lateral buckling restraint - attaches - steel check - creep - charges climatiques - dynamic analysis - lateral buckling - brandweerstandsanalyse - timber - 1st order - verstijvers - buisverbinding - diseño de planos de armaduras - pandeo lateral - verbindingen - shear connection - verificación - armatures longitudinales - pórtico - unión base columna - voorontwerp - unión tubular - haunch - connexion moment - cimbras - vérification acier - unity check - Eurocode 2 - mesh - retaining wall - raidisseur - Eurocode 3 - longitudes de pandeo - connections - ACI 138 - acero - 2nd ordre - portal frame - Eurocode 8 - andamios - kip - dwarskrachtverbinding - BS 8110 - dalle de fondation - seismische analyse - armaduras longitudinales - BIM - gelaste verbinding - 2de orde - buckling - funderingszool - poutre sur plusieurs appuis - maillage - malla - uniones - 2D raamwerken - fire resistance analysis - voiles - cracked deformation - gescheurde doorbuiging - longueurs de flambement - pandeo - reinforcement - unity check - cantonera - dynamische analyse - hout - ossatures 3D - koudgevormde profielen - placa de extreme - 1er orden - continuous beam - connexion soudée - momentverbinding - praktische wapening - renforts au déversement - fluencia - estribos - déformation fissurée - EHE - beugels - Eurocódigo 3 - platine de bout - análisis dinámico - column base plate - kruip - rigid link - welded connection - charpente métallique - moment connections - estructuras 2D - kniestuk - assemblage métallique - 3D raamwerken - second ordre - beam grid - cargas climáticas - Eurocode 2 - Eurocode 5 - wall - deformación fisurada - lien rigide - enlace rígido - 2D frames - estructuras 3D - éléments finis - vloerplaat - steel connection - scheurvorming - integrated connection design - armatures pratiques - analyse sismique - nieve y viento - practical reinforcement - charges mobiles - dalle - wapening - perfiles conformados en frío - Eurocode 3 - connexion tubulaire - unión a momento - 3D frames - treillis de poutres - roof truss - practical reinforcement design - portique - kipsteunen - análisis sísmico - Eurocode 8 - seismic analysis - B.A.E.L 91 - uniones atornilladas - bolts - ossatures 2D - eindige elementen - losa de cimentación - restricciones para el pandeo lateral - optimisation - wand - kniklengtes - end plate - dakspanten - kolomvoetverbinding - stirrups - acier - staalcontrole - cálculo de uniones integrado - paroi - dessin du plan de ferraillage - stiffeners - mobiele lasten - Eurocódigo 8 - Eurocódigo 5 - longitudinal reinorcement - doorlopende liggers - rigidizador - beton armé - fluage - CTE - connexion pied de poteau - langswapening - connexions - hormigón - neige et vent - elementos finitos -

armaduras - cold formed steel - jarret - uittekenen wapening - puente grúa - analyse dynamique - flambement - keerwanden - optimisation - steel - cercha - 2º orden - slab on grade foundation - entramado de vigas - Eurocode 5 - prédimensionnement - multi span beam - bouten - armatures - floor slab - poutre continue - pared - staal - 1er ordre - NEN 6770-6771 - connexion cisaillement - losa - déversement - viga continua - predimensionering - 1ste orde - unión metálica - CM 66 - madera - análisis resistencia al fuego - verbindingen - 2nd order - bois - Eurocode 2 - profilés formés à froid - verificación acero - predesign - unión soldada - fisuración - beton - muro de contención - optimalisatie - foundation pads - fissuration - concrete - AISC-LRFD - HCSS - assemblage métallique - Eurocode 3 - viga con varios apoyos - armaduras prácticas - balkenroosters - unión a cortante - buckling length - boulons - cracking - Eurocode 8 - knik - Eurocode 2 - radier - eindplaat - Eurocódigo 2 - FEM - tornillos - NEN 6720 - moving loads - balk op meerdere steunpunten - cargas móviles - funderingsplaat - étriers - analyse resistance au feu - cercha - globale knikfactor - dynamische analyse - wapening

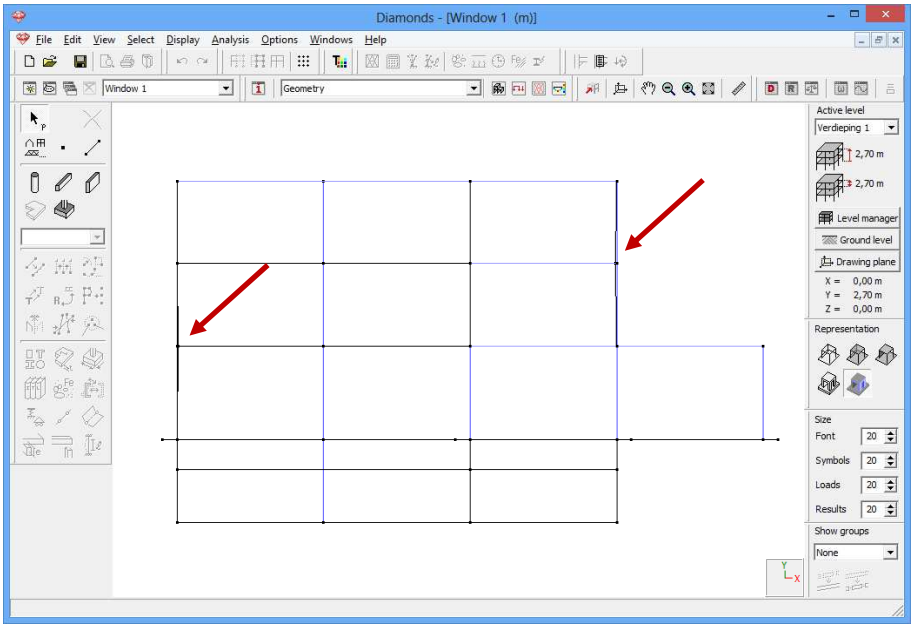
Table of contents

1. Geometry 4
   1.1. Projection 4
   1.2. Intersections 6
   1.3. Extended structure verification 6
   1.4. Releases at bar ends 7
   1.5. Copy and paste the design type 7
   1.6. Ground level 8
2. Loads 10
   2.1. Lay out of the window for the load groups 10
   2.2. NEN 8700 11
   2.3. Linked load groups 11
   2.4. Defining loads 12
      2.4.1. General 12
      2.4.2. Extension of the wind generator 13
      2.4.3. Extension of the snow generator 16
      2.4.4. Adjusted definition of temperature loads 19
      2.4.5. Lineair dynamic load 20
      2.4.6. Seismic load 23
      2.4.7. Masses as load 26
   2.5. Load combination generation 27
   2.6. Extended load configuration 27
3. Elastic analysis 29
   3.1. New calculation core 29
   3.2. Modified window for elastic analysis 29
      3.2.1. Structural 29
      3.2.2. Soil 33
      3.2.1. Dynamic 34
   3.3. Modal analysis 36
   3.4. Equilibrium check and global buckling factor 39
4. Design 41
   4.1. Extended filter on steel and timber verification 41
   4.2. Extended connection verification (according to AISC) 41
5. Report 43
   5.1. Terrain parameters wind and snow 43
   5.2. Dynamic and seismic loads 44
   5.3. Complete calculation of the design code verification (for steel or timber) 45

1. Geometry

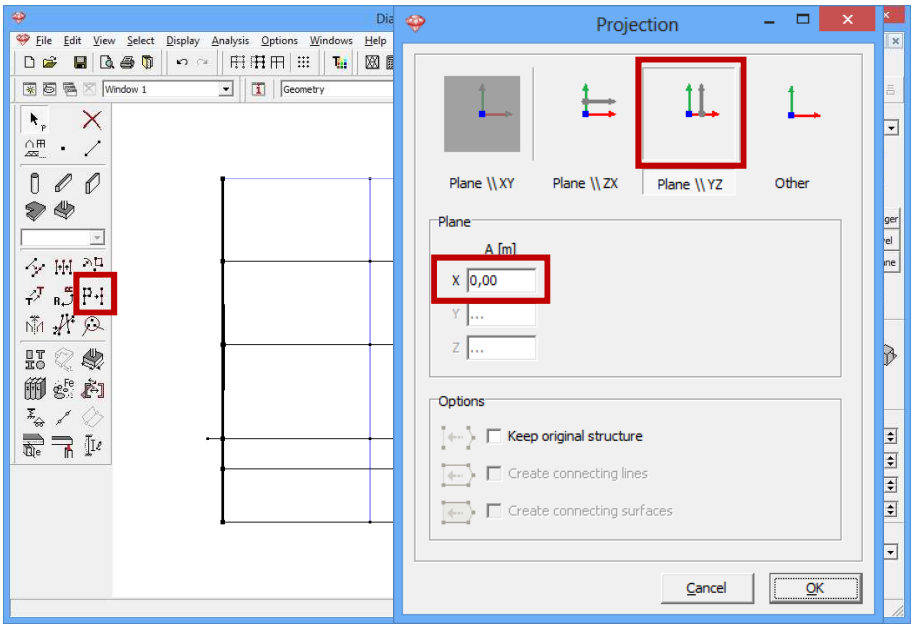
1.1. Projection

When making a 3D model it can happen that a few points lay out of the plane while this was not the purpose. In a front or side view you can clearly recognize this by the lines are drawn 'bolder' than others, for example:

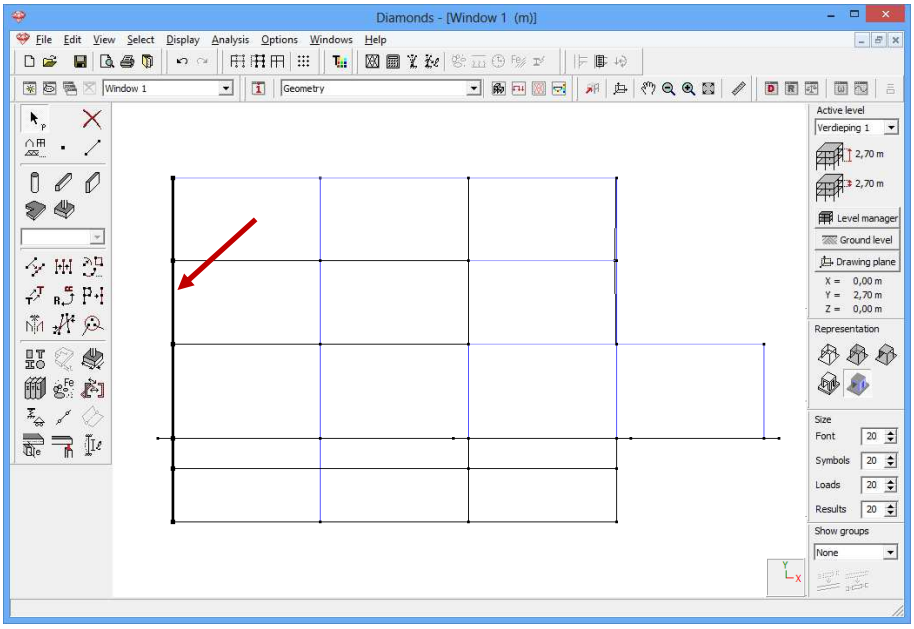


To achieve a good model (and thus a good mesh), it is important that such 'mistakes' are remedied. In Diamonds 2013 you can easily solved this by projecting the points on a certain plane:

- Select the points (all plates and lines lying in this plane may also be selected) that you would like to lie in the same plane.
- In the geometry window click on the button .



