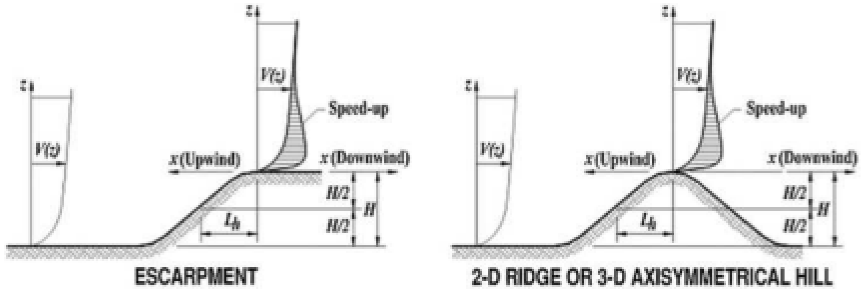
For Wind Load calculations there are basically 2 options:

- for an open structure only the wind load on the beams is calculated

- for enclosed or partially enclosed buildings, the calculation assumes the walls are closed and the wind acts on the surface of the entire area.

Wind Load calculation according to ASCE 7-16 starts from the basic wind speed Vb. This is the nominal design 3-second gust wind speed at 10m (33ft) above ground for Exposure C category. These correspond to approximately a 3% probability of exceedance in 50 years.
The velocity pressure q is calculated as $q = 0.613 . K_z. K_{zt}. K_d. V_b^2$
Where the Topographic Factor Kzt is determined by $K_{zt} = (1 + K_1 . K_2 . K_3)^2$ and derived from figure 26.8-1 below.

Diagrams



Topographic Multipliers for Exposure C[a,b,c]

	K_1 Multiplier				K_2 Multiplier			K_3 Multiplier		
H/L_h	2D Ridge	2D Escarpment	3D Axisym-metrical Hill	x/L_h	2D Escarpment	All Other Cases	z/L_h	2D Ridge	2D Escarpment	3D Axisym-metrical Hill
0.20	0.29	0.17	0.21	0.00	1.00	1.00	0.00	1.00	1.00	1.00
0.25	0.36	0.21	0.26	0.50	0.88	0.67	0.10	0.74	0.78	0.67
0.30	0.43	0.26	0.32	1.00	0.75	0.33	0.20	0.55	0.61	0.45
0.35	0.51	0.30	0.37	1.50	0.63	0.00	0.30	0.41	0.47	0.30
0.40	0.58	0.34	0.42	2.00	0.50	0.00	0.40	0.30	0.37	0.20
0.45	0.65	0.38	0.47	2.50	0.38	0.00	0.50	0.22	0.29	0.14
0.50	0.72	0.43	0.53	3.00	0.25	0.00	0.60	0.17	0.22	0.09
				3.50	0.13	0.00	0.70	0.12	0.17	0.06
				4.00	0.00	0.00	0.80	0.09	0.14	0.04
							0.90	0.07	0.11	0.03
							1.00	0.05	0.08	0.02
							0.50	0.01	0.02	0.00
							2.00	0.00	0.00	0.00

[a]For values of H/L_h, x/L_h, and z/L_h other than those shown, linear interpolation is permitted.
[b]For $H/L_h > 0.5$, assume that $H/L_h = 0.5$ for evaluating K_1 and substitute $2H$ for L_h for evaluating K_2 and K_3.
[c]Multipliers are based on the assumption that wind approaches the hill or escarpment along the direction of maximum slope.

Notation

H = Height of hill or escarpment relative to the upwind terrain, in ft (m).
K_1 = Factor to account for shape of topographic feature and maximum speed-up effect.
K_2 = Factor to account for reduction in speed-up with distance upwind or downwind of crest.
K_3 = Factor to account for reduction in speed-up with height above local terrain.
L_h = Distance upwind of crest to where the difference in ground elevation is half the height of hill or escarpment, in ft (m).
x = Distance (upwind or downwind) from the crest to the site of the building or other structure, in ft (m).
z = Height above ground surface at the site of the building or other structure, in ft (m).
μ = Horizontal attenuation factor.
γ = Height attenuation factor.

Equations

$K_{zt} = (1 + K_1 K_2 K_3)^2$
K_1 = determined from table below
$K_2 = (1 - |x|/\mu L_h)$
$K_3 = e^{-\gamma z/L_h}$

Parameters for Speed-Up over Hills and Escarpments

	$K_1/(H/L_h)$				μ	
	Exposure					
Hill Shape	B	C	D	γ	Upwind of Crest	Downwind of Crest
2D ridges (or valleys with negative H in $K_1/(H/L_h)$)	1.30	1.45	1.55	3	1.5	1.5
2D escarpments	0.75	0.85	0.95	2.5	1.5	4
3D axisymmetrical hill	0.95	1.05	1.15	4	1.5	1.5

FIGURE 26.8-1 Topographic Factor, K_{zt}

The velocity pressure exposure coefficient Kz = 2.01 . $(z/zg)^{2/alpha}$. For z < 15 ft (4.6m) : Kz = 2.01 . $(15/zg)^{2/alpha}$
The terrain exposure coefficients are derived from Table 26.11-1 below

Table 26.11-1 Terrain Exposure Constants

Customary Units										
Exposure	α	z_g (ft)	$\hat{a}$	$\hat{b}$	$\bar{\alpha}$	$\bar{b}$	c	ℓ (ft)	$\bar{\epsilon}$	z_{min} (ft)[a]
B	7.0	1,200	1/70	0.84	1/4.0	0.45	0.30	320	1/3.0	30
C	9.5	900	1/9.5	1.00	1/6.5	0.65	0.20	500	1/5.0	15
D	11.5	700	1/11.5	1.07	1/9.0	0.80	0.15	650	1/8.0	7
S.I. Units										
Exposure	α	z_g (m)	$\hat{a}$	$\hat{b}$	$\bar{\alpha}$	$\bar{b}$	c	ℓ (m)	$\bar{\epsilon}$	z_{min} (m)[a]
B	7.0	365.76	1/7	0.84	1/4.0	0.45	0.30	97.54	1/3.0	9.14
C	9.5	274.32	1/9.5	1.00	1/6.5	0.65	0.20	152.40	1/5.0	4.57
D	11.5	213.36	1/11.5	1.07	1/9.0	0.80	0.15	198.12	1/8.0	2.13

[a]z_{min} = minimum height used to ensure that the equivalent height $\bar{z}$ is the greater of $0.6h$ or z_{min}. For buildings or other structures with $h \leq z_{min}$, $\bar{z}$ shall be taken as z_{min}.

K_d is the Wind Directionality factor and can be derived from table 26.6-1 below.

Table 26.6-1 Wind Directionality Factor, K_d

Structure Type	Directionality Factor K_d
Buildings	
Main Wind Force Resisting System	0.85
Components and Cladding	0.85
Arched Roofs	0.85
Circular Domes	1.0[a]
Chimneys, Tanks, and Similar Structures	
Square	0.90
Hexagonal	0.95
Octagonal	1.0[a]
Round	1.0[a]
Solid Freestanding Walls, Roof Top Equipment, and Solid Freestanding and Attached Signs	0.85
Open Signs and Single-Plane Open Frames	0.85
Trussed Towers	
Triangular, square, or rectangular	0.85
All other cross sections	0.95

[a]Directionality factor $K_d = 0.95$ shall be permitted for round or octagonal structures with nonaxisymmetric structural systems.

Design wind pressures for the MWFRS (Main Wind Force Resisting System in Directional Procedure) of buildings of all heights is determined by the following equation: p = q . G . Cp – qi . GCpi (N/m2)

External pressure coefficients are calculated by Analysis3D based on Figure 27.3-1 below

Wall Pressure Coefficients, C_p

Surface	L/B	C_p	Use With
Windward wall	All values	0.8	q_z
Leeward wall	0–1	−0.5	q_h
	2	−0.3	q_h
	≥4	−0.2	q_h
Sidewall	All values	−0.7	q_h

Roof Pressure Coefficients, C_p, for use with q_h

		Windward								Leeward		
		Angle, θ (degrees)								Angle, θ (degrees)		
Wind Direction	h/L	10	15	20	25	30	35	45	≥60[c]	10	15	≥20
Normal	≤0.25	−0.7	−0.5	−0.3	−0.2	−0.2	0.0[a]					
to Ridge		−0.18	0.0[a]	0.2	0.3	0.3	0.4	0.4	0.01 θ	−0.3	−0.5	−0.6
for	0.5	−0.9	−0.7	−0.4	−0.3	−0.2	−0.2	0.0[a]				
θ ≥ 10°		−0.18	−0.18	0.0[a]	0.2	0.2	0.3	0.4	0.01 θ	−0.5	−0.5	−0.6
	≥1.0	−1.3[b]	−1.0	−0.7	−0.5	−0.3	−0.2	0.0[a]				
		−0.18	−0.18	−0.18	0.0[a]	0.2	0.2	0.3	0.01 θ	−0.7	−0.6	−0.6

Wind Direction	h/L	Horizontal Distance from Windward Edge	C_p
Normal to Ridge for θ < 10° and Parallel to Ridge for All θ	≤0.5	0 to $h/2$	−0.9, −0.18
		$h/2$ to h	−0.9, −0.18
		h to $2h$	−0.5, −0.18
		$>2h$	−0.3, −0.18
	≥1.0	0 to $h/2$	−1.3[b], −0.18
		$>h/2$	−0.7, −0.18

[a]Value is provided for interpolation purposes.
[b]Value can be reduced linearly with area over which it is applicable as follows:
[c]For roof slopes greater than 80°, use $C_p = 0.8$.

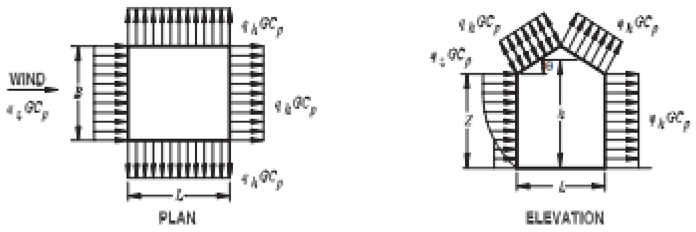
Area, ft²	Area, m²	Reduction Factor
≤100	≤9.3	1.0
250	23.2	0.9
≥1,000	≥92.9	0.8

Notes

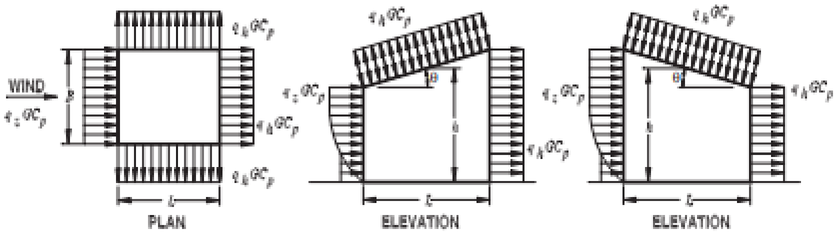
1. Plus and minus signs signify pressures acting toward and away from the surfaces, respectively.
2. Linear interpolation is permitted for values of L/B, h/L, and θ other than shown. Interpolation shall only be carried out between values of the same sign. Where no value of the same sign is given, assume 0.0 for interpolation purposes.
3. Where two values of C_p are listed, this indicates that the windward roof slope is subjected to either positive or negative pressures and the roof structure shall be designed for both conditions. Interpolation for intermediate ratios of h/L in this case shall only be carried out between C_p values of like sign.
4. For monoslope roofs, entire roof surface is either a windward or leeward surface.
5. Refer to Fig. 27.3-2 for domes and Fig. 27.3-3 for arched roofs.
6. For mansard roofs, the top horizontal surface and leeward inclined surface shall be treated as leeward surfaces from the table.
7. Except for MWFRSs at the roof consisting of moment-resisting frames, the total horizontal shear shall not be less than that determined by neglecting wind forces on roof surfaces.

FIGURE 27.3-1 (*Coninued*). Main Wind Force Resisting System, Part 1 (All Heights): External Pressure Coefficients, C_p, for Enclosed and Partially Enclosed Buildings—Walls and Roofs

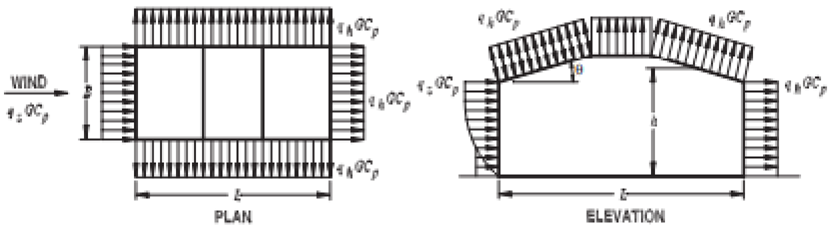
And the wind loads are applied according to :



GABLE, HIP ROOF



MONOSLOPE ROOF (NOTE 4)



MANSARD ROOF (NOTE 8)

All calculated values are shown in grey cells the values that require input are left white. However; most of the non-calculated cells are completed with default values.

The button Calculate determines the pressure coefficients as per Eurocode. However, the user has the freedom to overwrite any number before applying the Wind Loads to the structure. When pressing the button Apply Wind Load, *Analysis3D* will use the numbers displayed for the final calculation.

To apply these forces, the user can select the Load Case the Wind Loads should be attributed to and they will be added to the list of Member Loads.

With the option Load Combination from the main menu, the user can choose the relative weight of the Wind Load for the combined calculation.

Wind Loads for General application

Since it is almost impossible to include each and every standard for each and every country, the “General” option minimizes the predetermined calculations and just allow the user to let Analysis3D apply the Wind Loads as specified by the user (according to the national standard applicable).

Based on the velocity pressure and the pressure coefficients specified by the user, Analysis3D will calculate the Wind Load for each wall and roof. For the coefficients, the following rule applies: positive pressure coefficients are pointing towards the building, negative coefficients are pointing away from the building.

For Open structures, Analysis3D will only calculate the wind on the surface of the steel structure, assuming no walls exist.

4.4. Snow Load

Analysis3D offers 3 options to calculate Snow Loads. Snow Loads can be defined according to ASCE/SEI 7-16, according to Eurocode 1 - Actions on structures - Part 1-3 or using a General method not restricted by any specific standards.

Snow Load according to ASCE/SEI 7-16

Before we can apply any snow loads to our structure, we need to know the ground snow load (p_g) at our location, which can be found using Figure 7.2-1 from ASCE 7-16.
Also, you can directly find the ground snow load for your location through online Hazards by Location tool, provided by ATC.
In some special cases, site-specific case studies are needed to determine ground snow loads and therefore cannot directly be found on the map provided.

The snow load that is applied to our structure is not the ground snow load, but in most cases, the flat roof snow load (roof slope ≤ 5°).
The flat roof snow load is calculated using formula below:

$$pf = 0.7\ C_e\ C_t\ I_s\ p_g$$

Where:

p_g = Ground Snow Load
C_e = Exposure Factor from table 7.3-1 below

Table 7.3-1 Exposure Factor, C_e

Surface Roughness Category	Exposure of Roof[a] Fully Exposed	Partially Exposed	Sheltered
B (see Section 26.7)	0.9	1.0	1.2
C (see Section 26.7)	0.9	1.0	1.1
D (see Section 26.7)	0.8	0.9	1.0
Above the tree line in windswept mountainous areas	0.7	0.8	NA
In Alaska, in areas where trees do not exist within a 2-mi (3-km) radius of the site	0.7	0.8	NA

The terrain category and roof exposure condition chosen shall be representative of the anticipated conditions during the life of the structure. An exposure factor shall be determined for each roof of a structure.
[a]Definitions: Partially Exposed: All roofs except as indicated in the following text. Fully Exposed: Roofs exposed on all sides with no shelter[b] afforded by terrain, higher structures, or trees. Roofs that contain several large pieces of mechanical equipment, parapets that extend above the height of the balanced snow load (h_b), or other obstructions are not in this category. Sheltered: Roofs located tight in among conifers that qualify as obstructions.
[b]Obstructions within a distance of $10h_o$ provide "shelter," where h_o is the height of the obstruction above the roof level. If the only obstructions are a few deciduous trees that are leafless in winter, the "fully exposed" category shall be used. Note that these are heights above the roof. Heights used to establish the Exposure Category in Section 26.7 are heights above the ground.

Where the Surface Roughness Categories as defined as below:

Surface Roughness B: Urban and suburban areas, wooded areas, or other terrain with numerous, closely spaced obstructions that have the size of single-family dwellings or larger.

Surface Roughness C: Open terrain with scattered obstructions that have heights generally less than 30 ft (9.1 m). This category includes flat, open country and grasslands.

Surface Roughness D: Flat, unobstructed areas and water surfaces. This category includes smooth mud flats, salt flats, and unbroken ice

Ct = Thermal Factor from the Table 7.3-2 below

Table 7.3-2 Thermal Factor, C_t

Thermal Condition[a]	C_t
All structures except as indicated below	1.0
Structures kept just above freezing and others with cold, ventilated roofs in which the thermal resistance (R-value) between the ventilated space and the heated space exceeds $25°F \times h \times ft^2/Btu$ ($4.4\ K \times m^2/W$)	1.1
Unheated and open air structures	1.2
Freezer building	1.3
Continuously heated greenhouses[b] with a roof having a thermal resistance (R-value) less than $2.0°F \times h \times ft^2/Btu$ ($0.4\ K \times m^2/W$)	0.85

[a]These conditions shall be representative of the anticipated conditions during winters for the life of the structure.
[b]Greenhouses with a constantly maintained interior temperature of 50°F (10°C) or more at any point 3 ft (0.9 m) above the floor level during winters and having either a maintenance attendant on duty at all times or a temperature alarm system to provide warning in the event of a heating failure.

Is = Importance Factor: The value for Is shall be determined from Table 1.5-2 based on the Risk Category from Table 1.5-1 below.