- It’s the intention that the selected elements lie in the same plane, so we choose for a plane parallel to the YZ-plane.
  In this example the position of the projection plane is captured by a single coordinate, namely X=0m. In other words, Diamonds will now project all selected points perpendicular to the plane X=0m.
- Result: the wall is straight again. An analogous operation can be performed on the other wall.



It’s not only possible to project on a vertical or horizontal plane, you can also project in an arbitrary plane. Therefore enter the coordinates of 3 points describing the plane.

The options on the bottom of the projection window:

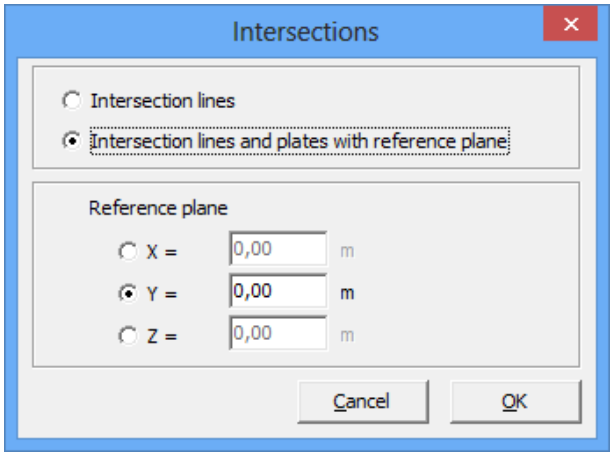
- ‘Keep original structure’: the selection being projected will remain in the model.
- ‘Create connecting lines’: Diamonds will draw the connecting lines between the original and the projected structure.
- ‘Create connecting surfaces’: Diamonds will generate the connecting surfaces between the original and the projected structure.

1.2. Intersections

In Diamonds 2013 the functions for intersections:

- of lines
- or of lines and plates with a reference plane (until Diamonds 2012 r02 available via)

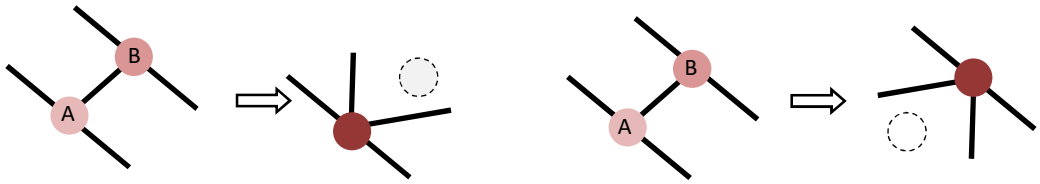
are available through the button . Diamonds will ask you which function you would like to apply:



1.3. Extended structure verification

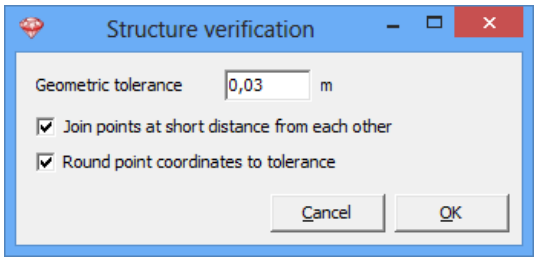
The structure verification includes i.e. the following checks:

- Merging points that are to close (depending on the geometric tolerance) to each other.
  Example: points A and B are located at a distance of 2cm from each other. The geometric tolerance is set to 5cm. Then Diamonds will either join point A with point B or point B with point A.



- Rounding the point coordinates to the specified tolerance.
  Example: when importing a DXF-file a point has a X-coordinate of 2,15940041m. When setting the geometric tolerance to 0,01m, the X-coordinate now becomes 2,16000000m. When the geometric tolerance is set to 1,00m, the X-coordinate becomes 2,00000000m.
- Remove plates for which not all the points lie in the same plane.
- …

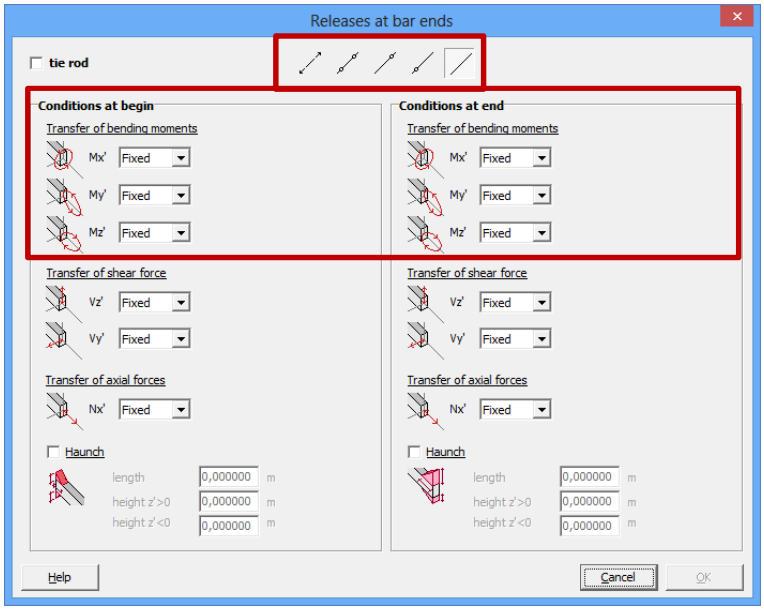
You'll see the following window when clicking on the button :



With the new structure verification you can choose whether or not the first two checks (merging points and rounding the coordinates) should be performed by (un)checking the corresponding check boxes. The used geometric tolerance can be indicated in this window. This has the same value as the geometric tolerance with 'Options -> Preferences -> tab Drawing'.

1.4. Releases at bar ends

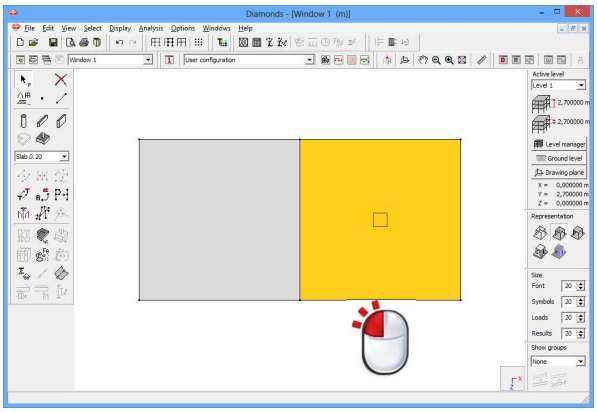
The window with the releases at bar ends is foreseen of several buttons which allow you to quickly define tie rods and hinges. The relevant moments are set free or fixed in the fields below.

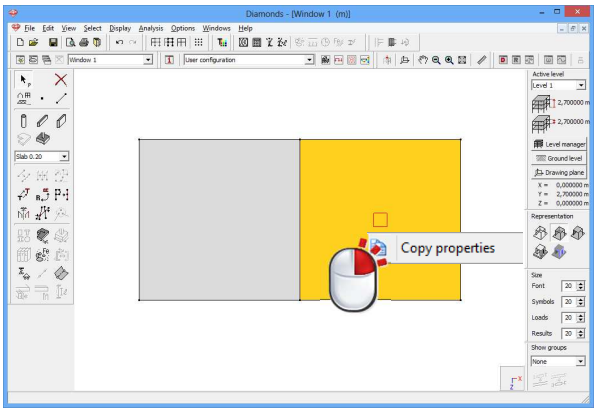


1.5. Copy and paste the design type

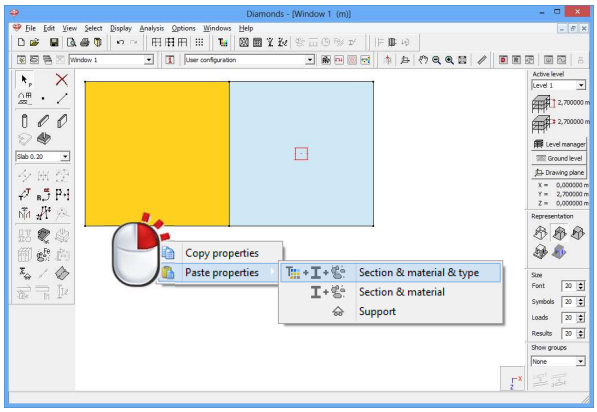
Copy-pasting the cross-section, the material, the supports and design types in Diamonds 2013 is done this way:

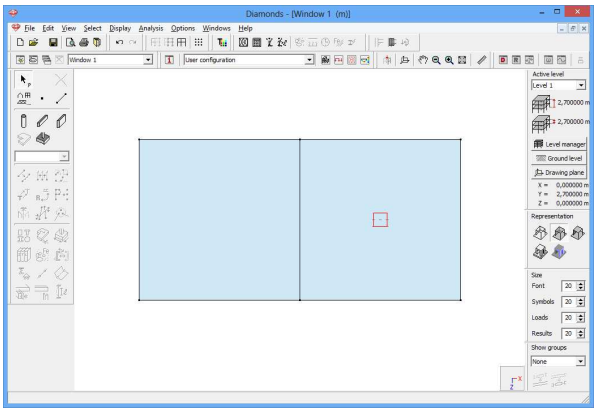
- Select the element from which you would like to copy one or multiple properties (in this case the plate on the right). Then click once with the right mouse button.