Table 1.5-2 Importance Factors by Risk Category of Buildings and Other Structures for Snow, Ice, and Earthquake Loads

Risk Category from Table 1.5-1	Snow Importance Factor, I_s	Ice Importance Factor—Thickness, I_i	Ice Importance Factor—Wind, I_w	Seismic Importance Factor, I_e
I	0.80	0.80	1.00	1.00
II	1.00	1.00	1.00	1.00
III	1.10	1.15	1.00	1.25
IV	1.20	1.25	1.00	1.50

Table 1.5-1 Risk Category of Buildings and Other Structures for Flood, Wind, Snow, Earthquake, and Ice Loads

Use or Occupancy of Buildings and Structures	Risk Category
Buildings and other structures that represent low risk to human life in the event of failure	I
All buildings and other structures except those listed in Risk Categories I, III, and IV	II
Buildings and other structures, the failure of which could pose a substantial risk to human life	III
Buildings and other structures, not included in Risk Category IV, with potential to cause a substantial economic impact and/or mass disruption of day-to-day civilian life in the event of failure	
Buildings and other structures not included in Risk Category IV (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, hazardous waste, or explosives) containing toxic or explosive substances where the quantity of the material exceeds a threshold quantity established by the Authority Having Jurisdiction and is sufficient to pose a threat to the public if released[a]	
Buildings and other structures designated as essential facilities	IV
Buildings and other structures, the failure of which could pose a substantial hazard to the community Buildings and other structures (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, or hazardous waste) containing sufficient quantities of highly toxic substances where the quantity of the material exceeds a threshold quantity established by the Authority Having Jurisdiction and is sufficient to pose a threat to the public if released[a]	
Buildings and other structures required to maintain the functionality of other Risk Category IV structures	

Sloped Roof Snow Loads: p_s
The Snow loads acting on a sloping surface shall be assumed to act on the horizontal projection of that surface. The sloped roof snow load, p_s, shall be obtained by multiplying the flat roof snow load, pf , by the

roof slope factor, Cs:

$$p_s = C_s\, p_f$$

The values of C_s for warm roofs and cold roofs are determined from Figure 7.4-1 below.
For roofs with an unobstructed slippery surface that allows snow to slide off the eaves, the roof slope factor Cs shall be determined using the dashed line in Fig. 7.4-1.
For all other roofs, the solid line in Fig. 7.4-1 shall be used to determine the roof slope factor Cs.

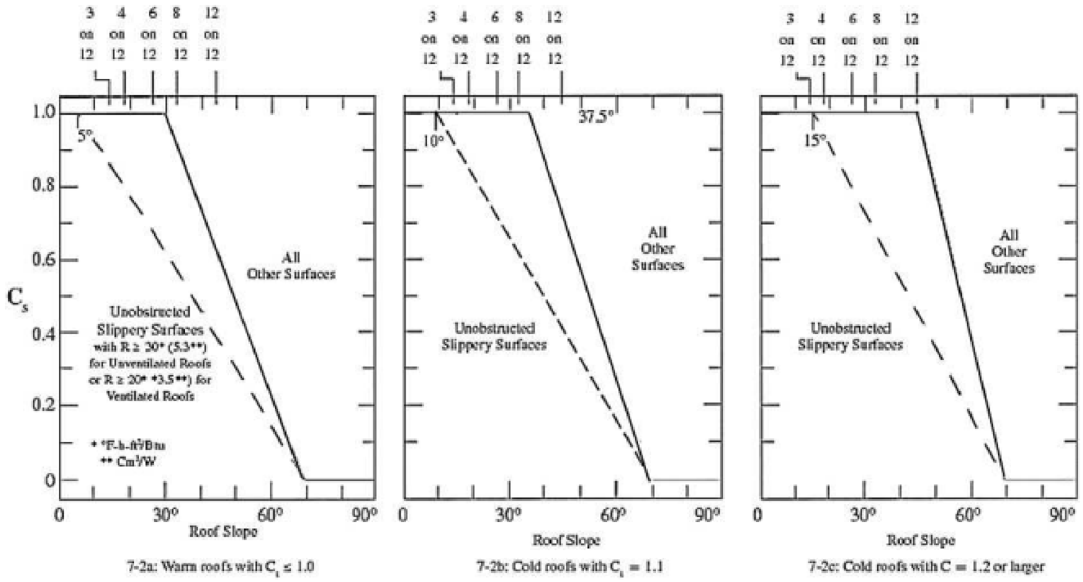


FIGURE 7.4-1 Graphs for Determining Roof Slope Factor, C_s, for Warm and Cold Roofs (See Table 7.3-2 for C_t Definitions)

After Calculating the Snow Load, we can use the Apply Snow Load button to distribute the Snow Load accordingly over the roof.

Snow Load according to Eurocode 1 (EN 1991-1-3:2003)

In order to calculate the Characteristic Snow Load on the ground Sk, we first, need to select the climatic region from the drop-down box.
In each climatic region a given load-altitude correlation formula applies and this is given in Table C.1 below.
Different zones are defined for each climatic region. Each zone is given a Zone number Z, which is used in the load altitude correction formula.
The National Annex specifies the characteristic values to be used. To cover unusual local conditions the National Annex may additionally allow the client and the relevant authority to agree upon a different characteristic value from that specified for an individual project.
Annex C of EN 1991-1-3:2003 gives the European ground snow load map. The National Annex may make reference to this map in order to eliminate, or to reduce, inconsistencies occurring at borderlines between countries.

Table C.1. Altitude - Snow Load Relationships

Climatic Region	*Expression*
Alpine Region	$s_k = (0{,}642Z + 0{,}009)\left[1 + \left(\frac{A}{728}\right)^2\right]$
Central East	$s_k = (0{,}264Z - 0{,}002)\left[1 + \left(\frac{A}{256}\right)^2\right]$
Greece	$s_k = (0{,}420Z - 0{,}030)\left[1 + \left(\frac{A}{917}\right)^2\right]$
Iberian Peninsula	$s_k = (0{,}190Z - 0{,}095)\left[1 + \left(\frac{A}{524}\right)^2\right]$
Mediterranean Region	$s_k = (0{,}498Z - 0{,}209)\left[1 + \left(\frac{A}{452}\right)^2\right]$
Central West	$s_k = 0{,}164Z - 0{,}082 + \frac{A}{966}$
Sweden, Finland	$s_k = 0.790Z \div 0{,}375 + \frac{A}{336}$
UK, Republic of Ireland	$s_k = 0{,}140Z - 0{,}1 + \frac{A}{501}$

Where:

S_k is the characteristic snow load on the ground [kN/m2]
A is the site altitude above Sea Level [m]
Z is the zone number given on the map.

The Snow Load on the Roof (s) is calculated usin the formula below:

$$s = C_t\, C_e\, \mu_i\, s_k$$

Where the thermal coefficient C_t should be used to account for the reduction of snow loads on roofs with high thermal transmittance (> 1 W/m2 K), in particular for some glass covered roofs, because of melting caused by heat loss. For all other cases: $C_t = 1{,}0$

C_e = Exposure coefficient determined from the table below.

Table 5.1 Recommended values of C_e for different topographies

Topography	C_e
Windswept [a]	0,8
Normal [b]	1,0
Sheltered [c]	1,2

[a] *Windswept topography*: flat unobstructed areas exposed on all sides without, or little shelter afforded by terrain, higher construction works or trees.

[b] *Normal topography*: areas where there is no significant removal of snow by wind on construction work, because of terrain, other construction works or trees.

[c] *Sheltered topography*: areas in which the construction work being considered is considerably lower than the surrounding terrain or surrounded by high trees and/or surrounded by higher construction works.

The snow load shape coefficient µ1 that should be used for sloped roofs is given in Table 5.2 below. The values given in Table 5.2 apply when the snow is not prevented from sliding off the roof. Where snow fences or other obstructions exist or where the lower edge of the roof is terminated with a parapet, then the snow load shape coefficient should not be reduced below 0,8.
The values of µ1 apply for undrafted load arrangements, µ2 applies for drifted load arrangements,

Table 5.2: Snow load shape coefficients

Angle of pitch of roof α	$0° \leq \alpha \leq 30°$	$30° < \alpha < 60°$	$\alpha \geq 60°$
μ_1	0,8	$0{,}8(60 - \alpha)/30$	0,0
μ_2	$0{,}8 + 0{,}8\ \alpha/30$	1,6	--

The following 3 cases need to be investigated for a pitched roof/

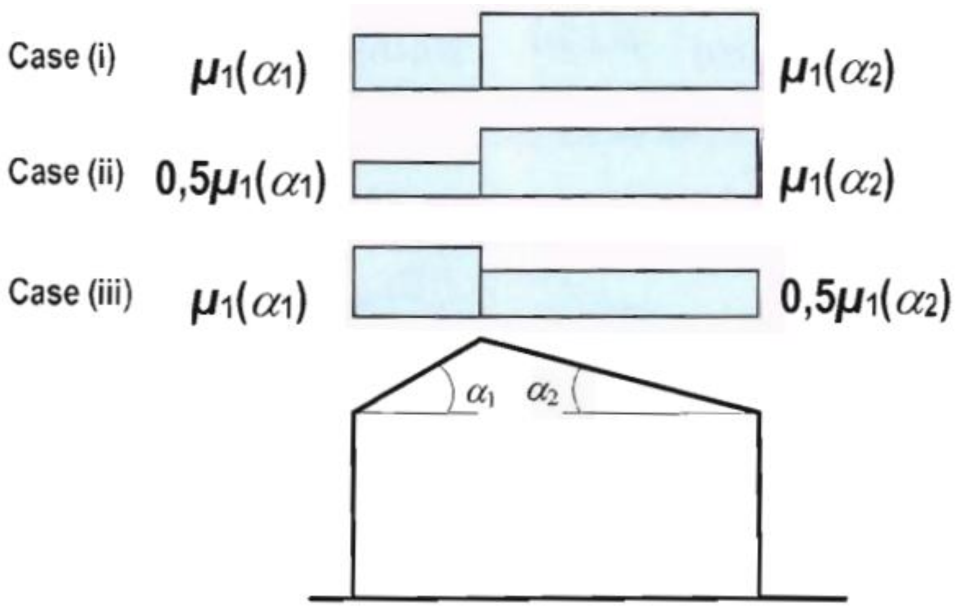


Figure 5.3: Snow load shape coefficients - pitched roofs

After Calculating the Snow Load, we can use the Apply Snow Load button to distribute the Snow Load accordingly over the roof.

Snow Load for general application

Since it is almost impossible to include each and every standard of each and every country, the "General" option minimizes the predetermined calculations and just allow the user to let Analysis3D apply the Snow Load on the Roof as given by the user (according to the national standard applicable).

4.5. Seismic Loads

Analysis3D offers 3 options to calculate Seismic Loads. Seismic Loads can be defined according to Eurocode 8 (EN 1998: 2004) or according to ASCE/SEI 7-16 or using a General method not restricted by any specific standards.

Seismic Load according to ASCE/SEI 7-16

First, we identify the map data properties for our building. We can obtain the seismic map data from free tools such as the "ASCE 7 Hazard Tool" at https://asce7hazardtool.online.
Here we can find the values for:

SS: the Long-period transition period, the spectral response acceleration parameter at short periods:
S1: the spectral response acceleration parameter at a period of 1s:

R is the response modification factor from ASCE/SEI 7-16 Table 12.2-1

All fields that require input are white Edit boxes. The fields that are the result of a calculation are shaded. When all input values are entered, we can determine the results by pressing the Calculate button.

Site Class is a classification assigned to a site based on the types of soils present and their engineering properties, as defined in the table 20.3-1 below. The calculations in Analysis3D are limited to the Site classifications A to D.

Table 20.3-1 Site Classification

Site Class	$\bar{v}_s$	$\bar{N}$ or $\bar{N}_{ch}$	$\bar{s}_u$
A. Hard rock	>5,000 ft/s	NA	NA
B. Rock	2,500 to 5,000 ft/s	NA	NA
C. Very dense soil and soft rock	1,200 to 2,500 ft/s	>50 blows/ft	>2,000 lb/ft^2
D. Stiff soil	600 to 1,200 ft/s	15 to 50 blows/ft	1,000 to 2,000 lb/ft^2
E. Soft clay soil	<600 ft/s	<15 blows/ft	<1,000 lb/ft^2
	Any profile with more than 10 ft of soil that has the following characteristics: — Plasticity index $PI > 20$, — Moisture content $w \geq 40\%$, — Undrained shear strength $\bar{s}_u < 500$ lb/ft^2		
F. Soils requiring site response analysis in accordance with Section 21.1	See Section 20.3.1		

Note: For SI: 1 ft = 0.3048 m; 1 ft/s = 0.3048 m/s; 1 lb/ft^2 = 0.0479 kN/m^2.

The Risk Category is found in the table 1.5-1 below.

Table 1.5-1 Risk Category of Buildings and Other Structures for Flood, Wind, Snow, Earthquake, and Ice Loads

Use or Occupancy of Buildings and Structures	Risk Category
Buildings and other structures that represent low risk to human life in the event of failure	I
All buildings and other structures except those listed in Risk Categories I, III, and IV	II
Buildings and other structures, the failure of which could pose a substantial risk to human life	III
Buildings and other structures, not included in Risk Category IV, with potential to cause a substantial economic impact and/or mass disruption of day-to-day civilian life in the event of failure	
Buildings and other structures not included in Risk Category IV (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, hazardous waste, or explosives) containing toxic or explosive substances where the quantity of the material exceeds a threshold quantity established by the Authority Having Jurisdiction and is sufficient to pose a threat to the public if released[a]	
Buildings and other structures designated as essential facilities	IV
Buildings and other structures, the failure of which could pose a substantial hazard to the community	
Buildings and other structures (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, or hazardous waste) containing sufficient quantities of highly toxic substances where the quantity of the material exceeds a threshold quantity established by the Authority Having Jurisdiction and is sufficient to pose a threat to the public if released[a]	
Buildings and other structures required to maintain the functionality of other Risk Category IV structures	

The structure type is determined from the table 12.8-2 below.

Table 12.8-2 Values of Approximate Period Parameters C_t and x

Structure Type	C_t	x
Moment-resisting frame systems in which the frames resist 100% of the required seismic force and are not enclosed or adjoined by components that are more rigid and will prevent the frames from deflecting where subjected to seismic forces:		
Steel moment-resisting frames	0.028 (0.0724)[a]	0.8
Concrete moment-resisting frames	0.016 (0.0466)[a]	0.9
Steel eccentrically braced frames in accordance with Table 12.2-1 lines B1 or D1	0.03 (0.0731)[a]	0.75
Steel buckling-restrained braced frames	0.03 (0.0731)[a]	0.75
All other structural systems	0.02 (0.0488)[a]	0.75

[a]Metric equivalents are shown in parentheses.

The Design earthquake spectral response acceleration parameters at short periods, S_{DS}, and at 1-s periods, S_{D1}, are determined from the equations:

SDS = 2/3 S_{MS}
SD1 = 2/3 S_{M1}

Where the MCER spectral response acceleration parameters for short periods (S_{MS}) and at 1 s (S_{M1}), adjusted for site class effects, are determined by equations

S_{MS} = F_a SS
S_{M1} = F_v S1

With Fa and Fv from the tables 11.4-1 and 11.4-2 below

Table 11.4-1 Short-Period Site Coefficient, F_a

	Mapped Risk-Targeted Maximum Considered Earthquake (MCE_R) Spectral Response Acceleration Parameter at Short Period					
Site Class	$S_S \leq 0.25$	$S_S = 0.5$	$S_S = 0.75$	$S_S = 1.0$	$S_S = 1.25$	$S_S \geq 1.5$
A	0.8	0.8	0.8	0.8	0.8	0.8
B	0.9	0.9	0.9	0.9	0.9	0.9
C	1.3	1.3	1.2	1.2	1.2	1.2
D	1.6	1.4	1.2	1.1	1.0	1.0

Table 11.4-2 Long-Period Site Coefficient, F_v

Site Class	Mapped Risk-Targeted Maximum Considered Earthquake (MCE_R) Spectral Response Acceleration Parameter at 1-s Period					
	$S_1 \le 0.1$	$S_1 = 0.2$	$S_1 = 0.3$	$S_1 = 0.4$	$S_1 = 0.5$	$S_1 \ge 0.6$
A	0.8	0.8	0.8	0.8	0.8	0.8
B	0.8	0.8	0.8	0.8	0.8	0.8
C	1.5	1.5	1.5	1.5	1.5	1.4
D	2.4	2.2[a]	2.0[a]	1.9[a]	1.8[a]	1.7[a]

The Risk Category is calculated from the tables 11.6-1 and 11.6-2 below. But for Risk Category I, II, or III structures located where the mapped spectral response acceleration parameter at 1-s period, S1, is greater than or equal to 0.75 shall be assigned to Seismic Design Category E. Risk Category IV structures located where the mapped spectral response acceleration parameter at 1-s period, S1, is greater than or equal to 0.75 shall be assigned to Seismic Design Category F

TABLE 11.6-1 Seismic Design Category Based on Short-Period Response Acceleration Parameter

Value of S_{DS}	Risk Category	
	I or II or III	IV
$S_{DS} < 0.167$	A	A
$0.167 \le S_{DS} < 0.33$	B	C
$0.33 \le S_{DS} < 0.50$	C	D
$0.50 \le S_{DS}$	D	D

TABLE 11.6-2 Seismic Design Category Based on 1-s Period Response Acceleration Parameter

Value of S_{D1}	Risk Category	
	I or II or III	IV
$S_{D1} < 0.067$	A	A
$0.067 \le S_{D1} < 0.133$	B	C
$0.133 \le S_{D1} < 0.20$	C	D
$0.20 \le S_{D1}$	D	D

The approximate fundamental period (T), in seconds, shall be determined from the following equation:

$$T = C_t h_n^x$$

where h_n is the structural height
and the coefficients Ct and x are determined from Table 12.8-2 below.

Table 12.8-2 Values of Approximate Period Parameters C_t and x

Structure Type	C_t	x
Moment-resisting frame systems in which the frames resist 100% of the required seismic force and are not enclosed or adjoined by components that are more rigid and will prevent the frames from deflecting where subjected to seismic forces:		
Steel moment-resisting frames	0.028 (0.0724)[a]	0.8
Concrete moment-resisting frames	0.016 (0.0466)[a]	0.9
Steel eccentrically braced frames in accordance with Table 12.2-1 lines B1 or D1	0.03 (0.0731)[a]	0.75
Steel buckling-restrained braced frames	0.03 (0.0731)[a]	0.75
All other structural systems	0.02 (0.0488)[a]	0.75

[a]Metric equivalents are shown in parentheses.

The effective seismic weight, W, of a structure shall include the dead load, above the base and other loads above the base as listed below:
1. In areas used for storage, a minimum of 25% of the floor live load shall be included.
EXCEPTIONS:
a. Where the inclusion of storage loads adds no more than 5% to the effective seismic weight at that level, it need not be included in the effective seismic weight.
b. Floor live load in public garages and open parking structures need not be included.
2. Where provision for partitions is required by Section 4.3.2 in the floor load design, the actual partition weight or a minimum weight of 10 psf (0.48 kN⁄m2) of floor area, whichever is greater.
3. Total operating weight of permanent equipment.
4. Where the flat roof snow load, Pf , exceeds 30 psf (1.44 kN⁄m2), 20% of the uniform design snow load, regardless of actual roof slope.
5. Weight of landscaping and other materials at roof gardens and similar areas

For Analysis3D to be able to calculate the effective weight, the input is required for each Load Case. We have the possibility to apply a combination coefficient to each Load Case (which is fully independent from existing Load case factors). As an example, we would apply a combination coefficient of 1.0 to Dead Loads and a combination coefficient of 0.25 to Live Loads.

The seismic base shear, V, in a given direction shall be determined in accordance with the following equation:

$$V = C_s W$$

where

C_s = the seismic response coefficient
W = the effective seismic weight.

The seismic response coefficient, C_s, is determined in accordance with equation

$$C_s = S_{DS} / (R/I_e)$$

where

I_e = the Importance Factor determined in accordance with Table 1.5-2 below.
R is the response modification factor from ASCE/SEI 7-16 Table 12.2-1

Table 1.5-2 Importance Factors by Risk Category of Buildings and Other Structures for Snow, Ice, and Earthquake Loads

Risk Category from Table 1.5-1	Snow Importance Factor, I_s	Ice Importance Factor—Thickness, I_i	Ice Importance Factor—Wind, I_w	Seismic Importance Factor, I_e
I	0.80	0.80	1.00	1.00
II	1.00	1.00	1.00	1.00
III	1.10	1.15	1.00	1.25
IV	1.20	1.25	1.00	1.50

The value of Cs shall not exceed the following:

for $T \le T_L$

$$C_s = \frac{S_{D1}}{T\left(\frac{R}{I_e}\right)}$$

for $T > T_L$

$$C_s = \frac{S_{D1} T_L}{T^2\left(\frac{R}{I_e}\right)}$$

C_s shall not be less than

$$C_s = 0.044 S_{DS} I_e \ge 0.01$$

When the value of the seismic base shear is calculated. We can distribute the base shear over the different floors.
The lateral seismic force induced at any level is determined from the following equations:

$$F_x = C_{vx} V$$

$$C_{vx} = \frac{w_x h_x^k}{\sum_{i=1}^{n} w_i h_i^k}$$

Where
C_{vx} = vertical distribution factor;
V = total design lateral force or shear at the base of the structure
wi and wx = portion of the total effective seismic weight of the structure (W) located or assigned to level i or x; hi and
hx = height from the base to level i or x

k = an exponent related to the structure period as follows:

- for structures that have a period of 0.5 s or less, k = 1;
- for structures that have a period of 2.5 s or more, k = 2; and
- for structures that have a period between 0.5 and 2.5 s, k shall be 2 or shall be determined by linear interpolation between 1 and 2

We can use the Apply Seismic Load button to distribute the Seismic base shear accordingly over the different levels of the structure. The factors Ex and Ez are the multipliers used for the Seismic Loads for each direction.

Seismic Load according to Eurocode EN 1998:2004

Analysis3D applies the Lateral force method of analysis. This type of analysis may be applied to buildings whose response is not significantly affected by contributions from modes of vibration higher than the fundamental mode in each principal direction. This requirement is deemed to be satisfied in buildings which fulfil both of the two following conditions:

a) they have fundamental periods of vibration T1 in the two main directions which are smaller than the following values $T_1 \leq \begin{cases} 4 \cdot T_C \\ 2,0\ \text{s} \end{cases}$

b) they meet the criteria for regularity in elevation given by EC8 section 4.2.3.3.

All fields that require input are white Edit boxes. The fields that are the result of a calculation are shaded. When all input values are entered, we can determine the results by pressing the Calculate button.

The ground type is determined from table 3.1 below.

Table 3.1: Ground types

Ground type	Description of stratigraphic profile	Parameters		
		$v_{s,30}$ (m/s)	N_{SPT} (blows/30cm)	c_u (kPa)
A	Rock or other rock-like geological formation, including at most 5 m of weaker material at the surface.	> 800	_	_
B	Deposits of very dense sand, gravel, or very stiff clay, at least several tens of metres in thickness, characterised by a gradual increase of mechanical properties with depth.	360 – 800	> 50	> 250
C	Deep deposits of dense or medium-dense sand, gravel or stiff clay with thickness from several tens to many hundreds of metres.	180 – 360	15 - 50	70 - 250
D	Deposits of loose-to-medium cohesionless soil (with or without some soft cohesive layers), or of predominantly soft-to-firm cohesive soil.	< 180	< 15	< 70
E	A soil profile consisting of a surface alluvium layer with v_s values of type C or D and thickness varying between about 5 m and 20 m, underlain by stiffer material with v_s > 800 m/s.			

The Shape of the elastic response spectrum: If deep geology is not accounted for, the recommended choice is the use of two of spectra: Type 1 and Type 2. If the earthquakes that contribute most to the seismic hazard defined for the site for the purpose of probabilistic hazard assessment have a surface-wave magnitude, Ms, not greater than 5,5, it is recommended that the Type 2 spectrum is adopted.

T_1 is the fundamental period of vibration of the building for lateral motion in the direction considered. For buildings with heights of up to 40 m the value of T_1 (in s) may be approximated by the following expression:

$$T_1 = C_t \cdot H^{3/4}$$

where C_t is 0,085 for moment resistant space steel frames, 0,075 for moment resistant space concrete frames and for eccentrically braced steel frames and 0,050 for all other structures; H is the height of the building, in m, from the foundation or from the top of a rigid basement.

q is the behaviour factor and is specified by the National Annex. The ranges are given by table 6.1 below.

Table 6.1: Design concepts, structural ductility classes and upper limit reference values of the behaviour factors

Design concept	Structural ductility class	Range of the reference values of the behaviour factor q
Concept a) Low dissipative structural behaviour	DCL (Low)	≤ 1,5 - 2
Concept b) Dissipative structural behaviour	DCM (Medium)	≤ 4 also limited by the values of Table 6.2
	DCH (High)	only limited by the values of Table 6.2

NOTE 1 The value ascribed to the upper limit of q for low dissipative behaviour, within the range of Table 6.1, for use in a country may be found in its National Annex. The recommended value of the upper limit of q for low-dissipative behaviour is 1,5.

NOTE 2 The National Annex of a particular country may give limitations on the choice of the design concept and of the ductility class which are permissible within that country.

Design ground acceleration on type A ground (ag = Y1 . agR); can be found from the National Annex.
agR is the reference peak ground acceleration on type A ground
Y1 is the importance factor. The value of Y1 for importance class II shall be, by definition, equal to 1.0.
NOTE The values to be ascribed to Y1 for use in a country may be found in its National Annex. The values of Y1 may be different for the various seismic zones of the country, depending on the seismic hazard conditions and on public safety considerations. The recommended values of Y1 for importance classes I, III and IV are equal to 0,8, 1,2 and 1,4, respectively.

β is the lower bound factor for the horizontal design spectrum. NOTE The value to be ascribed to β for use in a country can be found in its National Annex. The recommended value for β is 0,2.

Mass of building: the masses to be used in a seismic analysis should be those associated with the load combination: $G + \psi_{E,i} Q$

Where G equals the Dead load and Q the Imposed load. $\psi_{E,i}$ is the combination coefficient for a variable action i, to be used when determining the effects of the design seismic action. This can be set to 0.3 by default. Since Analysis3D cannot determine which Loads are Dead Loads and which are Imposed Loads to be taken as part of the Seismic Load, we need to clearly identify which Loads to account for.

Dead Loads need to be given a combination coefficient of 1 by the user and loads that don't need to be taken into account as Seismic action, need to be given a combination coefficient of 0.

$$\psi_{Ei} = \varphi \cdot \psi_{2i}$$

Table 4.2: Values of φ for calculating ψ_{Ei}

Type of variable action	Storey	φ
Categories A-C*	Roof	1,0
	Storeys with correlated occupancies	0,8
	Independently occupied storeys	0,5
Categories D-F* and Archives		1,0

* Categories as defined in EN 1991-1-1:2002.

For the horizontal components of the seismic action the design spectrum, Sd(T), is defined by the following expressions:

$$0 \le T \le T_B : S_d(T) = a_g \cdot S \cdot \left[\frac{2}{3} + \frac{T}{T_B} \cdot \left(\frac{2,5}{q} - \frac{2}{3} \right) \right]$$

$$T_B \le T \le T_C : S_d(T) = a_g \cdot S \cdot \frac{2,5}{q}$$

$$T_C \le T \le T_D : S_d(T) \begin{cases} = a_g \cdot S \cdot \frac{2,5}{q} \cdot \left[\frac{T_C}{T} \right] \\ \ge \beta \cdot a_g \end{cases}$$

$$T_D \le T : S_d(T) \begin{cases} = a_g \cdot S \cdot \frac{2,5}{q} \cdot \left[\frac{T_C T_D}{T^2} \right] \\ \ge \beta \cdot a_g \end{cases}$$

Where the values of S, T_B(S), T_C(S) and T_D(S) are found in the tables 3.2 and 3.3 below.

Table 3.2: Values of the parameters describing the recommended Type 1 elastic response spectra

Ground type	S	T_B (s)	T_C (s)	T_D (s)
A	1,0	0,15	0,4	2,0
B	1,2	0,15	0,5	2,0
C	1,15	0,20	0,6	2,0
D	1,35	0,20	0,8	2,0
E	1,4	0,15	0,5	2,0

Table 3.3: Values of the parameters describing the recommended Type 2 elastic response spectra

Ground type	S	T_B (s)	T_C (s)	T_D (s)
A	1,0	0,05	0,25	1,2
B	1,35	0,05	0,25	1,2
C	1,5	0,10	0,25	1,2
D	1,8	0,10	0,30	1,2
E	1,6	0,05	0,25	1,2

Base shear force: The seismic base shear force for each horizontal direction in which the building is analyzed, shall be determined using the following expression:

$$F_b = S_d(T_1) \cdot m \cdot \lambda \qquad (4.5)$$

where

$S_d(T_1)$ is the ordinate of the design spectrum (see **3.2.2.5**) at period T_1;

T_1 is the fundamental period of vibration of the building for lateral motion in the direction considered;

m is the total mass of the building, above the foundation or above the top of a rigid basement, computed in accordance with **3.2.4(2)**;

λ is the correction factor, the value of which is equal to: $\lambda = 0,85$ if $T_1 \leq 2\ T_C$ and the building has more than two storeys, or $\lambda = 1,0$ otherwise.

Distribution of the horizontal seismic forces:
The fundamental mode shapes in the horizontal directions of analysis of the building may be approximated by horizontal displacements increasing linearly along the height of the building.
The seismic action effects shall be determined by applying, to the two planar models, horizontal forces Fi to all storeys.

$$F_i = F_b \cdot \frac{z_i \cdot m_i}{\Sigma\, z_j \cdot m_j}$$

Where:

Fi is the horizontal force acting on storey i;

Fb is the seismic base shear;

zj, zj are the heights of masses mi mj above the level of application of the Seismic action (foundation or top of a rigid basement).

Mi,mj are the storey masses.

We can use the Apply Seismic Load button to distribute the Seismic base shear accordingly over the different levels of the structure. The factors Ex and Ez are the multipliers used for the Seismic Loads for each direction.

Seismic Load for general application

Since it is almost impossible to include each and every standard of each and every country, the “General” option minimizes the predetermined calculations and just allow the user to let Analysis3D calculate the Mass of the building depending on the input given by the user (according to the national standard applicable). The

Mass of the building is calculated using the combination coefficients given for each Load Case.

Only 1 Correction factor is applied (as a combination of all factors from the national standard) to calculate the Seismic base shear.

The Apply Seismic Load button will distribute the Seismic base shear over the different levels of the building.

The seismic action effects shall be determined by applying, to the two planar models, horizontal forces Fi to all storeys.

$$F_i = F_b \cdot \frac{z_i \cdot m_i}{\Sigma z_j \cdot m_j}$$

Where:
Fi is the horizontal force acting on storey i;
Fb is the seismic base shear;
zj, zj are the heights of masses mi mj above the level of application of the Seismic action (foundation or top of a rigid basement).
Mi,mj are the storey masses.

We can use the Apply Seismic Load button to distribute the Seismic base shear accordingly over the different levels of the structure. The factors Ex and Ez are the multipliers used for the Seismic Loads for each direction.

4.6. Load Combinations

All Joint Loads and Member Loads have a Load Case number from 1 to 9. This number corresponds to a given Load Case Name and Load Factor. The only purpose of the name is to act as a reminder for the type of Load Case that is used.

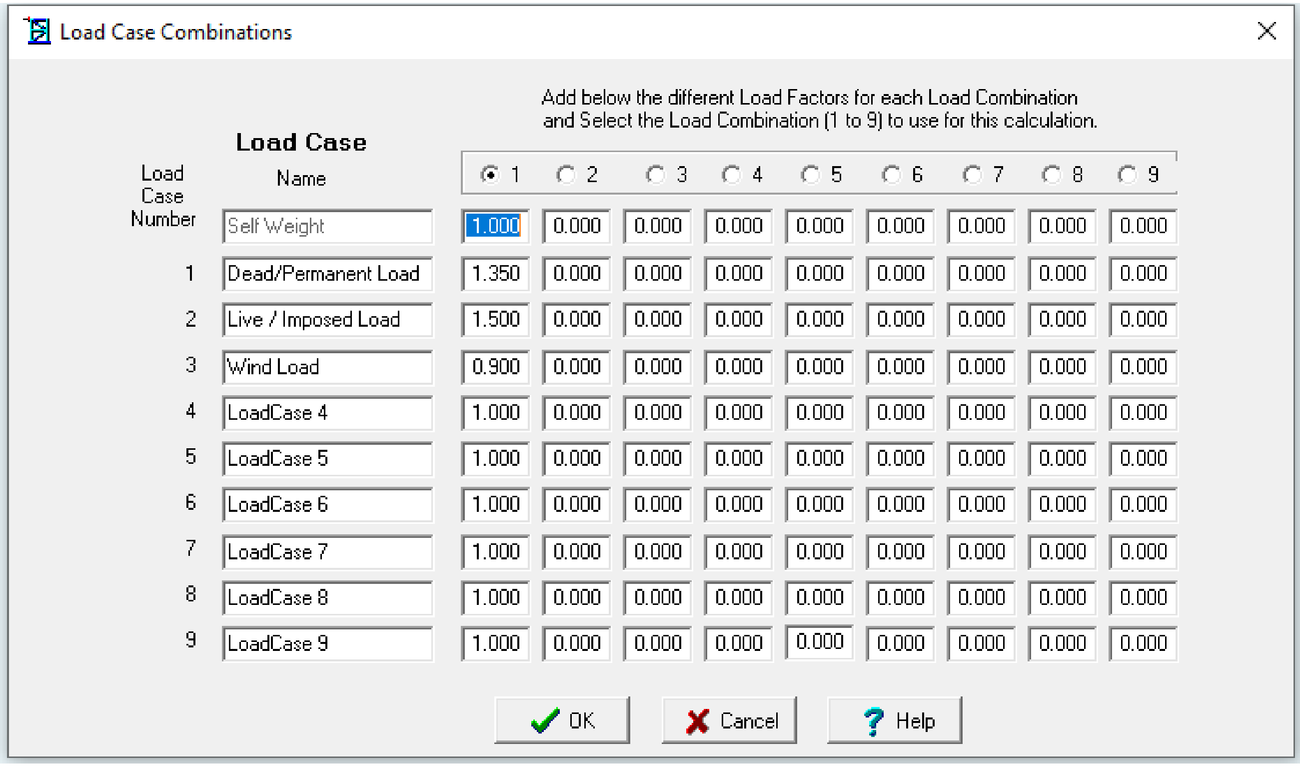
Load Case number 0 is reserved for the self-weight of the structure.

The Load factor is used to calculate the actual importance of a given Load Case. A Load Factor of 2 will double all Loads on the construction with this Load Case number. The total Load that is applied to the structure can be a combination of 9 different load cases, each with a specific importance. A Load factor of 0 eliminates all Loads having that Load Case from the calculation.

In the example below, Load Case number 1 represents the Dead Load or Permanent Load, which is given a Load Factor of 1.35. Load Case 2 represents the Live or Imposed Load and is given a Load Factor of 1.5. The Wind Load is assigned to Load Case 3 with a Load Factor of 0.9.

The columns refer to the different possible Load Combinations (from 1 to 9). Each Load Combination column can have its unique set of Load Factors you want to evaluate. By default, all load factors from Load Combinations 2 to 9 are set to 0.

On top of the table, you can select which column (from 1 to 9), with the corresponding set of Load Factors, that will be used first for the visualization and calculation of the results.



4.7. Calculate or Neglect the Self Weight

When the Loads on the Structure are being calculated, the contribution of the Self Weight of the construction can be neglected or taken into account.

This choice is made by the option **Calculate** or **Neglect Self Weight** in the Structure Menu.

The own Weight is calculated starting from the Mass **M** per meter (or per foot) as indicated in the Cross-Sectional Data.

4.8. Temperature

When the temperature of the Structure changes, there will be equivalent Joint displacements . In case that Support Joints restrict those displacements, the structure will experience a Load.

With the option Temperature from the Structure Menu, you can enter the temperature difference in Kelvin between the temperature at which the construction is build, and the temperature at which you want to calculate the structure.