- Click on ‘Copy properties’.
- Select the element on which you would like to paste one or multiple properties (in this case the plate on the left). Click once with the right mouse button.
- Finally choose the properties you would like to paste.



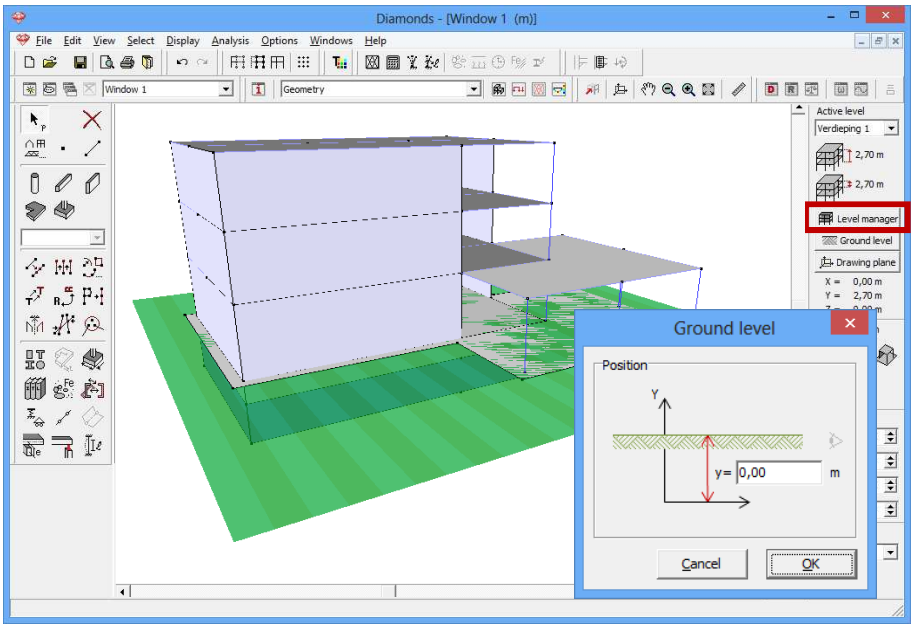


1.6. Ground level

The button Ground level allows you to defined the ground level in relation to the global coordinate system. This way Diamonds knows:

- how much soil you dig down when calculating the soil layers.
- how many meters above the ground level an element is located when generating wind.

By click on the icon you can choose to visualize the ground level in your model or not.



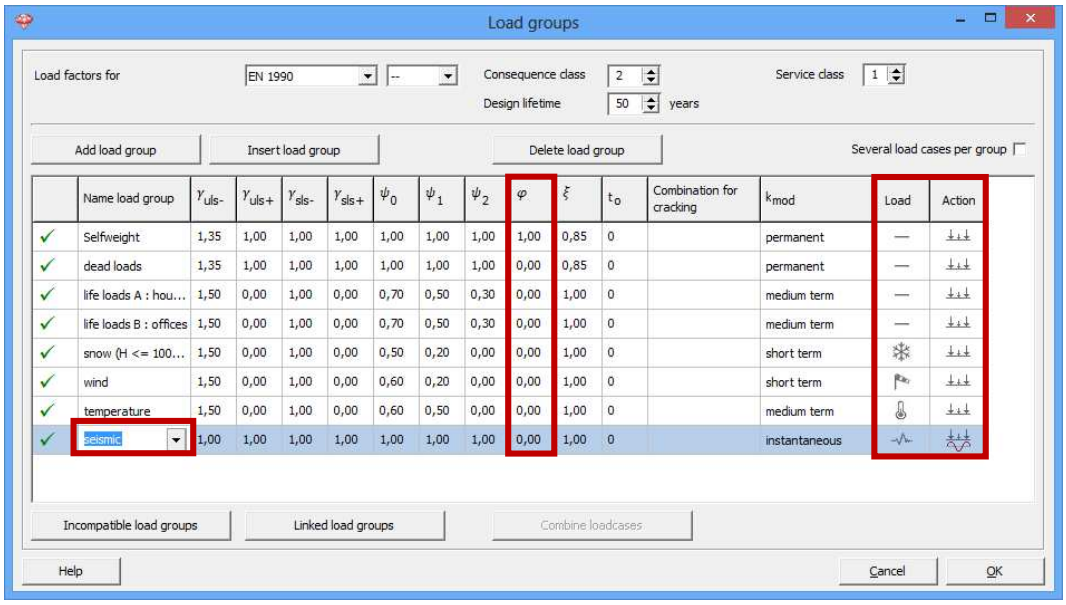
Remarks:

- **The soil calculations and the generation of wind don't have meaning if you haven't set the ground level!**
- Don't be mistaking, when generating snow you enter the height of the building in relation to the see level, not to the ground level! The generation of snow is thus independet of the location of the ground level!

2. Loads

2.1. Lay out of the window for the load groups

Because of the dynamic and seismic module in Diamonds the window for the load groups is expanded . We now go through the new features:



- In the column 'Name load group', you'll find the difference load types according to the selected standard. For the standard 'Eurocode EN 1990' you can choose between:
  - Self-weight
  - Dead loads
  - Life load types A until type H
  - Snow
  - Wind
  - Temperature
  - **Seismic (new since Diamonds 2013)**

- In the final two columns 'Load' and 'Action' indicate the character of the load groups. In the window above:
  - 'Self-weight' is a normal — static load.
  - 'Dead loads' is a normal — static load.
  - 'Life load A' is a normal — static load.
  - 'Life load B' is a normal — dynamic load.
  - 'Snow (H<=1000m)' a static snow load.
  - 'Wind' a static wind load.
  - 'Temperature' a static temperature load .
  - 'Seismic' a dynamic seismic load.

For ‘Life loads’ you can choose between a static or a dynamic load. You switch between both by click on the icon in the column ‘Action’. A seismic load groups always has a dynamic character.
For wind, snow, temperature and seismic the column ‘Load’ is filled in for you.

- In the column φ you enter the correlaction coefficient for the design mass.
  By setting the correlation coefficient on ‘1’ you can defines masse in the structure without have the draw the elements in the model (see §2.4.7).

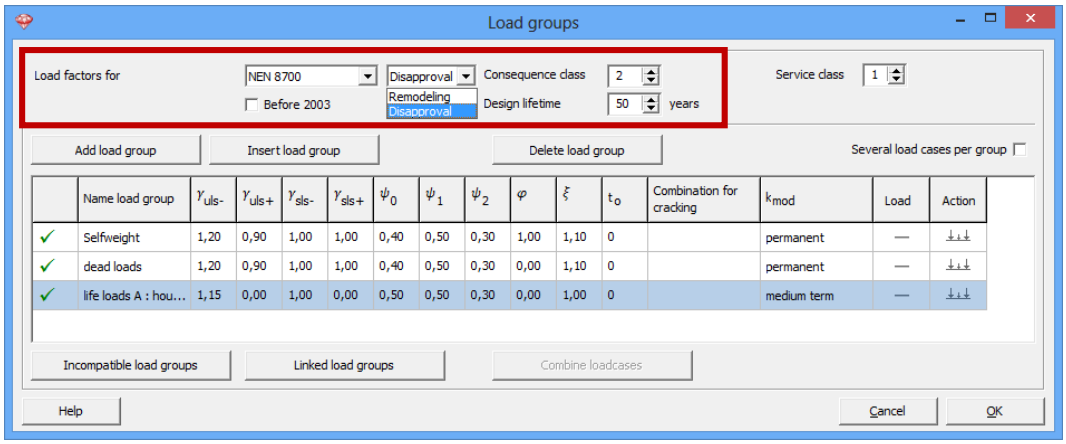
2.2. NEN 8700

In the Netherlands the standard NEN 8700 may be used to assess the structural safety of existing buildings in remodeling and disapproval.

This standard describes customized safety and combination factors in relation to:

- The application (remodel or disapproval)
- The consequence class
- The design life time
- The period during which time the building was build (before or after 2003).

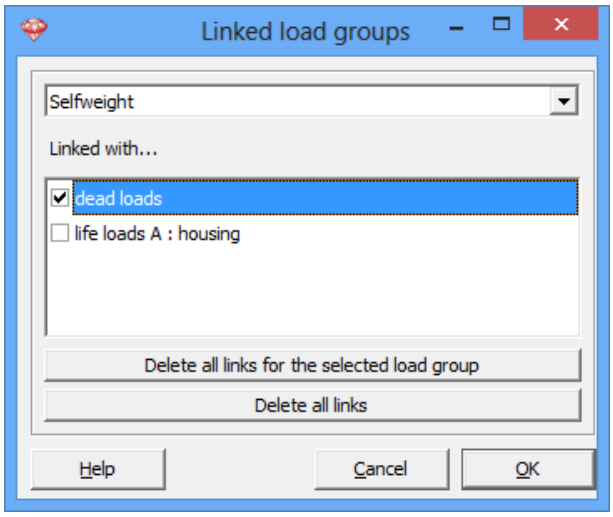
The above parameters can be set in the window with the load groups:



The used safety and combination factors can be found in NEN 8700 Table A1.1., Table A1.2(B) and (C).

2.3. Linked load groups

Diamonds 2013 offers you the possibility to link load groups that have the same favorable or unfavorable effect on the structure. This will reduce the number of load combinations, since the case where one load groups work favorable and the other one unfaborable will be eliminated by Diamonds. An example of this is the self-weight and a dead load with a downward direction. To define linked load groups, click on the button Linked load groups .