The Linear coefficient of expansion is given in the material list in 10^{-6} 1/K.

5. The Drawing menu

Redraw

This Menu option redraws the screen. Clicking the Redraw button from the tool bar has the same result.

Drawing Options

Here you can change the options on the drawing: A shortcut key can also toggle most of these options.

Drawing

By selecting these options, Forces, Moments, Support Joints, Joint Displacements and Detailed Member Forces and Stresses will be shown (when calculated). Forces and Moments are represented by default in Red, except for moments in 3D structures that are shown as purple arrows.

When the Detailed Member forces are drawn, it is possible to show or hide the different Detailed Member forces with the mouse. Click with the left mouse button on the name of the selected force in the top left corner of the screen. Click again to redraw.

The factor to magnify the Forces, Joint Displacements and Detailed Member Forces can be changed. By default, the Joint Displacements are ten times exaggerated.

Numbering

By selecting the Joint and/or Member check box, the Joint numbers and/or Member numbers will be shown in the drawing or the name of the cross sections will be displayed.
This option also allows you to visualize the values of the Utilization factor or ratio for each member, based on the Member Design or Detailed Design results. Values of the Utilization ratio >1 are displayed in red.

Viewing Angle

Here you can enter the horizontal and vertical viewing angle. This only makes sense with a 3-dimensional Structure. The horizontal Angle has to be between -360° and +360°, the vertical viewing Angle between -90° and +90°. The default setting is 20* horizontal and vertical.

5.1. Drawing Shortcut Keys

The keyboard

The arrow keys let you rotate the structure:

- **Ctrl arrow** left / right : changes the horizontal angle of view by 30°.
- Ctrl arrow up / down : changes the vertical angle of view by 30°.
- **Alt arrow** left / right : changes the horizontal angle of view by 10°.
- Alt arrow up / down : changes the vertical angle of view by 10°.

To view a certain aspect of the structure:

- Ctrl-F : Front view
- Ctrl-L : Left view
- Ctrl-R : Right view
- Ctrl-T : Top view

Display or hide features:

- Ctrl-J : toggles the display of Joint numbers.
- Ctrl-M : toggles the display of Member numbers or Section names.
- Ctrl-S : toggles the display of the Structure or the Displacements.

Ctrl-N : toggles the display of Detailed Member Forces. Visualize the values of the Member Forces.

The Mouse

Zoom window: Press down the left mouse button and move to select the window to zoom.

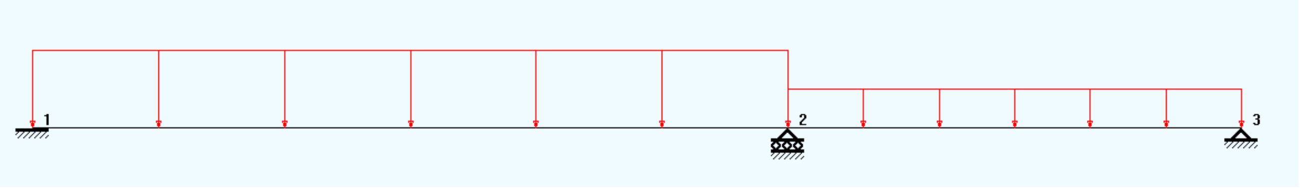
Select a Joint with the mouse by clicking on the Joint Number:

Left mouse button: selects the Joint Number which turns red.
Right mouse button: deselects the Joint.

Double clicking the left mouse button on a Joint Number lets you change or remove the Joint. Double clicking on the blank screen, creates a new Joint.
Selecting two different Joints lets you change, create or remove the Member between those Joints.

6. Calculate

Before *Analysis3D* starts calculating, the structure is tested for stability and consistency. If the Structure is not stable, an error message is generated and the calculation stops.
In that case an error message will show **'The global construction is not stable'.** This means that there is a shortage of Support restraints.
This error message can also occur when the structure is not homogeneous. To demonstrate this with an example of a simple beam with 3 supports: to allow Analysis3D to acknowledge that all 3 supports are connected to the beam, the beam should be split in 2 beams. One beam from support 1 to support 2 and another beam from support 2 to support 3.



When you would only connect support 1 and support 3 with a beam (passing the location of support joint 2). Analysis3D will not know that support 2 is actually connected to that same beam. Analysis3D will see it as a loose support floating in the air at a point that happens to be next to the beam, but unconnected. To make support 2 connected to that same beam you need one beam to go from support 1 to support 2 and a 2nd beam from support 2 to support 3.
The same goes for any connection. Analysis3D requires that each connection is actually connecting all the joints you want to be connected.

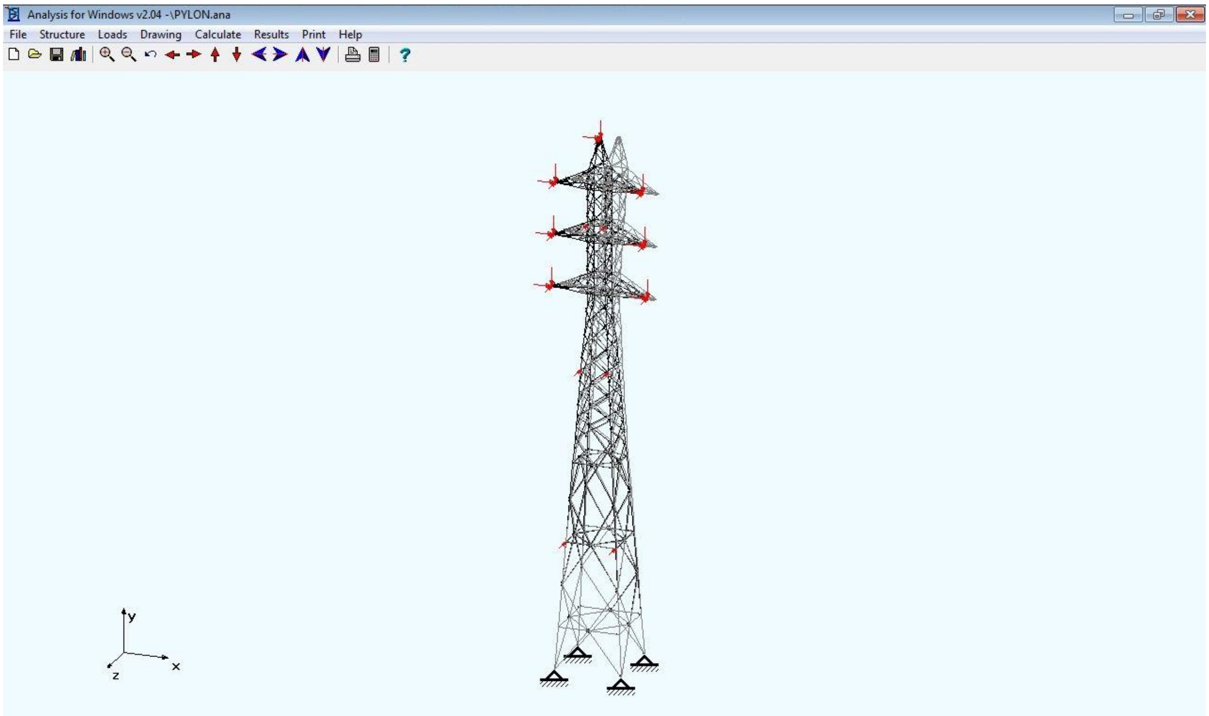
Another typical error message could be **'The Structure is not consistent'**. This occurs when there are duplicate joints with (almost) the same joint coordinates. Please remove the duplicate joints to continue. The structure will also be not consistent when one or more joints is not connected to the rest of the structure.

Nonlinear or 2nd order Calculation

In the standard Calculation *Analysis3D* assumes that the original geometry can be used throughout the analysis. This means that the loads are considered to be fixed in position so that all forces and moments are constant and acting in the same direction as the structure deforms. However, we know that this is actually not the case. When a load is applied to a joint in a real structure, that joint will move and the load will move with it. This movement of the load position can change both the direction at which the load acts with respect to the members connected to the joint, and the moment which the load causes about other points in the structure. If the displacements are large enough, the effect which the load have upon the structure can change significantly.

During Nonlinear or 2nd order Calculation, *Analysis3D* takes into account the deformed structure and incrementally calculates the effect on the structure. Each increment uses the geometry of the structure due to the application of the previous increment. Because of the incremental approach, Nonlinear calculation takes somewhat longer to complete.

Sway structures require 2nd order Calculations because of the magnitude of the sway. *Analysis3D* allows you to check if structures are classified as Sway or Non-Sway.



Very large structures can generate an error message, because there is not enough free memory. To solve this problem:

1. Close all other applications before starting the computation. In some cases, it can be useful to restart Windows.
2. Enlarge the virtual memory.

Analysis3D supports multitasking so that you can even start another copy of the program while the calculation is busy.

The Drawing will automatically be updated by showing the Joint Displacements (by default ten times exaggerated).

Finally, the Results can be consulted.

Stiffness Reduction Factor

Some standards require that the analysis of the structure to determine the required strengths of the components uses reduced stiffnesses. This reduction factor will be applied to all stiffnesses in the structure.

A stiffness reduction factor of 0.8 will reduce the stiffness with 20% and displacements will increase accordingly.

7. Results

7.1. Joint Displacements

The Joint Displacements are presented in a table, and made visible on the screen. The Displacements are by default ten times exaggerated in the drawing. Members that fail the requirements for Member Stress or Buckling are indicated in red. The resulting stresses are displayed in the corresponding list.

Displacements in [mm] or [inch] are expressed in the Global Coordinate System as indicated in the bottom left corner. Thus, a positive translation in the Y direction moves upwards.

The Rotations in [°] are given according the right-hand rule in the Global Coordinate System.

At the bottom right, you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

7.2. Member Forces

The Member Forces in [kN] or [Kips] and the Moments in [kNm] or [Kips-ft] are given according to the Local Coordinate System.

The units are specified by the Structure Menu option Units and Region.

With Trusses the Member Forces on both sides are equal and opposite. So only the Member Force at the end Joint is displayed. This conforms to the definition:

Positive Member Force = Tension
Negative Member Force = Compression

With Frames the Member Forces on both sides can be different. Thus, the Member Forces on both Joints are displayed.

The details of the selected member are displayed on the top of the page, including the weight of the member.
On the top right of the screen, you will find the total weight of the structure.

At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

7.3. Member Stresses

The Member Stresses in [N/mm²] or [Kips/inch²] are presented according to the Local Coordinate System.

The units are specified by the Structure Menu option Units and Region.

With Trusses the Member Stresses on both sides are equal and opposite. So only the Member Stress at the end Joint is displayed. This conforms to the definition:
Positive Member Stress = Tension
Negative Member Stress = Compression

With Frames the Member Stresses on both sides can be different. Thus, the Member Stresses on both Joints are shown.

SAx : the axial stress or strain.

SFy, SFz : the Shear stresses along the local y- respectively z-direction.
The Shear stresses are calculated according SFy = Fy . Sx / (tw . Ix).
This formula is reflecting that the Shear Force is working on the area of the web of the section.

SBy, SBz : the Bending stresses along the local y- respectively z-direction.
Bending Stress is calculated with SBz = Mz ex / Ix

STx : the Torsional stress.
The Torsional Stress is calculated with STx = Mx et / It.

SRes : the Resulting Stress computed according Huber and Hencky (also referred to as Von Mises yield criterion)

SRes = sqrt($SAx^2 + SBy^2 + SBz^2$ - SAx SBy – SBy SBz – SBz SAx + $3SFy^2 + 3SFz^2 + 3STx^2$)

In the case that the Structure is two-dimensional the equation becomes:

SRes = sqrt($SAx^2 + SBz^2$ - SAx SBz + $3SFy^2$)

So SRes is not the real composed Member Stress but the ideal comparative stress that can be compared with Yield Strength (Re) as defined in the material properties.

All members that are not meeting the requirements versus the Yield Strength **Re**, as given in the material data, are highlighted in the list.

At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

7.4. Sway and Natural Frequency

Sway Stability is considered to determine if a structure is sway or non-sway. This depends on the geometry and the load cases under consideration. It is determined and influenced by the of P-delta effect.

Non-sway structures:
- Horizontal loads are carried by the bracing or by horizontal support
- Change of geometry (2nd-order effect) is negligible

Sway structures:
- Horizontal loads are carried by the frame
- Change of geometry (2nd-order effect) is significant

Analysis3D calculates for each story if it is sway or non-sway. In the case the structure is classified as sway, 2^{nd} order analysis is required. *Analysis3D* allows you to calculate 2^{nd} order effects in the case of Sway structures.

In Eurocode 3 the critical Sway ratio is determined as:

$$\alpha_{\text{cr}} = \frac{H_{\text{Ed}}}{V_{\text{Ed}}} \frac{h}{\delta_{\text{h,Ed}}}$$

While ACI 318-19 works with the invers stability index ratio:

$$Q = \frac{\sum P_u \Delta_o}{V_{us} l_c} < 0.05$$

The **Natural Frequency** of the given structure is calculated using the formula below:

$$f = \frac{1}{2\pi} \sqrt{\frac{K_{tot}}{M_{tot}}} \text{ Hz}$$

Where: Ktot represents the total combined stiffness of the building and Mtot equals the total mass.

7.5. Buckling

The calculation of Buckling assumes that all Loads are entered with a Load Factor.

The **Load Factor** is the ratio between the Force used for the calculations and the real Force. This Load Factor is by default set to 1.5. If any other Load Factor has been used, the Load factor should be changed accordingly.

The **Safety Factor** depends upon the conditions. For steel constructions a default Safety Factor of 2.5 is used. Machine parts such as a piston rod need a Safety Factor of 4 to 10.

The resulting **Maximum admissible Buckling Load** depends on the given Load Factor and Safety Factor.

For each compressed Member, the slenderness and the Maximum admissible Buckling Load is calculated, taking into account the Support restraints.

A distinction has to be made dependent on the slenderness of the Member.

Slenderness = Member length / sqrt(I / Ax)
Eulers Limit Slenderness = 3,14 sqrt(E / 0,7 Re) = 111 (for FE360)

Slenderness <= 20: In this area the maximum admissible strain will be reached long before Buckling takes place. A computation on train is sufficient. The maximum admissible Buckling Load will be: Ax . Re.

20 < Slenderness < Eulers Limit Slenderness: This is the plastic area, where the present Stress may not exceed the Yield Point of the material. The maximum admissible Buckling Load follows the straight line of Von Tetmayer.

Slenderness > Eulers Limit Slenderness: This is the elastic area and can be calculated with Euler.
The maximum admissible Buckling Load = $3{,}14^2$ EI / (safety factor . buckling length^2)

At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

Remarks:

In the calculation of the maximum admissible Buckling Load the smallest radius of inertia is used. The program will choose Ix or Iy. In Angle Cross Sections the smallest radius of inertia (along the diagonal) is not available, this will result in a minor aberration.

The maximum limit Stress is set at 0,7 Re. This Stress corresponds to Eulers limit Slenderness. Since this only applies to rolled Sections, the calculations for Buckling only apply to these Sections.

7.6. Reactions

The Reaction Forces are calculated by adding the Member Forces in the Support Joints.

The Forces in [kN] or [Kips] are oriented along the Global Coordinate System as indicated in the bottom left corner of the screen. Positive Reactions in the Y direction are pointed up.

Moments in [kNm] or [Kips-ft] are shown according the right-hand rule in the Global Coordinate System.

The units are specified by the Structure Menu option Units and Region.

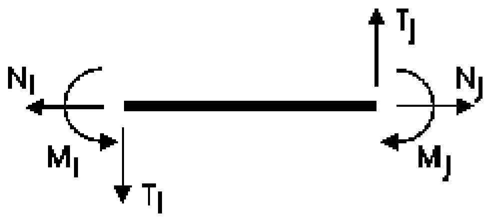
At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

7.7. Detailed Member Forces

The option Detail Member Forces calculates for one Member the values of:
Axial (Normal) Forces (N)
Shear Forces (Ty and Tz)
Torsion (Mx)
Bending Moments (My and Mz)
Deflection (fy and fz)

The sign definition for the member Forces is as follows:
A Normal Force is positive, when it induces Stress in the considered Section.
A Shear Force is positive, when it rotates the released part to the left.
A Bending Moment is positive, when she gives Stress in the part above the center of the Section.
Deflection is positive when it coincides with the positive Local Coordinate System.

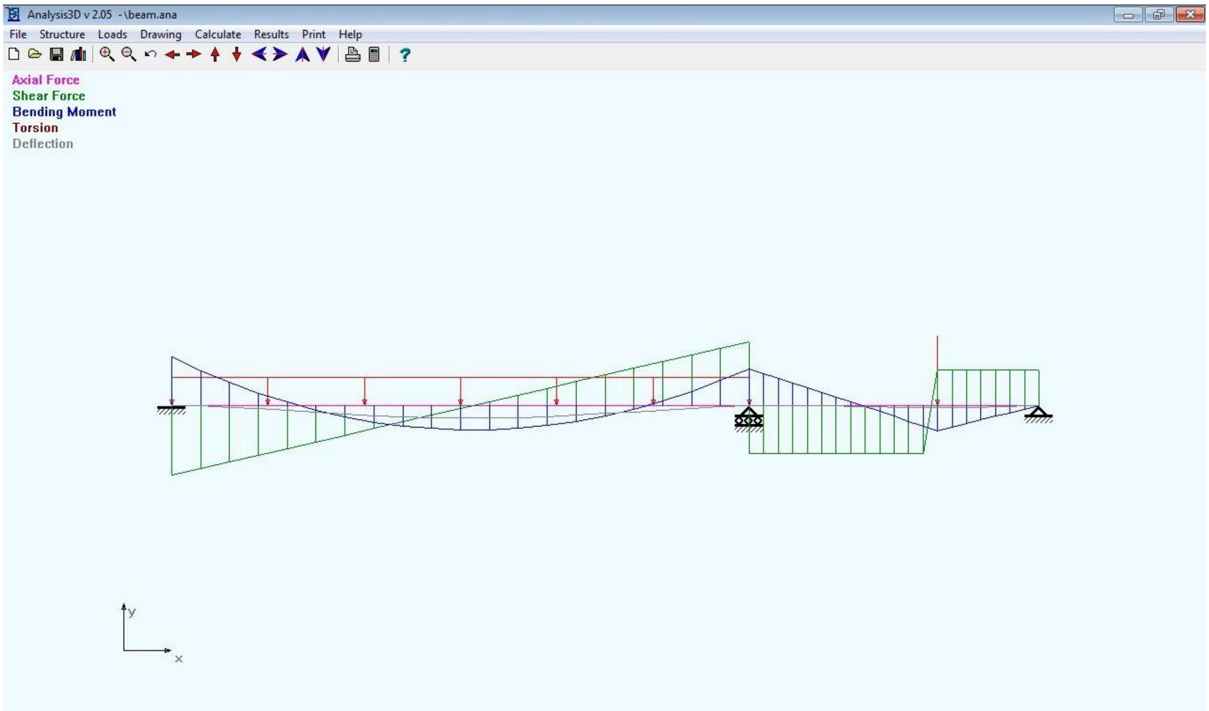
Thus, all Forces and Moments in the following drawing are positive.