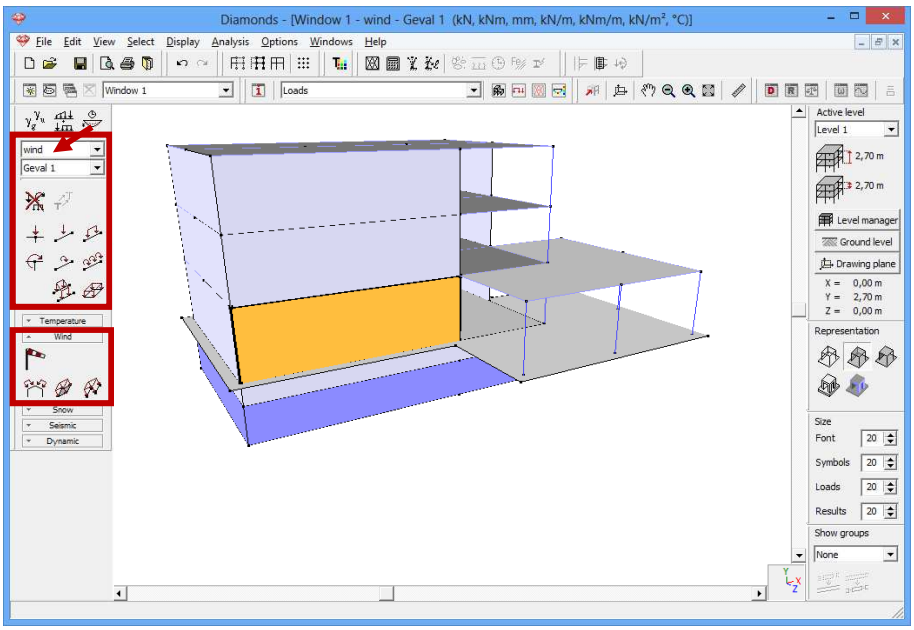
To link a number of load groups to another load group, select on the these load groups from the pull down menu on top of the dialog box and indicate below the linked load groups.

You can delete the linked groups with the selected load group by clicking on the button Delete all links for the selected load group. If you wish to delete all linked load groups, click on Delete all links.

2.4. Defining loads

2.4.1. General

Depending on the chosen load group with $\gamma_g^{\gamma_u}$, Diamonds will unfold only the relevant lists with buttons with regard to the selected load group. Doing so, Diamonds is not overloaded with buttons that you don't need. For example for the load group 'Wind' only these buttons are available.



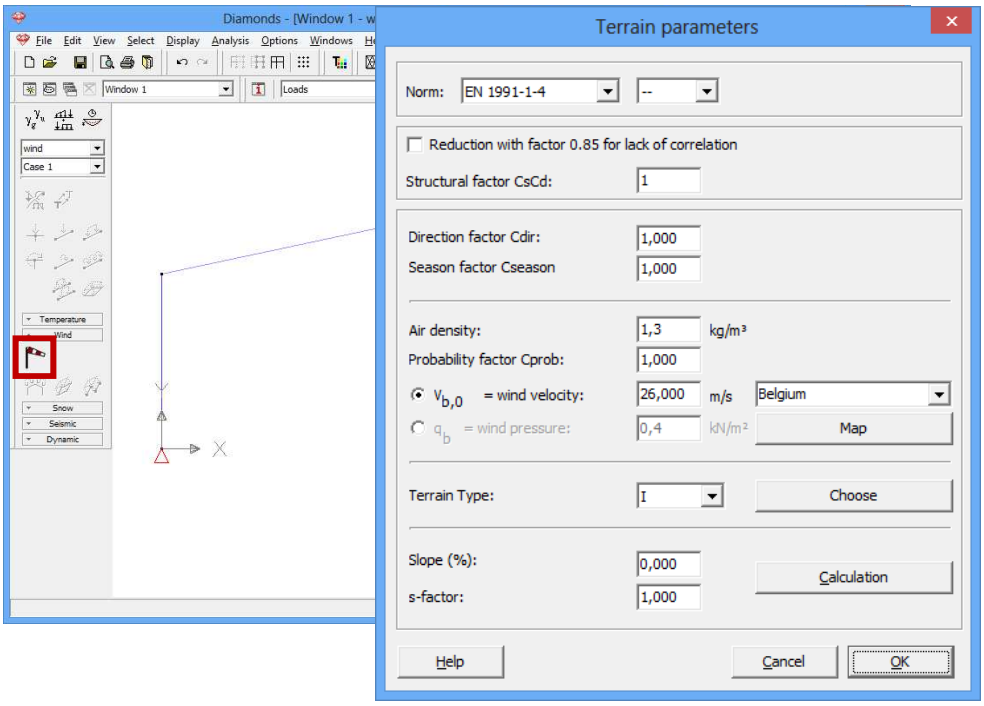
2.4.2. Extension of the wind generator

With Diamonds 2013 it's not only possible to generate wind on the frame, but also on a surface or on a group of bars. To generate wind, you first have to set the general wind parameters. Then you select the elements on which you want to apply wind and finally you launch the generator.

2.4.2.1. General wind parameters: ground level, wind standard and terrain parameters

Whether you generate wind on a frame, a surface or a group of bars, you should always set/verify the ground level (see §1.6).

Then select the wind standard and the terrain parameters via . These parameters have the same meaning as in previous version of Diamonds.



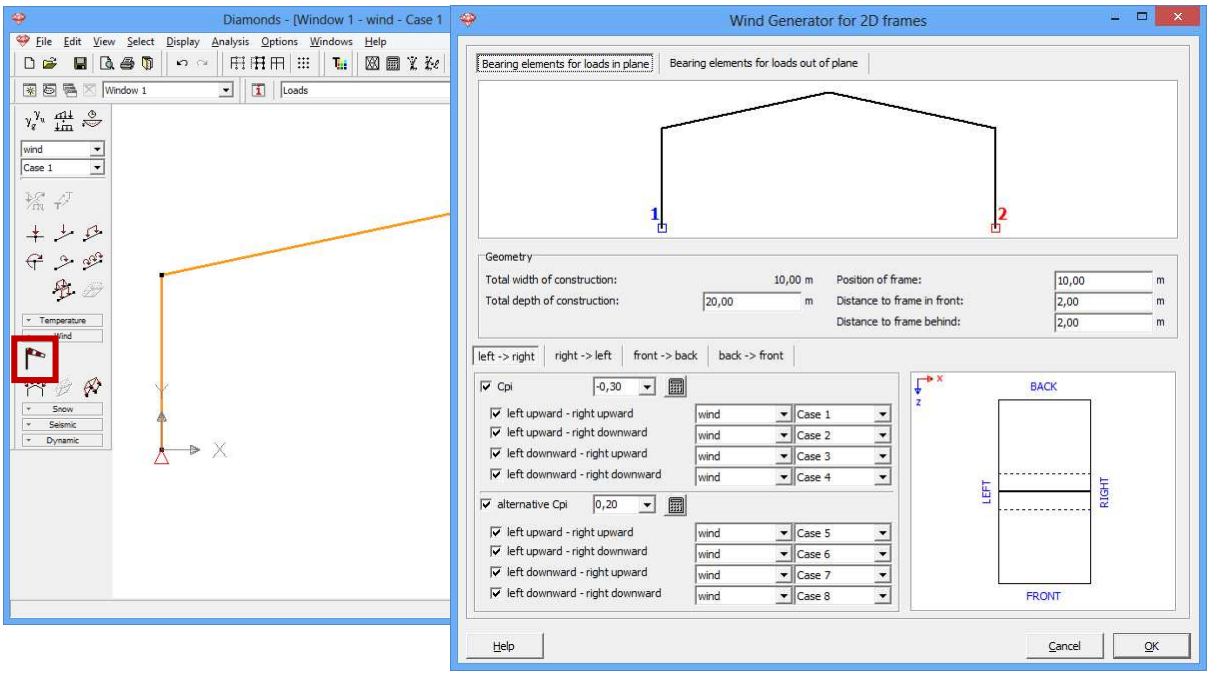
Remarks:

- The terrain parameters can be demanded in the report manager (with Diamonds 2013) (see 5.1).
- Adjusting the standard or the terrain parameters will cause the previous generated wind loads to be deleted!

2.4.2.2. Wind on a frame

Select the frame on which you wish to generate wind, both the buttons and become active. Click on the button for wind on a frame. You'll see the screen below. This window maintains it's functionalities like in previous versions of Diamonds.

Note that the selected frame must lie in a 2D plane.



2.4.2.3. Wind on a surface

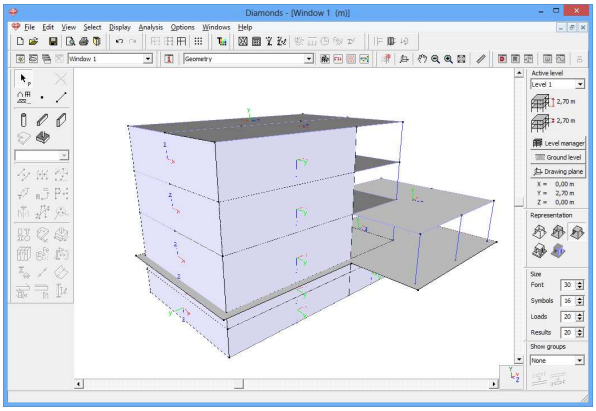
Select the surfaces on which you wish to generate wind. Click on the button . You can generate up to 4 cases of wind at once.

- Select the load groups in which Diamonds should place the wind.
- Enter the internal c_{pi} and external c_{pe} pressure coefficients. The internal w_i, the external w_e and the net wind pressure w_n are calculated automatically.

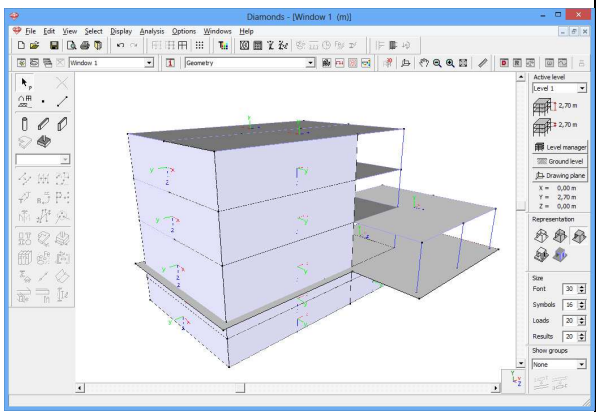
The selected surfaces **don't** have to lie in the same plane.

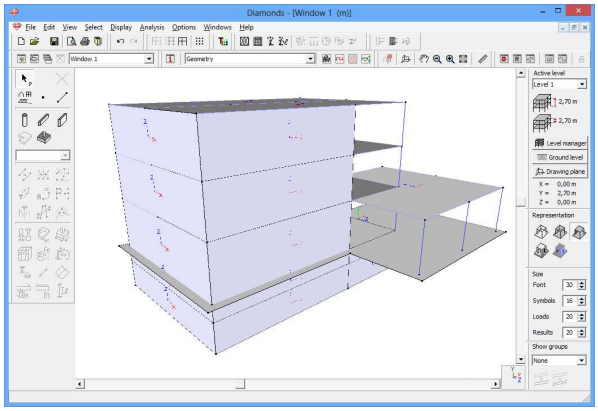
Make sure the local coordinate system of the surfaces is orientated in the same direction: either all local y'-axes directed outward or either all local y'-axes directed inward.

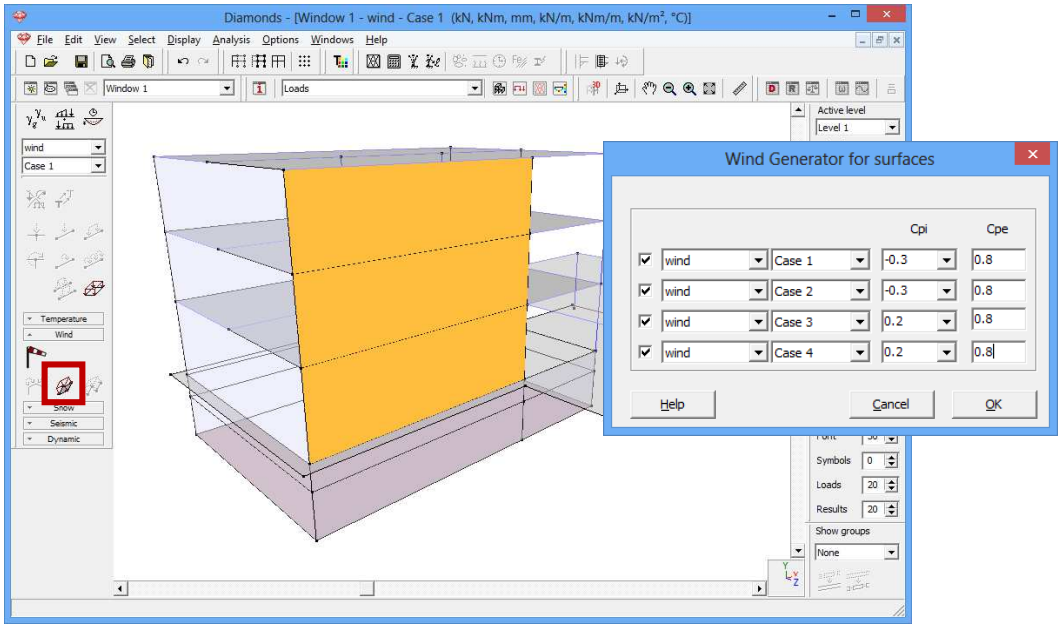
BAD
for some plates the local y'-axes is directed inward, for other plates outward



GOOD all local y'-axes are directed outward	GOOD all local y'-axes are directed inward







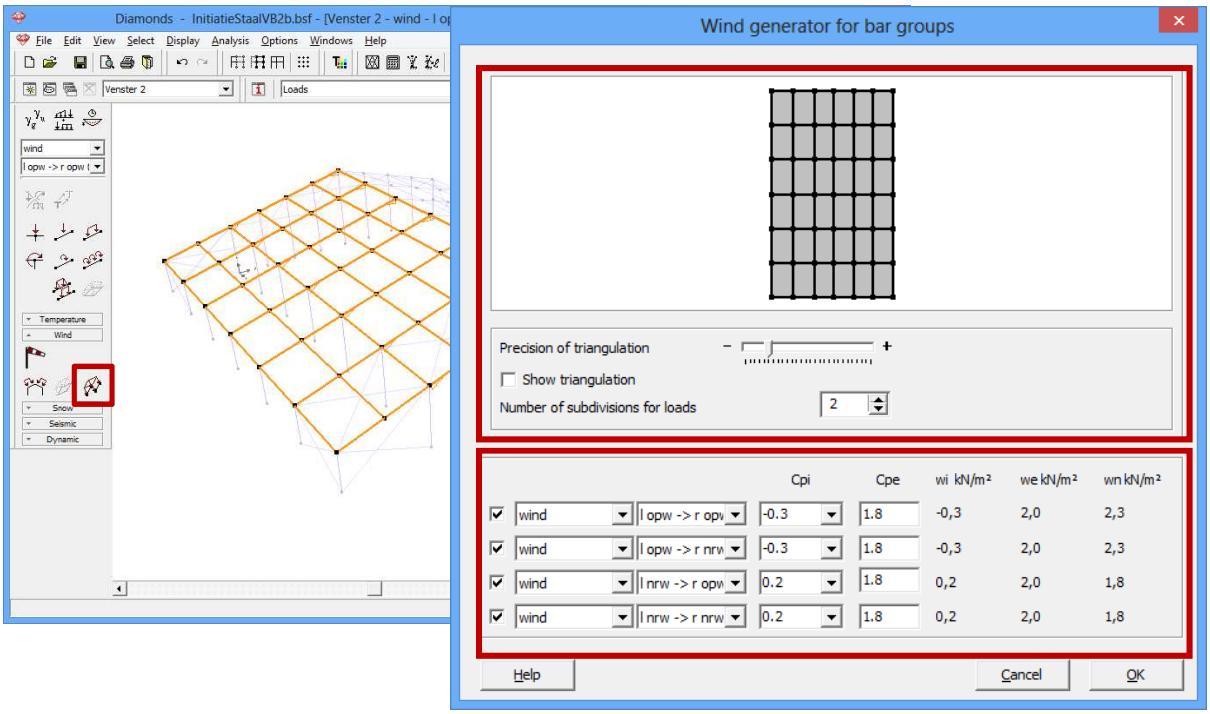
2.4.2.4. Wind on a group of bars

Select the group of bars on which you wish to generate wind, both the buttons and become active. Click on the button for wind on group of bars. You'll see the screen below.

The upper half of this window works the same way as the surface load generator (see Reference manual). In the lower half of this window:

- You select the load groups in which Diamonds should place the wind.
- Enter the internal c_{pi} and external c_{pe} pressure coefficients. The internal w_i, the external w_e and the net wind pressure w_n are calculated automatically.

The selected bars should lie **in the same** plane.

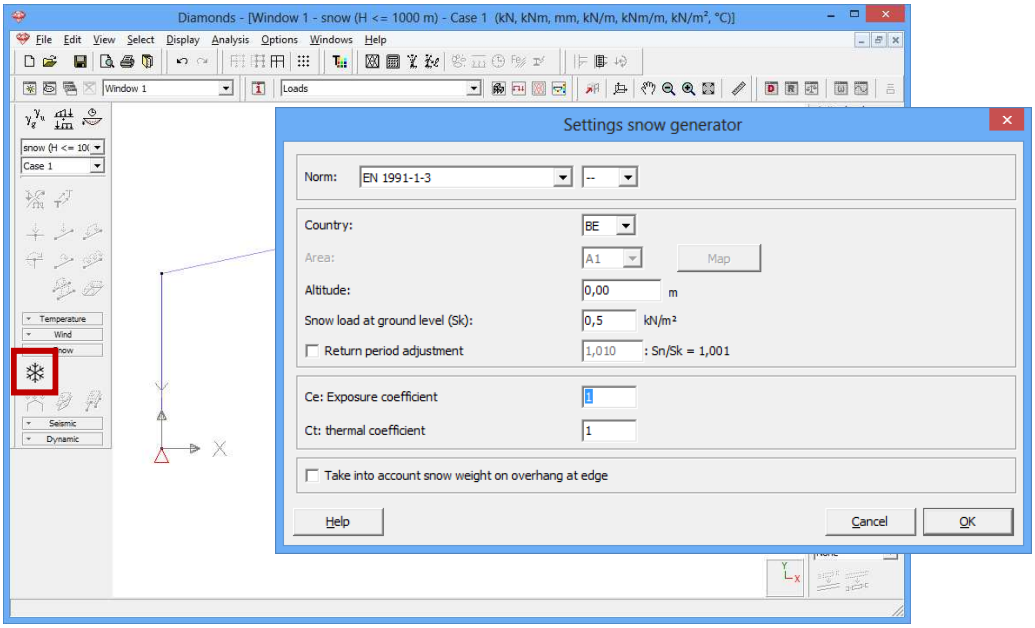


2.4.3. Extension of the snow generator

With Diamonds 2013 it's not only possible to generate snow on the frame, but also on a surface or on a group of bars. To generate snow you first have to set the general snow parameters. Then you select the elements on which you want to apply snow and finally you launch the generator.

2.4.3.1. Snow standard and terrain parameters

Choose the snow standard and the terrain parameters with . These parameters have the same meaning as in previous versions of Diamonds.