The number of **divisions** of the Member can be set.

The calculation shows the detailed member Forces in a table. The maximum values are highlighted.

All Members selected from the list to the right will be drawn together with their detailed member Forces.



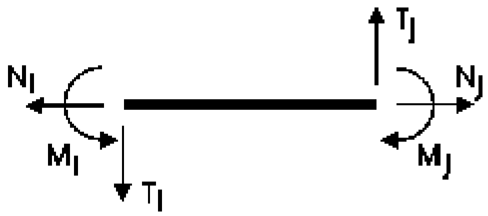
At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

7.8. Detailed member Stresses

The option Detail Member Stresses calculates for one Member the course of:
Normal (Axial) Stress (SAx),
Shear Stress (SFy en SFz),
Torsion Stress (STx)
Bending Stress (SBy en SBz),
Deflection (fy)
Resulting Comparative Stress (SRes)

SRes is not the real composed Member Stress but the Ideal Resulting Comparative Stress that can be compared with Re.
For the calculation of the Resulting Comparative Stress see Member Stresses

The sign definition is equal as for the Detail Member Forces:



The number of **divisions** of the Member can be changed.

The maximum value for each stress is highlighted, also any stresses that are not meeting the requirements versus the Yield Strength **Re**, as given in the material data, are highlighted in the list.

All Members selected from the list to the right will be drawn together with their detailed Stresses.

At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.

7.9. Member Design

This is a powerful tool to optimize your construction in 2 ways.
If a given member is not strong enough for the selected calculation method, Analysis3D will propose the smallest cross-section from the list with Cross-sections. These members are highlighted (selected) in the list with members.
If a given member is over designed, then Analysis3D will propose you the cross-section with minimum area Ax that is acceptable for the given load, to minimize cost.

Each cross-section can be checked in the following different ways as selected in the Units and Region menu option:
- using the AISC-ASD standard
- using the AISC-LRFD standard
- using the BS 5950 standard
- using the CISC 94 standard
- using the Eurocode 3 standard

The maximum utilization factor or utilization ratio is specified which should be ≤ 1. The utilization ratio is a measure for how close a section is to using its maximal section capacity and is obtained by the ratio between the applied loads and the loads at the design resistance.

You can choose to use all or just partly incorporate the proposed changes. After the changes are made, you need to recalculate the structure to see the effect. You will find that because of the changed sections, the forces in the structure are divided differently. This means that other sections are no longer ideally dimensioned. You will find that you may need a few iterations to come to the best solution.

Change button: Just change the one selected member after the button with the proposed Section. Before pressing this button, you can override the proposals by manually changing the section or angle.
Change All button: Change all members with the proposed sections.
Only Selected button: Change the Section of the selected (highlighted) members. These members are already selected for you and contain only those members that are not strong enough. Of course, you can always change any selection you want.

Based on our experience this is the best process to come to a structure with optimal sections.
1. Start with the first Load case you have to evaluate. In this process you can use the **Change All** button to find the smallest sections to minimize cost.
2. As you will move on by checking the other Load cases for the same construction, you then can only change those sections that are not strong enough. (Press the **Only Selected** button).
Each change of a cross-section will affect the distribution of the forces in the total structure. This means that the total structure needs to be recalculated for validation of the final result.

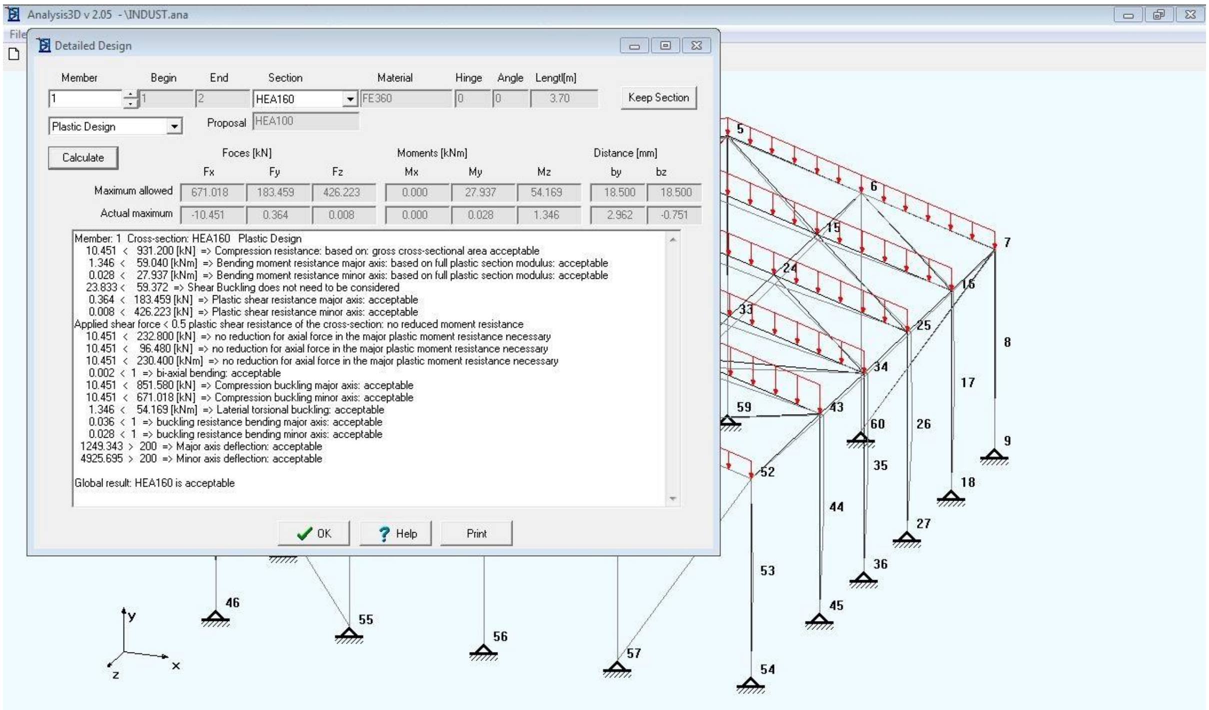
At the bottom right you can select from the drop-down list, which Load Combination you want to use for calculating the results.

7.10. Detailed Member Design

Each cross-section can be checked in the following different ways as selected in the Units and Region menu option:
- using the AISC ASD standard
- using the AISC LRFD standard
- using the BS 5950 standard
- using the CISC 94 standard
- using the Eurocode 3 standard

For each method, Analysis3D will not only test the given Cross-sections, but will also calculate optimized sections for the defined loads.

At the bottom right you can select from the drop-down list, which Load Combination you want to use for visualizing the results.



7.11. AISC-ASD

Design for Strength Using Allowable Strength Design according to ANSI/AISC 360-16

Reference: Specification for Structural Steel Buildings, July 7, 2016 AMERICAN INSTITUTE OF STEEL CONSTRUCTION

Design according to the provisions for allowable strength design (ASD) satisfies the requirements of this Specification when the allowable strength of each structural component equals or exceeds the required strength determined on the basis of the ASD load combinations.

Design shall be performed in accordance with the equation Ra ≤ Rn Ω
Where:

Ra = required strength using ASD load combinations
Rn = nominal strength
Ω = safety factor
Rn/Ω = allowable strength

The nominal strength, Rn, and the safety factor, Ω, for the applicable limit states are specified below:

TENSILE STRENGTH: $\Omega_t = 1.67$
COMPRESSION: $\Omega_c = 1.67$
FLEXURE: $\Omega_b = 1.67$
SHEAR: $\Omega_v = 1.67$
TORSION: $\Omega_T = 1.67$

Allowable Strength Design (ASD) requires 2nd order effects (P-Δ effect) to be included. This is achieved by performing 2nd order analysis or by using a Moment magnification factor. *Analysis3D* assumes ASD is performed on the results of a Nonlinear or 2nd order Calculation before the ASD analysis is done

All formulas and calculations are detailed in the LRFD design.

7.12. AISC-LRFD

Design for Strength Using Load and Resistance Factor Design (LRFD) according ANSI/AISC 360-16.

Reference: Specification for Structural Steel Buildings, July 7, 2016 AMERICAN INSTITUTE OF STEEL CONSTRUCTION

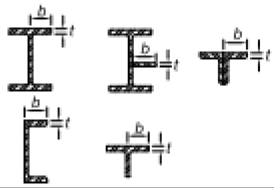
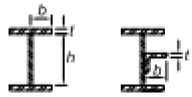
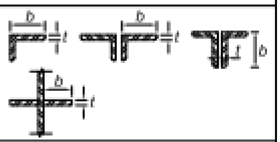
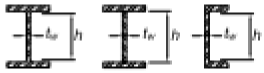
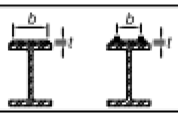
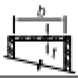
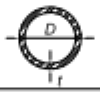
Design according to the provisions for load and resistance factor design (LRFD) satisfies the requirements of this Specification when the design strength of each structural component equals or exceeds the required strength determined on the basis of the LRFD load combinations.

LRFD requires 2nd order effects (P-Δ effect) to be included. Analysis3D assumes LRFD will by calculated using 2nd order Analysis results and that the second-order analysis be carried out under LRFD load combinations.

Classification of Sections for Local Buckling

For members subject to axial compression, sections are classified as nonslender-element or slender-element sections. For a nonslender-element section, the width-to-thickness ratios of its compression elements shall not exceed λr from Table B4.1a. If the width-to-thickness ratio of any compression element exceeds λr, the section is a slender-element section. For members subject to flexure, sections are classified as compact, noncompact or slender-element sections. For a section to qualify as compact, its flanges must be continuously connected to the web or webs, and the width-to-thickness ratios of its compression elements shall not exceed the limiting width-to-thickness ratios, λp, from Table B4.1b. If the width-to-thickness ratio of one or more compression elements exceeds λp, but does not exceed λr from Table B4.1b, the section is noncompact. If the width-to-thickness ratio of any compression element exceeds λr, the section is a slender-element section

TABLE B4.1a
Width-to-Thickness Ratios: Compression Elements
Members Subject to Axial Compression

	Case	Description of Element	Width-to-Thickness Ratio	Limiting Width-to-Thickness Ratio λ_r (nonslender/slender)	Examples
Unstiffened Elements	1	Flanges of rolled I-shaped sections, plates projecting from rolled I-shaped sections, outstanding legs of pairs of angles connected with continuous contact, flanges of channels, and flanges of tees	b/t	$0.56\sqrt{\frac{E}{F_y}}$	
	2	Flanges of built-up I-shaped sections and plates or angle legs projecting from built-up I-shaped sections	b/t	$0.64\sqrt{\frac{k_cE}{F_y}}$ [a]	
	3	Legs of single angles, legs of double angles with separators, and all other unstiffened elements	b/t	$0.45\sqrt{\frac{E}{F_y}}$	
	4	Stems of tees	d/t	$0.75\sqrt{\frac{E}{F_y}}$	
Stiffened Elements	5	Webs of doubly symmetric rolled and built-up I-shaped sections and channels	h/t_w	$1.49\sqrt{\frac{E}{F_y}}$	
	6	Walls of rectangular HSS	b/t	$1.40\sqrt{\frac{E}{F_y}}$	
	7	Flange cover plates and diaphragm plates between lines of fasteners or welds	b/t	$1.40\sqrt{\frac{E}{F_y}}$	
	8	All other stiffened elements	b/t	$1.49\sqrt{\frac{E}{F_y}}$	
	9	Round HSS	D/t	$0.11\frac{E}{F_y}$	

[a] $k_c = 4/\sqrt{h/t_w}$, but shall not be taken less than 0.35 nor greater than 0.76 for calculation purposes.

TABLE B4.1b
Width-to-Thickness Ratios: Compression Elements Members Subject to Flexure

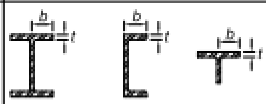
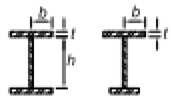
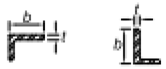
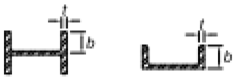
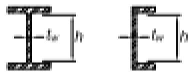
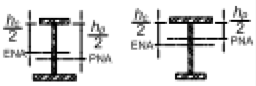
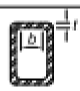
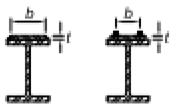
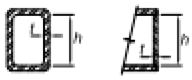
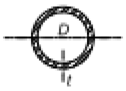
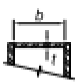
	Case	Description of Element	Width-to-Thickness Ratio	Limiting Width-to-Thickness Ratio: λ_p (compact/noncompact)	Limiting Width-to-Thickness Ratio: λ_r (noncompact/slender)	Examples
Unstiffened Elements	10	Flanges of rolled I-shaped sections, channels, and tees	b/t	$0.38\sqrt{\frac{E}{F_y}}$	$1.0\sqrt{\frac{E}{F_y}}$	
Unstiffened Elements	11	Flanges of doubly and singly symmetric I-shaped built-up sections	b/t	$0.38\sqrt{\frac{E}{F_y}}$	$0.95\sqrt{\frac{k_cE}{F_L}}$ [a] [b]	
Unstiffened Elements	12	Legs of single angles	b/t	$0.54\sqrt{\frac{E}{F_y}}$	$0.91\sqrt{\frac{E}{F_y}}$	
Unstiffened Elements	13	Flanges of all I-shaped sections and channels in flexure about the minor axis	b/t	$0.38\sqrt{\frac{E}{F_y}}$	$1.0\sqrt{\frac{E}{F_y}}$	
Unstiffened Elements	14	Stems of tees	d/t	$0.84\sqrt{\frac{E}{F_y}}$	$1.52\sqrt{\frac{E}{F_y}}$	

TABLE B4.1b (continued)
Width-to-Thickness Ratios: Compression Elements Members Subject to Flexure

	Case	Description of Element	Width-to-Thickness Ratio	Limiting Width-to-Thickness Ratio: λ_p (compact/noncompact)	Limiting Width-to-Thickness Ratio: λ_r (noncompact/slender)	Examples
Stiffened Elements	15	Webs of doubly symmetric I-shaped sections and channels	h/t_w	$3.76\sqrt{\frac{E}{F_y}}$	$5.70\sqrt{\frac{E}{F_y}}$	
Stiffened Elements	16	Webs of singly symmetric I-shaped sections	h_c/t_w	$\dfrac{\frac{h_c}{h_p}\sqrt{\frac{E}{F_y}}}{\left(0.54\frac{M_p}{M_y}-0.09\right)^2} \le \lambda_r$ [c]	$5.70\sqrt{\frac{E}{F_y}}$	
Stiffened Elements	17	Flanges of rectangular HSS	b/t	$1.12\sqrt{\frac{E}{F_y}}$	$1.40\sqrt{\frac{E}{F_y}}$	
Stiffened Elements	18	Flange cover plates and diaphragm plates between lines of fasteners or welds	b/t	$1.12\sqrt{\frac{E}{F_y}}$	$1.40\sqrt{\frac{E}{F_y}}$	
Stiffened Elements	19	Webs of rectangular HSS and box sections	h/t	$2.42\sqrt{\frac{E}{F_y}}$	$5.70\sqrt{\frac{E}{F_y}}$	
Stiffened Elements	20	Round HSS	D/t	$0.07\frac{E}{F_y}$	$0.31\frac{E}{F_y}$	
Stiffened Elements	21	Flanges of box sections	b/t	$1.12\sqrt{\frac{E}{F_y}}$	$1.49\sqrt{\frac{E}{F_y}}$	

[a] $k_c = 4/\sqrt{h/t_w}$, shall not be taken less than 0.35 nor greater than 0.76 for calculation purposes.
[b] $F_L = 0.7F_y$ for slender web I-shaped members and major-axis bending of compact and noncompact web built-up I-shaped members with $S_{xt}/S_{xc} \ge 0.7$; $F_L = F_yS_{xt}/S_{xc} \ge 0.5F_y$ for major-axis bending of compact and noncompact web built-up I-shaped members with $S_{xt}/S_{xc} < 0.7$, where S_{xc}, S_{xt} = elastic section modulus referred to compression and tension flanges, respectively, in.^{3} (mm^3).
[c] M_y is the moment at yielding of the extreme fiber. $M_p = F_yZ_x$, plastic bending moment, kip-in. (N-mm), where Z_x = plastic section modulus taken about x-axis, in.^{3} (mm^3).
E = modulus of elasticity of steel = 29,000 ksi (200 000 MPa)
F_y = specified minimum yield stress, ksi (MPa)
ENA = elastic neutral axis
PNA = plastic neutral axis

Design shall be performed in accordance with: $R_u \leq \varphi R_n$
where R_u = required strength using LRFD load combinations
R_n = nominal strength
φ = resistance factor
φR_n = design strength

Tension

For members designed on the basis of tension, the slenderness ratio, L/r, preferably should not exceed 300.

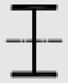
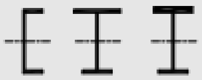
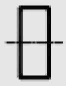
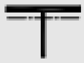
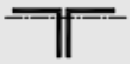
The design tensile strength of tension members is obtained according to the limit states of tensile yielding in the gross section: $P_n = F_y A_g$

The resistance factor φt = 0.90

Compression

For members designed on the basis of compression, the effective slenderness ratio, Lc /r, preferably should not exceed 200.

The resistance factor φc = 0.90

TABLE USER NOTE E1.1 Selection Table for the Application of Chapter E Sections				
Cross Section	Without Slender Elements		With Slender Elements	
	Sections in Chapter E	Limit States	Sections in Chapter E	Limit States
	E3 E4	FB TB	E7	LB FB TB
	E3 E4	FB FTB	E7	LB FB FTB
	E3	FB	E7	LB FB
	E3	FB	E7	LB FB
	E3 E4	FB FTB	E7	LB FB FTB
	E6 E3 E4	FB FTB	E6 E7	LB FB FTB
	E5		E5	
	E3	FB	N/A	N/A
Unsymmetrical shapes other than single angles	E4	FTB	E7	LB FTB

FB = flexural buckling, TB = torsional buckling, FTB = flexural-torsional buckling, LB = local buckling, N/A = not applicable

FLEXURAL BUCKLING OF MEMBERS WITHOUT SLENDER ELEMENTS

The nominal compressive strength, P_n, shall be determined based on the limit state of flexural buckling: $P_n = F_{cr} A_g$

The critical stress, F_{cr}, is determined as follows:

(a) When $\frac{L_c}{r} \le 4.71\sqrt{\frac{E}{F_y}}$ (or $\frac{F_y}{F_e} \le 2.25$)

$$F_{cr} = \left(0.658^{\frac{F_y}{F_e}}\right) F_y$$

(b) When $\frac{L_c}{r} > 4.71\sqrt{\frac{E}{F_y}}$ (or $\frac{F_y}{F_e} > 2.25$)

$$F_{cr} = 0.877 F_e$$

TORSIONAL AND FLEXURAL-TORSIONAL BUCKLING OF SINGLE ANGLES AND MEMBERS WITHOUT SLENDER ELEMENTS

The nominal compressive strength, P_n, shall be determined based on the limit states of torsional and flexural-torsional buckling: $P_n = F_{cr} A_g$

The critical stress, F_{cr}, is determined as above.

For doubly symmetric members twisting about the shear center:

$$F_e = \left(\frac{\pi^2 E C_w}{L_{cz}^{\ 2}} + GJ\right)\frac{1}{I_x + I_y}$$

For singly symmetric members twisting about the shear center where y is the axis of symmetry:

$$F_e = \left(\frac{F_{ey} + F_{ez}}{2H}\right)\left[1 - \sqrt{1 - \frac{4 F_{ey} F_{ez} H}{\left(F_{ey} + F_{ez}\right)^2}}\right]$$

For unsymmetric members twisting about the shear center, Fe is the lowest root of the cubic equation:

$$(F_e - F_{ex})(F_e - F_{ey})(F_e - F_{ez}) - F_e^2(F_e - F_{ey})\left(\frac{x_o}{\bar{r}_o}\right)^2 - F_e^2(F_e - F_{ex})\left(\frac{y_o}{\bar{r}_o}\right)^2 = 0$$

MEMBERS WITH SLENDER ELEMENTS

The nominal compressive strength, P_n, shall be the lowest value based on the applicable limit states of flexural buckling, torsional buckling, and flexural-torsional buckling in interaction with local buckling.

$P_n = F_{cr} A_e$

Slender Element Members Excluding Round HSS: The effective width, be, (for tees, this is de; for webs, this is he) for slender elements is determined as follows:

(a) When $\lambda \le \lambda_r \sqrt{\frac{F_y}{F_{cr}}}$

$$b_e = b$$

(b) When $\lambda > \lambda_r \sqrt{\frac{F_y}{F_{cr}}}$

$$b_e = b\left(1 - c_1\sqrt{\frac{F_{el}}{F_{cr}}}\right)\sqrt{\frac{F_{el}}{F_{cr}}}$$

and $$F_{el} = \left(c_2 \frac{\lambda_r}{\lambda}\right)^2 F_y$$

TABLE E7.1
Effective Width Imperfection Adjustment Factors, c_1 and c_2

Case	Slender Element	c_1	c_2
(a)	Stiffened elements except walls of square and rectangular HSS	0.18	1.31
(b)	Walls of square and rectangular HSS	0.20	1.38
(c)	All other elements	0.22	1.49

Round HSS:

The effective area, A_e, is determined as follows:

(a) When $\frac{D}{t} \leq 0.11\frac{E}{F_y}$

$$A_e = A_g$$

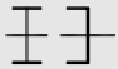
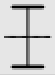
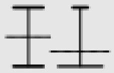
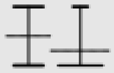
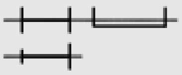
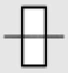
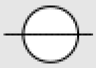
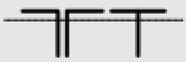
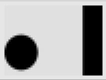
(b) When $0.11\frac{E}{F_y} < \frac{D}{t} < 0.45\frac{E}{F_y}$

$$A_e = \left[\frac{0.038E}{F_y(D/t)} + \frac{2}{3}\right]A_g$$

DESIGN OF MEMBERS FOR FLEXURE

For guidance in determining the appropriate sections of this chapter to apply the table below is used.

TABLE USER NOTE F1.1
Selection Table for the Application of Chapter F Sections

Section in Chapter F	Cross Section	Flange Slenderness	Web Slenderness	Limit States
F2		C	C	Y, LTB
F3		NC, S	C	LTB, FLB
F4		C, NC, S	C, NC	CFY, LTB, FLB, TFY
F5		C, NC, S	S	CFY, LTB, FLB, TFY
F6		C, NC, S	N/A	Y, FLB
F7		C, NC, S	C, NC, S	Y, FLB, WLB, LTB
F8		N/A	N/A	Y, LB
F9		C, NC, S	N/A	Y, LTB, FLB, WLB
F10		N/A	N/A	Y, LTB, LLB
F11		N/A	N/A	Y, LTB
F12	Unsymmetrical shapes, other than single angles	N/A	N/A	All limit states

Y = yielding, CFY = compression flange yielding, LTB = lateral-torsional buckling, FLB = flange local buckling, WLB = web local buckling, TFY = tension flange yielding, LLB = leg local buckling, LB = local buckling, C = compact, NC = noncompact, S = slender, N/A = not applicable

The design flexural strength, $\varphi_b M_n$ shall be determined using $\varphi_b = 0.90$

DOUBLY SYMMETRIC COMPACT I-SHAPED MEMBERS AND CHANNELS BENT ABOUT THEIR MAJOR AXIS

The nominal flexural strength, Mn, shall be the lower value obtained according to the limit states of yielding (plastic moment) and lateral-torsional buckling.

Yielding: $M_n = M_p = F_y Z_x$

Lateral-Torsional Buckling:

- When $L_b \le L_p$, the limit state of lateral-torsional buckling does not apply.
- When $L_p < L_b \le L_r$:

$$M_n = C_b\left[M_p - \left(M_p - 0.7F_yS_x\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le M_p$$

with $$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}}$$ and $$L_r = 1.95 r_{ts} \frac{E}{0.7F_y}\sqrt{\frac{Jc}{S_x h_o} + \sqrt{\left(\frac{Jc}{S_x h_o}\right)^2 + 6.76\left(\frac{0.7F_y}{E}\right)^2}}$$

- When $L_b > L_r$: $M_n = F_{cr} S_x \le M_p$

with $$F_{cr} = \frac{C_b \pi^2 E}{\left(\frac{L_b}{r_{ts}}\right)^2}\sqrt{1 + 0.078\frac{Jc}{S_x h_o}\left(\frac{L_b}{r_{ts}}\right)^2}$$ and $$C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_C}$$

DOUBLY SYMMETRIC I-SHAPED MEMBERS WITH COMPACT WEBS AND NONCOMPACT OR SLENDER FLANGES BENT ABOUT THEIR MAJOR AXIS

The nominal flexural strength, M_n, shall be the lower value obtained according to the limit states of lateral-torsional buckling and compression flange local buckling.

For lateral-torsional buckling: the provisions of the previous paragraph shall apply.

Compression Flange Local Buckling:

(a) For sections with noncompact flanges: $$M_n = M_p - \left(M_p - 0.7F_yS_x\right)\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right)$$

(b) For sections with slender flanges: $$M_n = \frac{0.9Ek_cS_x}{\lambda^2}$$

OTHER I-SHAPED MEMBERS WITH COMPACT OR NONCOMPACT WEBS BENT ABOUT THEIR MAJOR AXIS

The nominal flexural strength, Mn, shall be the lowest value obtained according to the limit states of compression flange yielding, lateral-torsional buckling, compression flange local buckling, and tension flange yield.

Compression Flange Yielding: $M_n = R_{pc} M_{yc}$

with $M_{yc} = F_y S_{xc}$ = yield moment in the compression flange

and $$R_{pc} = \left[\frac{M_p}{M_{yc}} - \left(\frac{M_p}{M_{yc}} - 1\right)\left(\frac{\lambda - \lambda_{pw}}{\lambda_{rw} - \lambda_{pw}}\right)\right] \le \frac{M_p}{M_{yc}}$$

Lateral-Torsional Buckling:

When $L_b \le L_p$, the limit state of lateral-torsional buckling does not apply.

When $L_p < L_b \le L_r$:

$$M_n = C_b\left[R_{pc}M_{yc} - \left(R_{pc}M_{yc} - F_L S_{xc}\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le R_{pc}M_{yc}$$

When $L_b > L_r$: $M_n = F_{cr}\, S_{xc} \le R_{pc}\, M_{yc}$

With $L_p = 1.1 r_t \sqrt{\dfrac{E}{F_y}}$ and $L_r = 1.95 r_t \dfrac{E}{F_L}\sqrt{\dfrac{J}{S_{xc}h_o} + \sqrt{\left(\dfrac{J}{S_{xc}h_o}\right)^2 + 6.76\left(\dfrac{F_L}{E}\right)^2}}$

$r_t = \dfrac{b_{fc}}{\sqrt{12\left(1 + \dfrac{1}{6}a_w\right)}}$ where $a_w = \dfrac{h_c t_w}{b_{fc} t_{fc}}$

and $F_{cr} = \dfrac{C_b \pi^2 E}{\left(\dfrac{L_b}{r_t}\right)^2}\sqrt{1 + 0.078\dfrac{J}{S_{xc}h_o}\left(\dfrac{L_b}{r_t}\right)^2}$

Compression Flange Local Buckling:

For sections with compact flanges, the limit state of local buckling does not apply.

For sections with noncompact flanges: $M_n = R_{pc}M_{yc} - \left(R_{pc}M_{yc} - F_L S_{xc}\right)\left(\dfrac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right)$

For sections with slender flanges: $M_n = \dfrac{0.9 E k_c S_{xc}}{\lambda^2}$

DOUBLY SYMMETRIC AND SINGLY SYMMETRIC I-SHAPED MEMBERS WITH SLENDER WEBS BENT ABOUT THEIR MAJOR AXIS

Compression Flange Yielding: $M_n = R_{pg}\, F_y\, S_{xc}$

where $R_{pg} = 1 - \dfrac{a_w}{1{,}200 + 300 a_w}\left(\dfrac{h_c}{t_w} - 5.7\sqrt{\dfrac{E}{F_y}}\right) \le 1.0$

Lateral-Torsional Buckling: $M_n = R_{pg}\, F_{cr}\, S_{xc}$

When $L_b \le L_p$, the limit state of lateral-torsional buckling does not apply.

When $L_p < L_b \le L_r$: $F_{cr} = C_b\left[F_y - \left(0.3F_y\right)\left(\dfrac{L_b - L_p}{L_r - L_p}\right)\right] \le F_y$

When $L_b > L_r$: $F_{cr} = \dfrac{C_b \pi^2 E}{\left(\dfrac{L_b}{r_t}\right)^2} \le F_y$

Compression Flange Local Buckling: $M_n = R_{pg}\, F_{cr}\, S_{xc}$

For sections with compact flanges, the limit state of compression flange local buckling does not apply.

For sections with noncompact flanges: $$F_{cr} = \left[F_y - (0.3F_y)\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}} \right) \right]$$

For sections with slender flanges: $$F_{cr} = \frac{0.9Ek_c}{\left(\frac{b_f}{2t_f} \right)^2}$$

I-SHAPED MEMBERS AND CHANNELS BENT ABOUT THEIR MINOR AXIS

Yielding: $M_n = M_p = F_y Z_y \leq 1.6 F_y S_y$

Flange Local Buckling:

For sections with compact flanges, the limit state of flange local buckling does not apply.

For sections with noncompact flanges: $$M_n = M_p - (M_p - 0.7F_y S_y)\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}} \right)$$

For sections with slender flanges: $M_n = F_{cr} S_y$ where $$F_{cr} = \frac{0.69E}{\left(\frac{b}{t_f} \right)^2}$$

SQUARE AND RECTANGULAR HSS AND BOX SECTIONS

Yielding: $M_n = M_p = F_y Z$

Flange Local Buckling:

For compact sections, the limit state of flange local buckling does not apply.

For sections with noncompact flanges: $$M_n = M_p - (M_p - F_y S)\left(3.57 \frac{b}{t_f} \sqrt{\frac{F_y}{E}} - 4.0 \right) \leq M_p$$

For sections with slender flanges: $M_n = F_y S_e$

For HSS

$$b_e = 1.92 t_f \sqrt{\frac{E}{F_y}} \left(1 - \frac{0.38}{b / t_f} \sqrt{\frac{E}{F_y}} \right) \leq b$$

For box sections

$$b_e = 1.92 t_f \sqrt{\frac{E}{F_y}} \left(1 - \frac{0.34}{b / t_f} \sqrt{\frac{E}{F_y}} \right) \leq b$$

Web Local Buckling

For compact sections, the limit state of web local buckling does not apply.

For sections with noncompact webs: $$M_n = M_p - (M_p - F_y S)\left(0.305 \frac{h}{t_w} \sqrt{\frac{F_y}{E}} - 0.738 \right) \leq M_p$$

For sections with slender webs:

Compression flange yielding: $M_n = R_{pg} F_y S$

Compression flange local buckling: $M_n = R_{pg} F_{cr} S_{xc}$

and $F_{cr} = \dfrac{0.9Ek_c}{\left(\dfrac{b}{t_f}\right)^2}$

Lateral-Torsional Buckling

When Lb ≤ Lp, the limit state of lateral-torsional buckling does not apply.

When Lp < Lb ≤ Lr: $M_n = C_b\left[M_p - (M_p - 0.7F_y S_x)\left(\dfrac{L_b - L_p}{L_r - L_p}\right)\right] \le M_p$

When Lb > Lr: $M_n = 2EC_b \dfrac{\sqrt{JA_g}}{L_b/r_y} \le M_p$

with $L_p = 0.13Er_y \dfrac{\sqrt{JA_g}}{M_p}$ and $L_r = 2Er_y \dfrac{\sqrt{JA_g}}{0.7F_y S_x}$

ROUND HSS

Yielding: $M_n = M_p = F_y Z$

Local Buckling:

For compact sections, the limit state of flange local buckling does not apply.

For noncompact sections: $M_n = \left[\dfrac{0.021E}{\left(\dfrac{D}{t}\right)} + F_y\right]S$

For sections with slender walls: $M_n = F_{cr} S$ with $F_{cr} = \dfrac{0.33E}{\left(\dfrac{D}{t}\right)}$

TEES AND DOUBLE ANGLES LOADED IN THE PLANE OF SYMMETRY

Yielding:

For tee stems and web legs in tension $M_p = F_y Z_x \le 1.6\, F_y S_x$

For tee stems in compression $Mp = F_y S_x$

Lateral-Torsional Buckling

For stems and web legs in tension

When Lb ≤ Lp, the limit state of lateral-torsional buckling does not apply.

When Lp < Lb ≤ Lr: $M_n = M_p - (M_p - M_y)\left(\dfrac{L_b - L_p}{L_r - L_p}\right)$

When Lb > Lr: $M_n = M_{cr}$ with

$$M_{cr} = \frac{1.95E}{L_b}\sqrt{I_y J}\left(B + \sqrt{1 + B^2}\right)$$

$$B = 2.3\left(\frac{d}{L_b}\right)\sqrt{\frac{I_y}{J}}$$

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}}$$

where $$L_r = 1.95 \left(\frac{E}{F_y}\right) \frac{\sqrt{I_y J}}{S_x} \sqrt{2.36 \left(\frac{F_y}{E}\right) \frac{d S_x}{J} + 1}$$

For stems and web legs in compression anywhere along the unbraced length: $M_n = M_{cr} \le F_y S_x$

with $$B = -2.3\left(\frac{d}{L_b}\right)\sqrt{\frac{I_y}{J}}$$

Flange Local Buckling

For sections with a compact flange in flexural compression, the limit state of flange local buckling does not apply.

For sections with a noncompact flange in flexural compression:

$$M_n = \left[M_p - (M_p - 0.7F_y S_{xc})\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right)\right] \le 1.6 M_y$$

For sections with a slender flange in flexural compression: $$M_n = \frac{0.7 E S_{xc}}{\left(\frac{b_f}{2t_f}\right)^2}$$

Local Buckling of Tee Stems in Flexural Compression: $M_n = F_{cr} S_x$

(1) When $\frac{d}{t_w} \le 0.84\sqrt{\frac{E}{F_y}}$

$$F_{cr} = F_y$$

(2) When $0.84\sqrt{\frac{E}{F_y}} < \frac{d}{t_w} \le 1.52\sqrt{\frac{E}{F_y}}$

$$F_{cr} = \left(1.43 - 0.515\frac{d}{t_w}\sqrt{\frac{F_y}{E}}\right)F_y$$

(3) When $\frac{d}{t_w} > 1.52\sqrt{\frac{E}{F_y}}$

$$F_{cr} = \frac{1.52E}{\left(\frac{d}{t_w}\right)^2}$$

SINGLE ANGLES

Yielding: $M_n = 1.5\ M_y$

Lateral-Torsional Buckling:

(a) When $\frac{M_y}{M_{cr}} \le 1.0$

$$M_n = \left(1.92 - 1.17\sqrt{\frac{M_y}{M_{cr}}}\right)M_y \le 1.5 M_y$$

(b) When $\frac{M_y}{M_{cr}} > 1.0$

$$M_n = \left(0.92 - \frac{0.17 M_{cr}}{M_y}\right)M_{cr}$$

For equal-leg angles with no axial compression: $M_{cr} = \frac{0.58Eb^4tC_b}{L_b^2}\left[\sqrt{1+0.88\left(\frac{L_bt}{b^2}\right)^2}-1\right]$

For other angles: $M_{cr} = \frac{9EAr_ztC_b}{8L_b}\left[\sqrt{1+\left(4.4\frac{\beta_w r_z}{L_bt}\right)^2}+4.4\frac{\beta_w r_z}{L_bt}\right]$

Leg Local Buckling:

For compact sections, the limit state of leg local buckling does not apply.