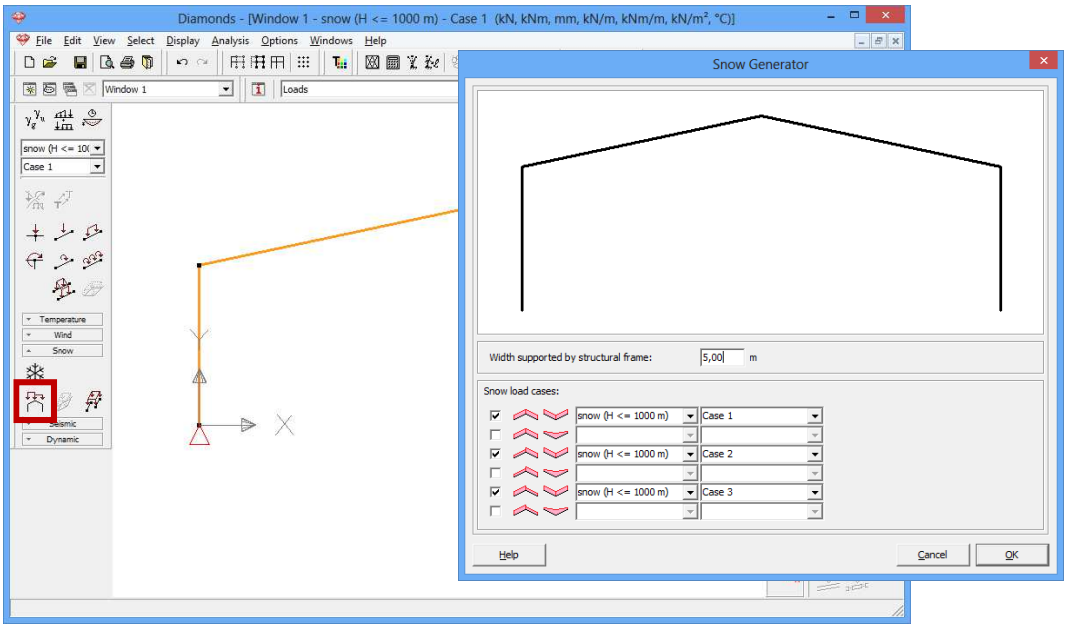
Remarks:

- The terrain parameters can be demanded in the report manager (with Diamonds 2013) (see 5.1).
- Adjusting the standard or the terrain parameters will cause the previous generated snow loads to be deleted!

2.4.3.2. Snow on a frame

Select the frame on which you wish to generate snow, both the buttons and become active. Click on the button for snow on a frame. You'll see the screen below. This window maintains it's functionalities like in previous versions of Diamonds.

Note that the selected frame must lie in **a 2D plane**.



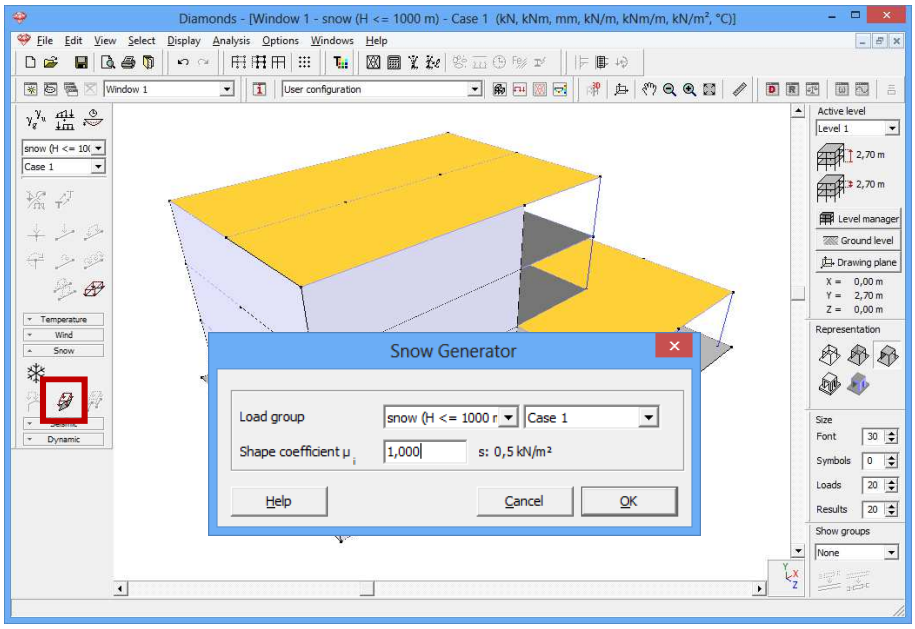
2.4.3.3. Snow on a surface

Select the surfaces on which you wish to generate snow. Click on the button .

- Select the load group in which Diamonds should place the snow.
- Enter the shape coefficient μ_i.

Note that the selected surfaces **don't** have to lie in the same plane.

Make sure the local coordinate system of the surfaces is orientated in the same direction: either all local y'-axes directed outward or either all local y'-axes directed inward (see §2.4.2.3).



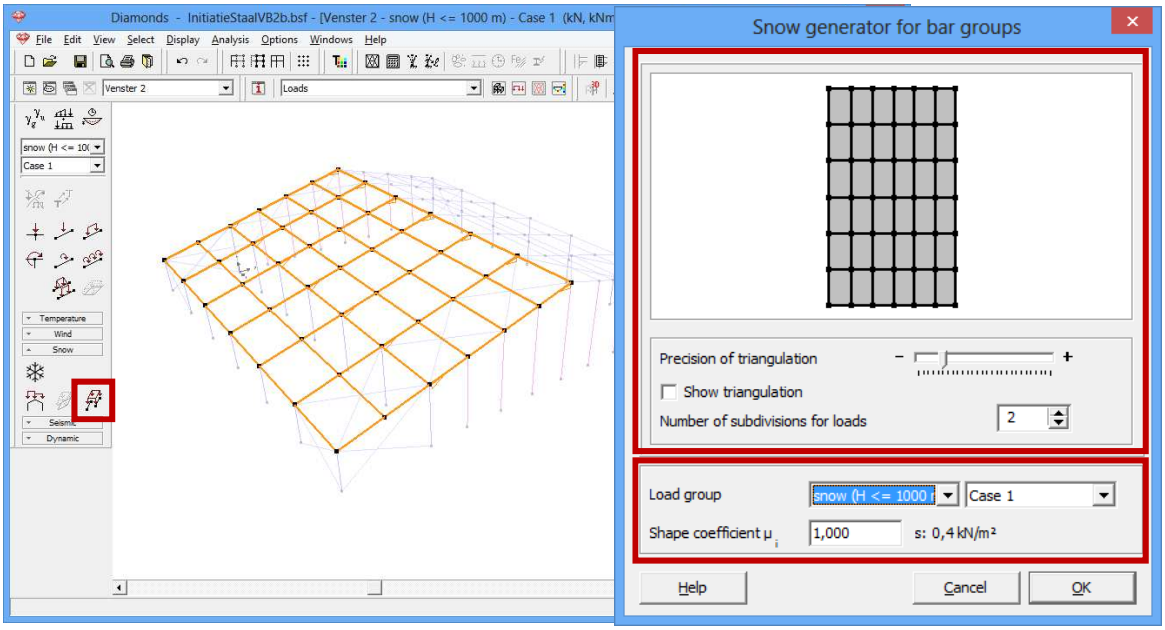
2.4.3.4. Snow on a group of bars

Select the group of bars on which you wish to generate snow, both the buttons and become active. Click on the button for snow on a group of bars. You'll see the screen below. This window maintains its functionalities like in previous versions of Diamonds.

The upper half of this window works the same way as the surface load generator (see Reference manual). In the lower half of this window:

- Select the load group in which Diamonds should place the snow.
- Enter the shape coefficient μ_i.

The selected bars should lie **in the same** plane.



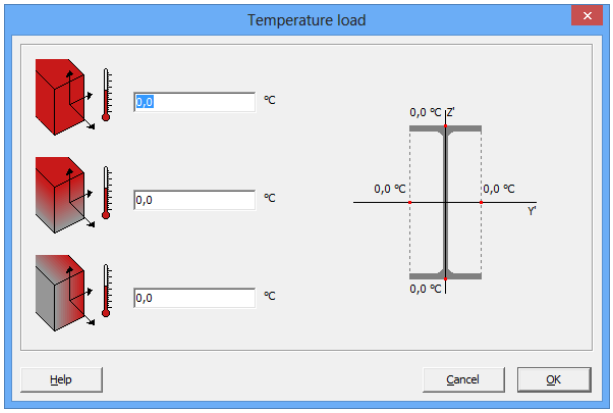
2.4.4. Adjusted definition of temperature loads

2.4.4.1. On bars

To add a temperature load on a bar, select the relevant bars and click on the icon . You can choose between:

- : a global temperature change (N)
- : a temperature gradient according to the local z'-axis ($M_{y'}$)
- : a temperature gradient according to the local y'-axis ($M_{z'}$)

The image on the right shows clearly how many degrees each sides warms up or cools down.

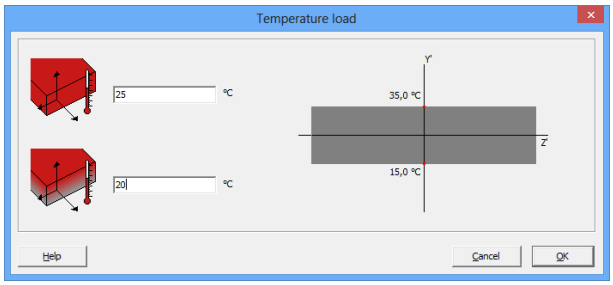


2.4.4.2. On surfaces

To add a temperature load on a surface, select the relevant surfaces and click on the icon . You can choose between:

- : a global temperature change
- : a temperature gradient according to the local y'-axis

The image on the right shows clearly how many degrees each sides warms up or cools down.



2.4.5. Lineair dynamic load

To be able to add dynamic functions to a load case, you need to have given the load case a dynamic character in $\gamma_g \gamma_u$, see §2.4.1.

The dynamic load case can have several sub load cases, which will be applied to the structure always together or separately .

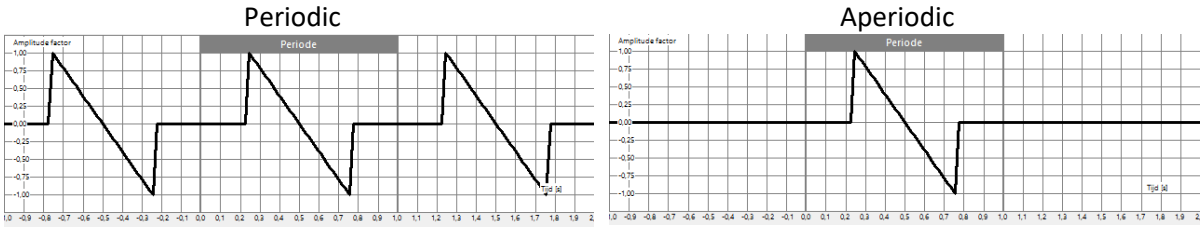
The definition of dynamic loads are consists of 3 steps:

1. : defining periodicity, synchronization and period of the (main) load case
2. Adding (static) loads for the load case or each of the sub load cases.
3. : shape of the wave, amplitude, phase,… of the (sub) load case(s)

2.4.5.1. Periodicity, synchronization and period

In this window you specify whether the (sub) load case(s) are

- Periodic (the wave repeats in time) or not (=aperiodic wave):



- Linked or not.

If there are no sub-load cases, only the periodicity applies. In case there are sub load cases in the dynamic load case, the synchronization is also important. This option is only relevant is the sub load cases are applied always together . If the sub load cases are working separately this option has no meaning. In case the sub loads work together and they are defined as 'asynchronously', Diamonds will find the most unfavorable situation (in this case period) between the sub load waves. If you choose 'synchronously', you need to know the time frame between the sub load waves and this needs to be constant. If you do not know the time frame between the sub load waves, or it does not remain constant, it is better to choose 'asynchronously'.

Next, you enter the period of the wave. In case of sub load cases, you need to enter the maximum period of all sub load waves. You can define the period in 2 ways:

- As an absolute value of the period(s) or the pulsation (rad/s)
  On entering one of both, the other one is automatically calculated.
- Relatively with respect to a chosen eigen period.

The period factor allows you to multiply the chosen eigen period with a value. The meaning here is that you can see the effect of a loading with a period close to (or equal) to the eigen period. You expect for this resonance of the structure, taking into account the damping of course.

If you click 'Advanced', you will find following options:

- **'# points'** is the number of points for which the displacements as a result of the loads per eigen mode are determined (= solving the kinematic equations). 100 points is a good choice.
- The larger the eigen frequency of a mode, the smaller the eigen period. To have also sufficient number of integration points for this kind of modes, you can specify a minimum number of points per wave. 15 is a good number.
- **'# evaluation points'** is the number of points where the response is calculated by means of super position. Having set '# evaluation points' and '#points' equal is the most accurate, but will take up the most time. You can set '# evaluation points' less than '# points', the mistake will be negligible. There is no point in setting '# evaluation points' higher than '# points'.
- If the sum of the effective modal masses is too small for one or more directions, you can proceed as follows:

  - You recalculate the modal analysis, however with more eigen modes.
  - You use the **quasi static correction.**

On the right side of the window you get an overview of all defined waves in the (sub) load case(s). Initially, when there are no waves defined yet, this graph will be empty. Later on, when you have defined all waves and you return to this window, you will see the combination of the defined waves. The active sub load case will be in bold.

2.4.5.2. Adding (static) loads for the load case or each of the sub load cases

For the load case or each sub load case, you need to define the maximum load, by means of a point load, a line loads, surface load, …

2.4.5.3. Shape of the wave, amplitude, phase, start and end

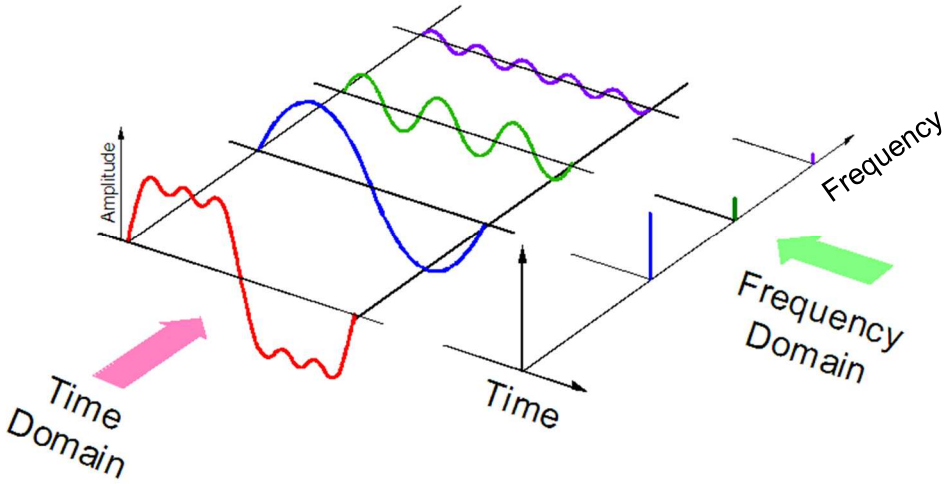
In this dialog window, you set for the load case or each of the sub load cases:

- Shape of the wave
  - Sinus (or harmonic) ,
  - Square (or impulse) ,
  - Sawtooth (or triangle or ramp) ,
  - Arbitrary (see 2.3.5.3)
- Amplitude factor: this is the maximum value of the wave, this is also the multiplication factor for the static load. The amplitude has to be in the range -1 and 1.
- Amplitude 2 (only for square or sawtooth): this is the maximum downwards value.
- N° period : with this option you specify how many times the cycle should run within 1 period.
- Phase (only for harmonic): to define a phase shift.

- 'Start': this is the start point of the wave, relative with respect to the period (start time = start * period). This value is always between 0 and 1. If this value is 0, the wave starts at the beginning of the period. In case of a value > 0, the wave will start later, there is a part of the period where the amplitude is 0 (a so called 'dead' part)
- Step (only for square or sawtooth): this is where the wave crosses the horizontal axis, so where the wave changes sign, the zero point of the wave.
- 'End': this is the end point of the wave, relative with respect to the period (end time = end * period). This value is always between 0 and 1. If this value is 1, the wave finishes at the end of the period. In case of a value < 1, the wave will end earlier, there is a part of the period where the amplitude is 0 (a so called 'dead' part)

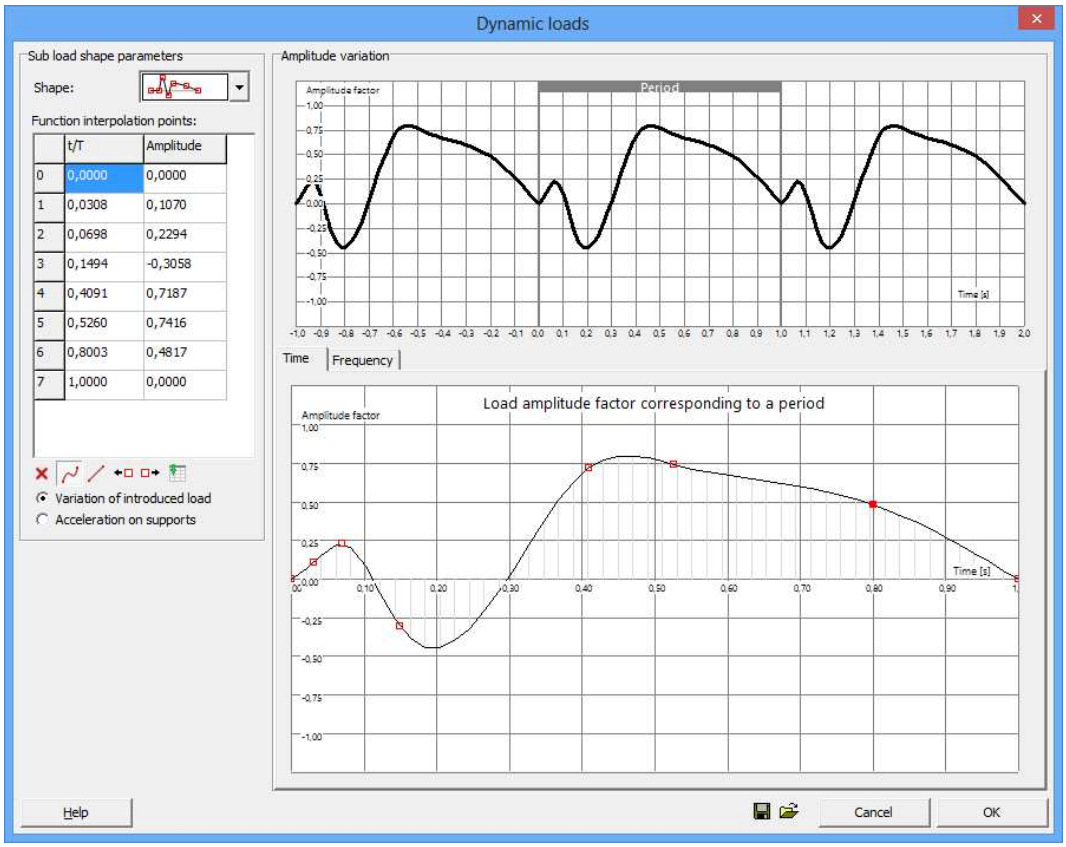
Finally you specify whether the wave must be applied to the (static) loads of the load case or as an acceleration of the supports. For this last option, you enter the value of the acceleration and the direction with the angles α and β.

On the right hand side, you will see the shape of the wave and the detail over 1 period. In the detail, you will have the possibility to show the wave in the (classic) time domain or in the frequency domain. By means of a Fourier transformation, every arbitrary wave (in red) can be described as a sum of harmonic (sinus and cosinus) functions (in red, green and purple). This decomposition can be displayed in the frequency domain. Here you can see which frequencies with which amplitude occur most in the wave. The frequencies with the highest amplitude will be most determinative and can be even critical, especially when they correspond with the eigen frequency of the structure.



2.4.5.4. Arbitrary wave

You can define an arbitrary wave in Diamonds. On selecting this option, you need to fill out some interpolation points. You can use the built-in Diamonds tools or past an external table with values.



- deletes a point
- 'fluent' interpolation between points. The points are being connected through a cubic spline.
- linear interpolation between points. The points are being connected through straight.
- adds a point, before the current and half way with the previous one.
- adds a point, after the current and half way with the next one.
- pastes an external table with value from the clipboard. In an external table (for example MS Excel), you need to have 2 columns: one with the time/period values and a second one with the amplitudes. Select both columns with values and copy these to the clipboard (via CTRL+C) and paste the values in Diamonds with .