For sections with noncompact legs: $M_n = F_yS_c\left[2.43-1.72\left(\frac{b}{t}\right)\sqrt{\frac{F_y}{E}}\right]$

For sections with slender legs: $M_n = F_{cr}\,S_c$ with $F_{cr} = \frac{0.71E}{\left(\frac{b}{t}\right)^2}$

RECTANGULAR BARS AND ROUNDS

Yielding: for rounds and for rectangular bars with $\frac{L_bd}{t^2} \le \frac{0.08E}{F_y}$: $M_n = M_p = F_y Z \le 1.6\, F_y S_x$

Lateral-Torsional Buckling:

For rectangular bars with $\frac{L_bd}{t^2} \le \frac{0.08E}{F_y}$ bent about their major axis, the limit state of lateral-torsional buckling does not apply.

For rectangular bars with $\frac{0.08E}{F_y} < \frac{L_bd}{t^2} \le \frac{1.9E}{F_y}$: $M_n = C_b\left[1.52-0.274\left(\frac{L_bd}{t^2}\right)\frac{F_y}{E}\right]M_y \le M_p$

For rectangular bars with $\frac{L_bd}{t^2} > \frac{1.9E}{F_y}$: $M_n = F_{cr}\,S_x \le M_p$ where $F_{cr} = \frac{1.9EC_b}{\frac{L_bd}{t^2}}$

DESIGN OF MEMBERS FOR SHEAR

The design shear strength, $\varphi_v V_n$ shall be determined using $\varphi_v = 0.90$ and the nominal shear strength, V_n, shall be determined according to the sections below.

I-SHAPED MEMBERS AND CHANNELS

Shear Strength of Webs without Tension Field Action:

The nominal shear strength: $V_n = 0.6\, F_y A_w C_{v1}$

For webs of rolled I-shaped members with $h/t_w \le 2.24\sqrt{E/F_y}$ $C_{v1} = 1.0$

For all other I-shaped members and channels:

when $h/t_w \le 1.10\sqrt{k_vE/F_y}$: $C_{v1} = 1.0$

when $h/t_w > 1.10\sqrt{k_vE/F_y}$: $C_{v1} = \frac{1.10\sqrt{k_vE/F_y}}{h/t_w}$

with $k_v = 5.34$

SINGLE ANGLES AND TEES

The nominal shear strength of a single-angle leg or a tee stem is: $V_n = 0.6\ F_y\ b\ t\ C_{v2}$

When $h / t_w \leq 1.10\sqrt{k_v E / F_y}$

$$C_{v2} = 1.0$$

When $1.10\sqrt{k_v E / F_y} < h / t_w \leq 1.37\sqrt{k_v E / F_y}$

$$C_{v2} = \frac{1.10\sqrt{k_v E / F_y}}{h / t_w}$$

When $h / t_w > 1.37\sqrt{k_v E / F_y}$

$$C_{v2} = \frac{1.51 k_v E}{(h / t_w)^2 F_y}$$

RECTANGULAR HSS, BOX SECTIONS, AND OTHER SINGLY AND DOUBLY SYMMETRIC MEMBERS

The nominal shear strength $V_n = 0.6\ F_y\ A_w\ C_{v2}$

ROUND HSS

$V_n = F_{cr}\ A_g / 2$

where

F_{cr} shall be the larger of

$$F_{cr} = \frac{1.60E}{\sqrt{\frac{L_v}{D}}\left(\frac{D}{t}\right)^{\frac{5}{4}}}$$

and

$$F_{cr} = \frac{0.78E}{\left(\frac{D}{t}\right)^{\frac{3}{2}}}$$

but shall not exceed $0.6F_y$

WEAK-AXIS SHEAR IN DOUBLY SYMMETRIC AND SINGLY SYMMETRIC SHAPES

For doubly and singly symmetric shapes loaded in the weak axis without torsion, the nominal shear strength for each shear resisting element is: $V_n = 0.6\ F_y\ b_f\ t_f\ C_{v2}$

DESIGN OF MEMBERS FOR COMBINED FORCES AND TORSION

DOUBLY AND SINGLY SYMMETRIC MEMBERS SUBJECT TO FLEXURE AND AXIAL FORCE

Doubly and Singly Symmetric Members Subject to Flexure and Compression or Tension:

The interaction of flexure and compression or tension in doubly symmetric members and singly symmetric members constrained to bend about a geometric axis (x and/or y) shall be limited by the Equations:

When $\frac{P_r}{P_c} \geq 0.2$

$$\frac{P_r}{P_c} + \frac{8}{9}\left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0$$

When $\frac{P_r}{P_c} < 0.2$

$$\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0$$

Doubly Symmetric Rolled Compact Members Subject to Single-Axis Flexure and Compression

If M_{ry}/M_{cy} <= 0.05 :

For the limit state of in-plane instability, the previous Equations shall be used with Pc taken as the available compressive strength in the plane of bending and Mcx taken as the available flexural strength based on the limit state of yielding.

For the limit state of out-of-plane buckling and lateral-torsional buckling: $\frac{P_r}{P_{cy}}\left(1.5 - 0.5\frac{P_r}{P_{cy}}\right) + \left(\frac{M_{rx}}{C_b M_{cx}}\right)^2 \leq 1.0$

UNSYMMETRIC AND OTHER MEMBERS SUBJECT TO FLEXURE AND AXIAL FORCE

$$\left|\frac{f_{ra}}{F_{ca}} + \frac{f_{rbw}}{F_{cbw}} + \frac{f_{rbz}}{F_{cbz}}\right| \leq 1.0$$

MEMBERS SUBJECT TO TORSION AND COMBINED TORSION, FLEXURE, SHEAR, AND/OR AXIAL FORCE

Round and Rectangular HSS Subject to Torsion

The design torsional strength, $\varphi_T T_n$ for round and rectangular HSS according to the limit states of torsional yielding and torsional buckling shall be determined as follows: $T_n = F_{cr} C$ and $\varphi_T = 0.90$

For round HSS, F_{cr} shall be the larger of: $F_{cr} = \frac{1.23E}{\sqrt{\frac{L}{D}}\left(\frac{D}{t}\right)^{\frac{5}{4}}}$ and $F_{cr} = \frac{0.60E}{\left(\frac{D}{t}\right)^{\frac{3}{2}}}$ but shall not exceed 0.6 F_y

For rectangular HSS:

When $h/t \leq 2.45\sqrt{E/F_y}$

$$F_{cr} = 0.6F_y$$

When $2.45\sqrt{E/F_y} < h/t \leq 3.07\sqrt{E/F_y}$

$$F_{cr} = \frac{0.6F_y\left(2.45\sqrt{E/F_y}\right)}{\left(\frac{h}{t}\right)}$$

When $3.07\sqrt{E/F_y} < h/t \leq 260$

$$F_{cr} = \frac{0.458\pi^2 E}{\left(\frac{h}{t}\right)^2}$$

HSS Subject to Combined Torsion, Shear, Flexure and Axial Force

When Tr exceeds 20% of Tc, the interaction of torsion, shear, flexure and/or axial force shall be limited by $\left(\frac{P_r}{P_c}+\frac{M_r}{M_c}\right)+\left(\frac{V_r}{V_c}+\frac{T_r}{T_c}\right)^2 \leq 1.0$

To summarize the maximum utilization ratio is specified which should be ≤ 1.
The utilization ratio is a measure for how close a section is to using its maximal section capacity and is obtained by the ratio between the applied loads and the loads at the design resistance.

7.13. BS 5950

BS 5950 starts from the classification of sections into 4 categories: Plastic, Compact, Semi-compact or Slender. The classification of the cross-section will determine the way the allowable stress is determined. Analysis3D will evaluate Plastic, Compact, Semi-compact sections. Slender sections are beyond the scope of Analysis3D.

When using the BS 5950 code, Analysis3D design assumes that a 2^{nd} order analysis has already been performed, so that moment magnification factors for the moments causing side-sway can be taken as unity.

Each cross-section is tested for:

- Compression Resistance
- Tension Capacity
- Shear Capacity
- Moment capacity
- Lateral-Torsional Buckling Moment Capacity
- Local Capacity Check
- Overall Buckling Check

7.14. CISC 94

CISC 94 starts for the determination of the nominal strengths for axial compression and flexure, the sections are classified as either Class 1 (Plastic), Class 2 (Compact), Class 3 (Noncompact), or Class 4 (Slender). If a section fails to satisfy the limits for Class 3 sections, the section is classified as Class 4. Currently Analysis3D does not check stresses for Class 4 sections.
When using the CISC 94 code, Analysis3D assumes that a P- analysis has been performed so that moment magnification factors for moments causing sidesway can be taken as unity.
The strength reduction factor is taken as 0.9 (CISC 13.1)
Each cross-section is tested for:

- Compression Strength
- Tension Strength
- Bending Strength
- Shear Strength
- Capacity Ratio for Compressive Axial and Bending Stresses
- Capacity Ratio for Tensile Axial and Bending Stresses

7.15. Eurocode 3

The Eurocode 3 (EN 1993-1-1: 2005) standard first evaluates the section and defines a section class depending on the cross-section properties and the section load combination. The section class will determine the method of analysis.

- Class 1 cross-sections are those which can form a plastic hinge with the rotation capacity required from plastic analysis without reduction of the resistance.
- Class 2 cross-sections arc those which can develop their plastic moment resistance, but have limited rotation capacity because of local buckling.

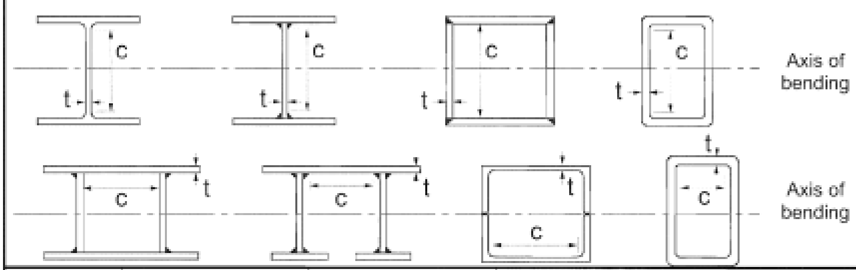
- Class 3 cross-sections are those in which the stress in the extreme compression fiber of the steel member assuming an elastic distribution of stresses can reach the yield strength, but local buckling is liable to prevent development of the plastic moment resistance.
- Class 4 cross-sections are those in which local buckling will occur before the attainment of yield stress in one or more parts of the cross-section.

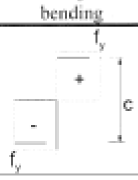
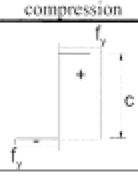
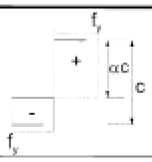
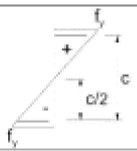
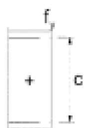
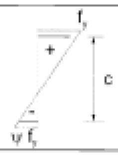
Classification of Sections:

EN 1993-1-1:2005 (E)

Table 5.2 (sheet 1 of 3): Maximum width-to-thickness ratios for compression parts

Internal compression parts



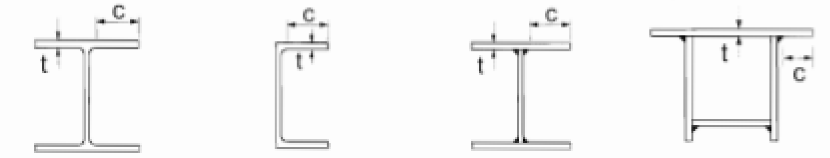
Class	Part subject to bending	Part subject to compression	Part subject to bending and compression
Stress distribution in parts (compression positive)			
1	$c/t \leq 72\varepsilon$	$c/t \leq 33\varepsilon$	when $\alpha > 0{,}5$: $c/t \leq \dfrac{396\varepsilon}{13\alpha - 1}$ when $\alpha \leq 0{,}5$: $c/t \leq \dfrac{36\varepsilon}{\alpha}$
2	$c/t \leq 83\varepsilon$	$c/t \leq 38\varepsilon$	when $\alpha > 0{,}5$: $c/t \leq \dfrac{456\varepsilon}{13\alpha - 1}$ when $\alpha \leq 0{,}5$: $c/t \leq \dfrac{41{,}5\varepsilon}{\alpha}$
Stress distribution in parts (compression positive)			
3	$c/t \leq 124\varepsilon$	$c/t \leq 42\varepsilon$	when $\psi > -1$: $c/t \leq \dfrac{42\varepsilon}{0{,}67 + 0{,}33\psi}$ when $\psi \leq -1$*): $c/t \leq 62\varepsilon(1 - \psi)\sqrt{(-\psi)}$

$\varepsilon = \sqrt{235/f_y}$	f_y	235	275	355	420	460
	ε	1,00	0,92	0,81	0,75	0,71

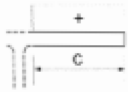
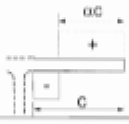
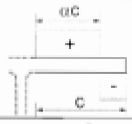
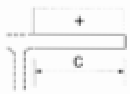
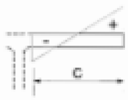
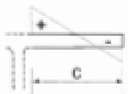
*) $\psi \leq -1$ applies where either the compression stress $\sigma \leq f_y$ or the tensile strain $\varepsilon_y > f_y/E$

Table 5.2 (sheet 2 of 3): Maximum width-to-thickness ratios for compression parts

Outstand flanges



Rolled sections — Welded sections

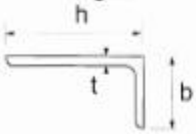
Class	Part subject to compression	Part subject to bending and compression: Tip in compression	Part subject to bending and compression: Tip in tension
Stress distribution in parts (compression positive)			
1	$c / t \leq 9\varepsilon$	$c / t \leq \frac{9\varepsilon}{\alpha}$	$c / t \leq \frac{9\varepsilon}{\alpha\sqrt{\alpha}}$
2	$c / t \leq 10\varepsilon$	$c / t \leq \frac{10\varepsilon}{\alpha}$	$c / t \leq \frac{10\varepsilon}{\alpha\sqrt{\alpha}}$
Stress distribution in parts (compression positive)			
3	$c / t \leq 14\varepsilon$	$c / t \leq 21\varepsilon\sqrt{k_\sigma}$ For k_σ see EN 1993-1-5	

$\varepsilon = \sqrt{235/f_y}$						
	f_y	235	275	355	420	460
	ε	1,00	0,92	0,81	0,75	0,71

Table 5.2 (sheet 3 of 3): Maximum width-to-thickness ratios for compression parts

Angles

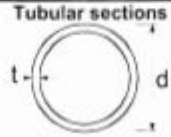
Refer also to "Outstand flanges" (see sheet 2 of 3)



Does not apply to angles in continuous contact with other components

Class	Section in compression
Stress distribution across section (compression positive)	
3	$h / t \leq 15\varepsilon$ and $\frac{b+h}{2t} \leq 11{,}5\varepsilon$

Tubular sections



Class	Section in bending and/or compression
1	$d / t \leq 50\varepsilon^2$
2	$d / t \leq 70\varepsilon^2$
3	$d / t \leq 90\varepsilon^2$ NOTE For $d / t > 90\varepsilon^2$ see EN 1993-1-6.

$\varepsilon = \sqrt{235/f_y}$						
	f_y	235	275	355	420	460
	ε	1,00	0,92	0,81	0,75	0,71
	ε^2	1,00	0,85	0,66	0,56	0,51

A cross-section is classified according to the highest (least favorable) class of its compression parts. Class 4 cross-sections are beyond the scope of Analysis3D.

When using Eurocode 3, Analysis3D design assumes that a P- analysis or 2nd order analysis has been performed so that moment magnification factors for moments causing sidesway can be taken as unity.
No allowance is made for fastener holes. This is outside of the scope of Analysis3D and requires separate investigation.

The nominal strengths in compression, tension, bending, and shear are computed for Class 1, 2, and 3 sections according to the following subsections. The material partial safety factors used by the program are:

$$\gamma_{M0} = 1.1,$$
$$\gamma_{M1} = 1.1.$$

Tension: The design value of the tension force N_{Ed} at each cross section shall satisfy:

$$\frac{N_{Ed}}{N_{t,Rd}} \leq 1{,}0$$

and the design expression for yielding of the gross cross-section (plastic resistance) is given as:

$$N_{pl,Rd} = \frac{A f_y}{\gamma_{M0}}$$

Compression: The design value of the compression force N_{Ed} at each cross-section shall satisfy:

$$\frac{N_{Ed}}{N_{c,Rd}} \leq 1{,}0$$

and

$$N_{c,Rd} = \frac{A f_y}{\gamma_{M0}} \quad \text{for class 1, 2 or 3 cross-sections}$$

Bending moment: The design value of the bending moment M_{Ed} at each cross-section shall satisfy:

$$\frac{M_{Ed}}{M_{c,Rd}} \leq 1{,}0$$

with

$$M_{c,Rd} = M_{pl,Rd} = \frac{W_{pl} f_y}{\gamma_{M0}} \quad \text{for class 1 or 2 cross sections}$$

$$M_{c,Rd} = M_{el,Rd} = \frac{W_{el,min} f_y}{\gamma_{M0}} \quad \text{for class 3 cross sections}$$

Shear: The design value of the shear force V_{Ed} at each cross section shall satisfy:

$$\frac{V_{Ed}}{V_{c,Rd}} \leq 1{,}0$$

For plastic design is the design plastic shear resistance V_{pLRd} is given as:

$$V_{pl,Rd} = \frac{A_v (f_y/\sqrt{3})}{\gamma_{M0}}$$

If

$$\frac{h_w}{t_w} > 72 \frac{\varepsilon}{\eta}$$

then additional shear buckling check is required by

$$\frac{V_{Ed}}{V_{b,Rd}} \leq 1{,}0$$ where $$V_{b,Rd} = V_{bw,Rd} + V_{bf,Rd} \leq \frac{\eta\, f_{yw}\, h_w\, t}{\sqrt{3}\,\gamma_{M1}}$$ and $$V_{bw,Rd} = \frac{\chi_w\, f_{yw}\, h_w\, t}{\sqrt{3}\,\gamma_{M1}}$$

Combined bending and shear:
Where the shear force is less than half the plastic shear resistance its effect on the moment resistance may be neglected. Otherwise, the reduced moment resistance should be taken as the design resistance of the cross-section, calculated using a reduced yield strength $f_{yr} = (1-\rho) f_y$

where $$\rho = \left(\frac{2V_{Ed}}{V_{pl,Rd}} - 1\right)^2$$

Combined bending and axial force:
For class 1 and 2 cross sections, the following criterion shall be satisfied:

$$M_{Ed} \leq M_{N,Rd}$$

For a rectangular solid sections:

$$M_{N,Rd} = M_{pl,Rd}\left[1 - \left(N_{Ed} / N_{pl,Rd}\right)^2\right]$$

For I sections:

$$M_{N,y,Rd} = M_{pl,y,Rd}\,(1\text{-}n)/(1\text{-}0{,}5a) \quad \text{but } M_{N,y,Rd} \leq M_{pl,y,Rd}$$

$$\text{for } n \leq a: \quad M_{N,z,Rd} = M_{pl,z,Rd}$$

$$\text{for } n > a: \quad M_{N,z,Rd} = M_{pl,z,Rd}\left[1 - \left(\frac{n-a}{1-a}\right)^2\right]$$

For bi-axial bending the following criterion may be used:

$$\left(\frac{M_{y,Ed}}{M_{N,y,Rd}}\right)^{\alpha} + \left(\frac{M_{z,Ed}}{M_{N,z,Rd}}\right)^{\beta} \leq 1$$

For Class 3 cross-sections the maximum longitudinal stress shall satisfy the criterion:

$$\sigma_{x,Ed} = \frac{f_y}{\gamma_{M0}}$$

Combined bending, shear and axial force:
Where V_{Ed} exceeds 50% of $V_{pl,Rd}$ the design resistance of the cross-section to combinations of moment and axial force should be calculated using a reduced yield strength

$(1\text{-}\rho)f_y$ for the shear area

where $\rho = (2V_{Ed} / V_{pl,Rd} - 1)^2$

Buckling resistance:

The design compression force N_{Ed} shall satisfy:

$$\frac{N_{Ed}}{N_{b,Rd}} \leq 1{,}0$$

where

$$N_{b,Rd} = \frac{\chi A f_y}{\gamma_{M1}}$$

and

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}^2}} \quad \text{but } \chi \leq 1.0$$

where

$$\Phi = 0.5[1 + \alpha(\bar{\lambda} - 0.2) + \bar{\lambda}^2]$$

The non-dimensional s1enden1ess is given by

$$\bar{\lambda} = \sqrt{\frac{Af_y}{N_{cr}}} = \frac{L_{cr}}{i}\frac{1}{\lambda_1}$$

with

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}} = 93.9\varepsilon \quad \text{and} \quad \varepsilon = \sqrt{\frac{235}{f_y}} \quad (f_y \text{ in N/mm}^2)$$

Torsional and torsional-flexural buckling:

For torsional and torsional-flexural buckling the slenderness is calculated as:

$$\bar{\lambda}_T = \sqrt{\frac{Af_y}{N_{cr}}}$$

where $N_{cr} = N_{cr,TF}$ but $N_{cr} < N_{cr,T}$

$N_{cr,TF}$ is the elastic critical torsional–flexural buckling force

$N_{cr,T}$ is the elastic critical torsional buckling force

$$N_{cr,T} = \frac{1}{i_0^2}\left(GI_t + \frac{\pi^2 EI_w}{l_T^2}\right)$$

$$i_0^2 = i_y^2 + i_z^2 + y_0^2 + z_0^2$$

and

$$N_{cr,TF} = \frac{N_{cr,y}}{2\beta}\left(1 + \frac{N_{cr,T}}{N_{cr,y}} - \sqrt{\left(1 - \frac{N_{cr,T}}{N_{cr,y}}\right)^2 + 4\left(\frac{y_0}{i_0}\right)^2\frac{N_{cr,T}}{N_{cr,y}}}\right)$$

where

$$\beta = 1 - \left(\frac{y_0}{i_0}\right)^2$$

Buckling resistance in bending:

Laterally unrestrained members subject to major axis bending are verified against lateral- torsional buckling as follows:

$$\frac{M_{Ed}}{M_{b,Rd}} \leq 1{,}0$$

where

$$M_{b,Rd} = \chi_{LT} W_y \frac{f_y}{\gamma_{M1}}$$

The Lateral torsional buckling curves General case

$$\chi_{LT} = \frac{1}{\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \bar{\lambda}_{LT}^2}} \quad \text{but } \chi_{LT} \leq 1.0$$

$$\Phi_{LT} = 0.5[1 + \alpha_{LT}(\bar{\lambda}_{LT} - 0.2) + \bar{\lambda}_{LT}^2]$$

$$\bar{\lambda}_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}}$$

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L_{cr}^2}\left(\frac{I_w}{I_z} + \frac{L_{cr}^2 GI_T}{\pi^2 EI_z}\right)^{0.5}$$

The Lateral torsional buckling curves for rolled sections or equivalent welded sections

For rolled or equivalent welded sections in bending the values of XLT for the appropriate nondimensional slenderness may be determined as:

$$\chi_{LT} = \frac{1}{\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\bar{\lambda}_{LT}^2}} \quad \text{but } \chi_{LT} \le 1.0 \text{ and } \chi_{LT} \le \frac{1}{\bar{\lambda}_{LT}^2}$$

and

$$\Phi_{LT} = 0.5[1 + \alpha_{LT}(\bar{\lambda}_{LT} - \bar{\lambda}_{LT,0}) + \beta\bar{\lambda}_{LT}^2]$$

Uniform members in bending and axial compression:

Members which are subjected to combined bending and axial compression should satisfy:

$$\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}} + k_{yy}\frac{M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M1}} + k_{yz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M1}} \le 1$$

$$\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}} + k_{zy}\frac{M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M1}} + k_{zz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M1}} \le 1$$

The values of the interaction factors are give below:

Table B.1: Interaction factors k_{ij} for members not susceptible to torsional deformations

Interaction factors	Type of sections	Design assumptions	
		elastic cross-sectional properties class 3, class 4	plastic cross-sectional properties class 1, class 2
k_{yy}	I-sections RHS-sections	$C_{my}\left(1+0{,}6\bar{\lambda}_y\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$ $\le C_{my}\left(1+0{,}6\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$	$C_{my}\left(1+(\bar{\lambda}_y-0{,}2)\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$ $\le C_{my}\left(1+0{,}8\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M1}}\right)$
k_{yz}	I-sections RHS-sections	k_{zz}	$0{,}6\ k_{zz}$
k_{zy}	I-sections RHS-sections	$0{,}8\ k_{yy}$	$0{,}6\ k_{yy}$
k_{zz}	I-sections	$C_{mz}\left(1+0{,}6\bar{\lambda}_z\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$ $\le C_{mz}\left(1+0{,}6\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$	$C_{mz}\left(1+(2\bar{\lambda}_z-0{,}6)\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$ $\le C_{mz}\left(1+1{,}4\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$
	RHS-sections		$C_{mz}\left(1+(\bar{\lambda}_z-0{,}2)\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$ $\le C_{mz}\left(1+0{,}8\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right)$
For I- and H-sections and rectangular hollow sections under axial compression and uniaxial bending $M_{y,Ed}$ the coefficient k_{zy} may be $k_{zy} = 0$.			

Table B.2: Interaction factors k_{ij} for members susceptible to torsional deformations

Interaction factors	Design assumptions	
	elastic cross-sectional properties class 3, class 4	plastic cross-sectional properties class 1, class 2
k_{yy}	k_{yy} from Table B.1	k_{yy} from Table B.1
k_{yz}	k_{yz} from Table B.1	k_{yz} from Table B.1
k_{zy}	$\left[1-\frac{0,05\bar{\lambda}_z}{(C_{mLT}-0,25)}\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right] \geq \left[1-\frac{0,05}{(C_{mLT}-0,25)}\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right]$	$\left[1-\frac{0,1\bar{\lambda}_z}{(C_{mLT}-0,25)}\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right] \geq \left[1-\frac{0,1}{(C_{mLT}-0,25)}\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}\right]$ for $\bar{\lambda}_z < 0,4$: $k_{zy} = 0,6+\bar{\lambda}_z \leq 1-\frac{0,1\bar{\lambda}_z}{(C_{mLT}-0,25)}\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M1}}$
k_{zz}	k_{zz} from Table B.1	k_{zz} from Table B.1

To summarize the maximum utilization factor is specified which should be ≤ 1.
The utilization factor is a measure for how close a section is to using its maximal section capacity and is obtained by the ratio between the applied loads and the loads at the design resistance.

8. The Print Menu

Print Heading
The five lines enable you to enter a Heading, for your printed output. Since the Heading is saved with the configuration file, it will stay unchanged after rebooting.

Structural Data:
- -Joints and Support Joints
- -Members
- -Joint Loads
- -Member Loads
- -Cross Section Data
- -Material Data

If the construction is calculated, *Analysis3D* will also print the results:
- -Joint Displacements
- -Member Forces
- -Member Stresses
- -Buckling
- -Reactions
- -Detailed Member Forces

Detailed Design
This option will send the results of the detailed design calculations to the printer.

Drawing
The presented drawing is copied to the printer as shown on the screen.

Printer Setup
This option allows you select your printer and paper together with changing your printing properties.

9. End-User License Agreement ("Agreement")

Please read this End-User License Agreement carefully before downloading or using *Analysis3D*.

Interpretation and Definitions

Interpretation

The words of which the initial letter is capitalized have meanings defined under the following conditions. The following definitions shall have the same meaning regardless of whether they appear in singular or in plural.

Definitions

For the purposes of this End-User License Agreement:

- **Agreement** means this End-User License Agreement that forms the entire agreement between You and the Company regarding the use of the Application.
- **Application** means the software program provided by the Company downloaded by You to a Device, named Analysis3D
- **Company** (referred to as either "the Company", "We", "Us" or "Our" in this Agreement) refers to Cuylaerts Engineering.
- **Content** refers to content such as text, images, or other information that can be posted, uploaded, linked to or otherwise made available by You, regardless of the form of that content.
- **Country** refers to Belgium
- **Device** means any device that can access the Application such as a computer, a cellphone or a digital tablet.
- **Third-Party Services** means any services or content (including data, information, applications and other products services) provided by a third-party that may be displayed, included or made available by the Application.
- **You** means the individual accessing or using the Application or the company, or other legal entity on behalf of which such individual is accessing or using the Application, as applicable.

Acknowledgment

By clicking the "I Agree" button, downloading or using the Application, you are agreeing to be bound by the terms and conditions of this Agreement. If You do not agree to the terms of this Agreement, do not click on the "I Agree" button, do not download or do not use the Application.

This Agreement is a legal document between You and the Company and it governs your use of the Application made available to You by the Company.

The Application is licensed, not sold, to You by the Company for use strictly in accordance with the terms of this Agreement.

License

Scope of License

The Company grants You a revocable, non-exclusive, non-transferable, limited license to download, install and use the Application strictly in accordance with the terms of this Agreement.

The license that is granted to You by the Company is solely for your personal, non-commercial purposes strictly in accordance with the terms of this Agreement.

You are not permitted to

- Sell, transmit, host or otherwise commercially exploit our application
- Copy or use our application for any other purposes except for personal, non-commercial purposes
- Modify, decrypt, reverse compile or reverse engineer our application

Third-Party Services

The Application may display, include or make available third-party content (including data, information, applications and other products services) or provide links to third-party websites or services.

You acknowledge and agree that the Company shall not be responsible for any Third-party Services, including their accuracy, completeness, timeliness, validity, copyright compliance, legality, decency, quality or any other aspect thereof. The Company does not assume and shall not have any liability or responsibility to You or any other person or entity for any Third-party Services.

You must comply with applicable Third parties' Terms of agreement when using the Application. Third-party Services and links thereto are provided solely as a convenience to You and You access and use them entirely at your own risk and subject to such third parties' Terms and conditions.

Term and Termination

This Agreement shall remain in effect until terminated by You or the Company. The Company may, in its sole discretion, at any time and for any or no reason, suspend or terminate this Agreement with or without prior notice.

This Agreement will terminate immediately, without prior notice from the Company, in the event that you fail to comply with any provision of this Agreement. You may also terminate this Agreement by deleting the Application and all copies thereof from your Device or from your computer.

Upon termination of this Agreement, You shall cease all use of the Application and delete all copies of the Application from your Device.

Termination of this Agreement will not limit any of the Company's rights or remedies at law or in equity in case of breach by You (during the term of this Agreement) of any of your obligations under the present Agreement.

Indemnification

You agree to indemnify and hold the Company and its parents, subsidiaries, affiliates, officers, employees, agents, partners and licensors (if any) harmless from any claim or demand, including reasonable attorneys' fees, due to or arising out of your: (a) use of the Application; (b) violation of this Agreement or any law or regulation; or (c) violation of any right of a third party.

No Warranties

The Application is provided to You "AS IS" and "AS AVAILABLE" and with all faults and defects without warranty of any kind. To the maximum extent permitted under applicable law, the Company, on its own behalf and on behalf of its affiliates and its and their respective licensors and service providers, expressly disclaims all warranties, whether express, implied, statutory or otherwise, with respect to the Application, including all implied warranties of merchantability, fitness for a particular purpose, title and non-infringement, and warranties that may arise out of course of dealing, course of performance, usage or trade practice. Without limitation to the foregoing, the Company provides no warranty or undertaking, and makes no representation of any kind that the Application will meet your requirements, achieve any intended results, be compatible or work with any other software, applications, systems or services, operate without interruption, meet any performance or reliability standards or be error free or that any errors or defects can or will be corrected.

Without limiting the foregoing, neither the Company nor any of the company's provider makes any representation or warranty of any kind, express or implied: (i) as to the operation or availability of the Application, or the information, content, and materials or products included thereon; (ii) that the Application will be uninterrupted or error-free; (iii) as to the accuracy, reliability, or currency of any information or content provided through the Application; or (iv) that the Application, its servers, the content, or e-mails sent from or on behalf of the Company are free of viruses, scripts, trojan horses, worms, malware, timebombs or other harmful components.

Some jurisdictions do not allow the exclusion of certain types of warranties or limitations on applicable statutory rights of a consumer, so some or all of the above exclusions and limitations may not apply to You. But in such a case the exclusions and limitations set forth in this section shall be applied to the greatest extent enforceable under applicable law. To the extent any warranty exists under law that cannot be disclaimed, the Company shall be solely responsible for such warranty.

Limitation of Liability

Notwithstanding any damages that You might incur, the entire liability of the Company and any of its suppliers under any provision of this Agreement and your exclusive remedy for all of the foregoing shall be limited to the amount actually paid by You for the Application or through the Application or 100 USD if You haven't purchased anything through the Application.

To the maximum extent permitted by applicable law, in no event shall the Company or its suppliers be liable for any special, incidental, indirect, or consequential damages whatsoever (including, but not limited to, damages for loss of profits, loss of data or other information, for business interruption, for personal injury, loss of privacy arising out of or in any way related to the use of or inability to use the Application, third-party software and/or third-party hardware used with the Application, or otherwise in connection with any provision of this Agreement), even if the Company or any supplier has been advised of the possibility of such damages and even if the remedy fails of its essential purpose.

Some states/jurisdictions do not allow the exclusion or limitation of incidental or consequential damages, so the above limitation or exclusion may not apply to You.

Severability and Waiver

Severability

If any provision of this Agreement is held to be unenforceable or invalid, such provision will be changed and interpreted to accomplish the objectives of such provision to the greatest extent possible under applicable law and the remaining provisions will continue in full force and effect.

Waiver

Except as provided herein, the failure to exercise a right or to require performance of an obligation under this Agreement shall not affect a party's ability to exercise such right or require such performance at any time thereafter nor shall be the waiver of a breach constitute a waiver of any subsequent breach.

Product Claims

The Company does not make any warranties concerning the Application.

United States Legal Compliance

You represent and warrant that (i) You are not located in a country that is subject to the United States government embargo, or that has been designated by the United States government as a "terrorist supporting" country, and (ii) You are not listed on any United States government list of prohibited or restricted parties.

Changes to this Agreement

The Company reserves the right, at its sole discretion, to modify or replace this Agreement at any time. If a revision is material, we will provide at least 30 days' notice prior to any new terms taking effect. What constitutes a material change will be determined at the sole discretion of the Company.

By continuing to access or use the Application after any revisions become effective, You agree to be bound by the revised terms. If You do not agree to the new terms, You are no longer authorized to use the Application.

Governing Law

The laws of the Country, excluding its conflicts of law rules, shall govern this Agreement and your use of the Application. Your use of the Application may also be subject to other local, state, national, or international laws.

Entire Agreement

The Agreement constitutes the entire agreement between You and the Company regarding your use of the Application and supersedes all prior and contemporaneous written or oral agreements between You and the Company.

You may be subject to additional terms and conditions that apply when You use or purchase other Company's services, which the Company will provide to You at the time of such use or purchase.

Contact Us

If you have any questions about this Agreement, please contact Us:

- By email: frank@cuylaerts.net