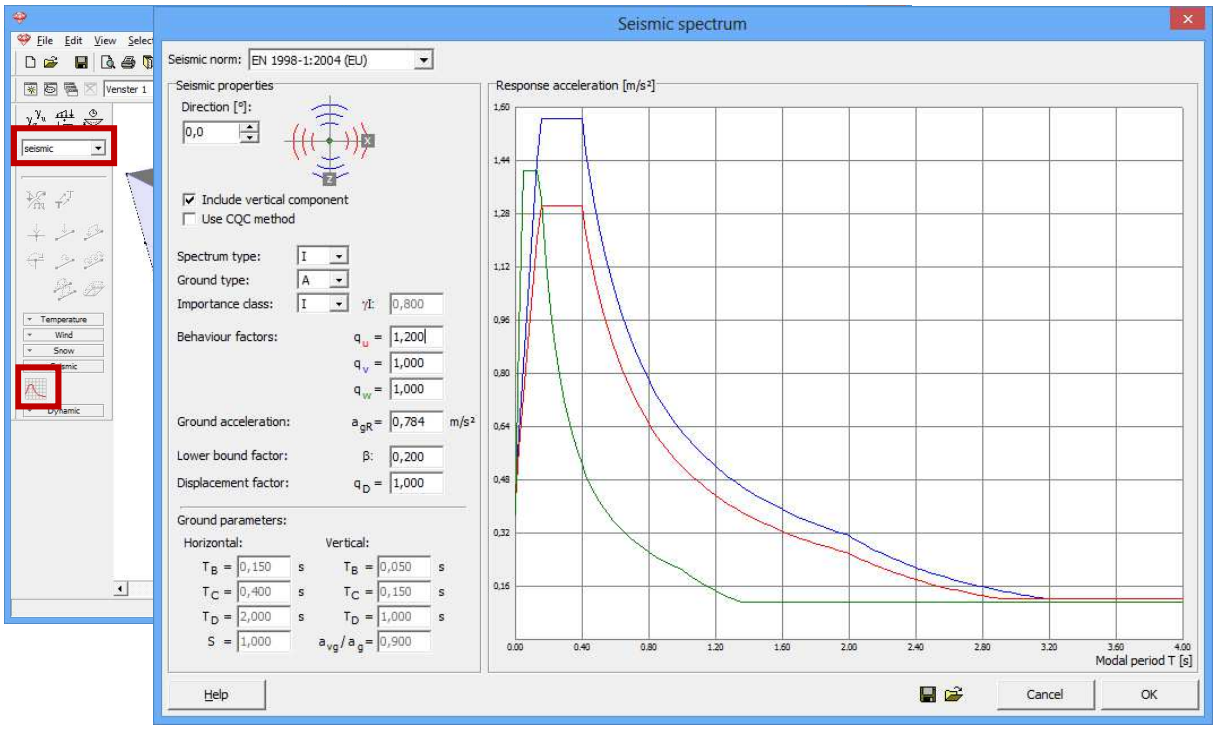
Next to these tabular operations, you can also pick points with the cursor in the detailed graph and move them as you please.

Remark:

- On importing a Diamonds model with seismic and/or dynamic loads in PowerFrame or vice versa, we strongly recommend to redefine these load after importing the model.

2.4.6. Seismic load

When you have created a seismic load case, you can define the seismic spectrum via . You will get following screen:

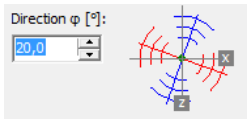


- Specify the design code for which you wish to generate the seismic spectrum. You can choose between:
  - EN 1998-1:2004 (EU)
  - ASCE (USA)
  - INPRES-CIRSOC (ARG)
  - NSR-10 (COL)
  - NCH433 (CHL)

  In the above illustration EN 1998-1:2004 was chosen.

- A seismic spectrum consists of a
  - Horizontal spectrum in the u direction
  - Horizontal spectrum in the v direction
  - Optional vertical spectrum in the w direction

  The response acceleration in function of the modal period is given for each spectrum (u, v and w direction) in the graph.



- Enter the angle between the direction of the spectrum and the global axes. The image shows how the spectrum is rotated with respect to the **global** axes.
  If the angle is 0°, the u and v-direction coincide respectively with the global X and Z direction.
- Select whether or not you want to take into account the vertical component.
- If the eigen modes are *independent* of each other, it is allowed to calculate the total response with the 'sum of the squares' method. If the eigen modes are dependent of each other, you need to use Complete Quadric Combination-method (CQC- method).
- Choose the spectrum type. You have the choice between type I or II.

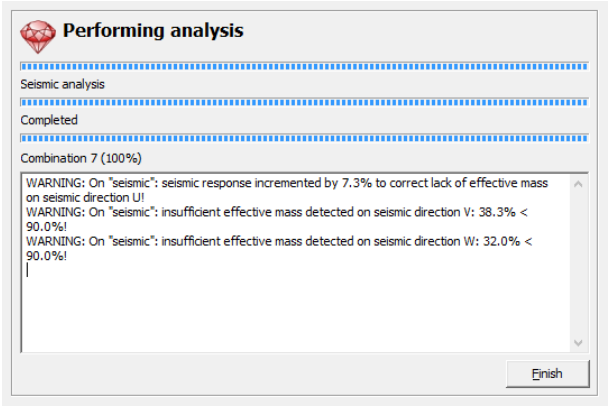
- Choose the soil type. You have the choice between types A → E .
- Choose the building's important class. You have the choice between I until IV. Depending on the important class, Diamonds will fill out a importance factor γ_I for you.
  If you choose '—', you will need to enter yourself a value for the importance factor γ_I.
- Enter the behaviour factors per direction (q_u, q_v, q_w).
- Ground acceleration
- Displacement factor

More information about these parameters can be found in the design codes, such as EN 1998-1:2004.

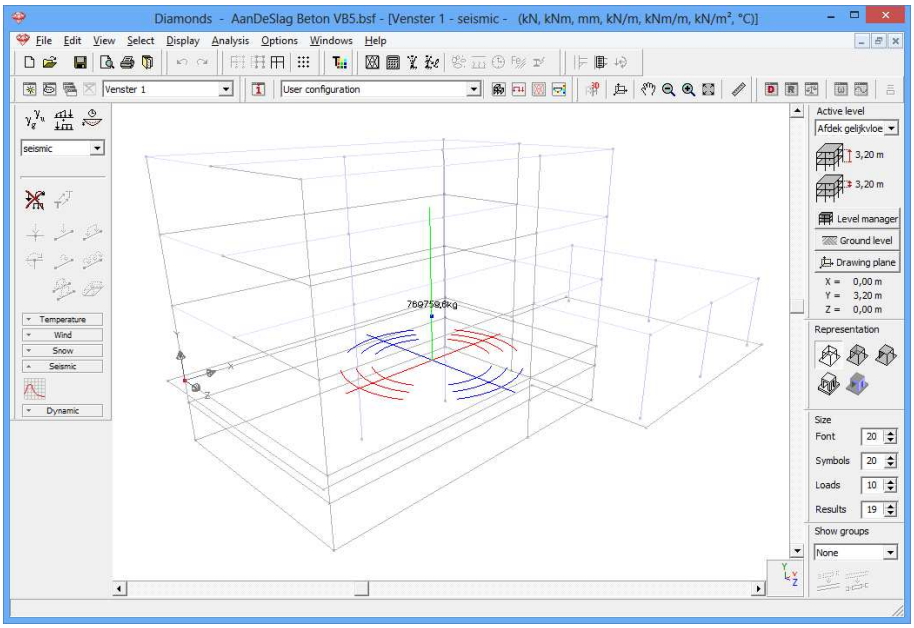
You can save a defined spectrum in an external TXT-file, by using the button on the bottom of the window. Saved spectrums can be opened via .

Remarks:

- A seismic action is considered to act on all supports simultaneous.
- A seismic action can be removed with the 'Delete loads' button
- There is no quasi-static correction for seismic loads. If you have not obtained sufficient effective modal mass, you'll see this error.



  You need to recalculate with a larger number of eigen modes.
- A seismic load is represented by the symbol below. This symbol is always displayed in the XZ plane. De mass that is taken into account in the modal analysis is shown in the center of gravity of the structure.



2.4.7. Masses related to vertical loads

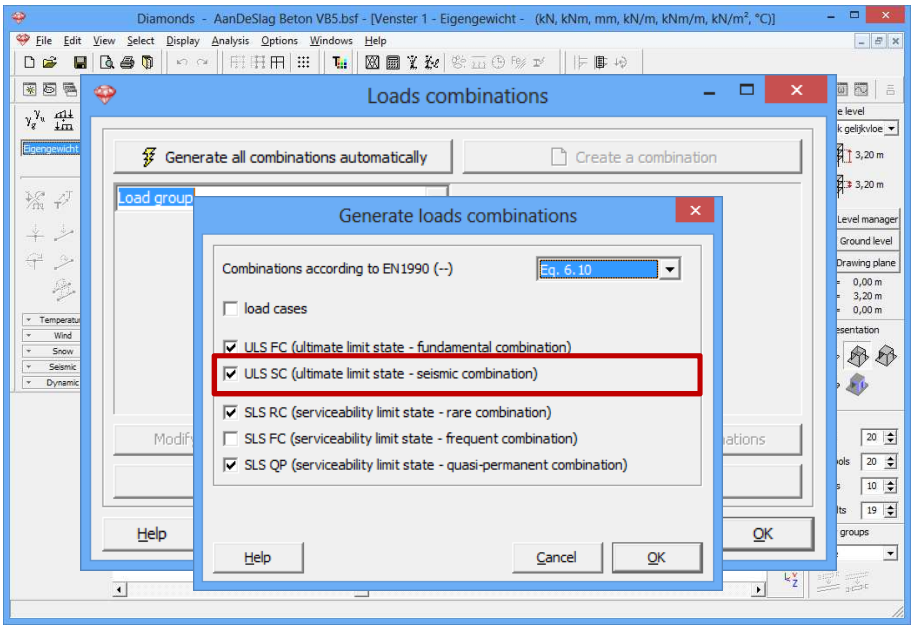
The modal parameters of a structure (eigenmodes and - frequencies) are calculated based on the stiffness and mass distribution within the structure.

The stiffness of the structure only depends on the geometry of the structure itself.

But for the mass distribution, masses are related to the vertical loads (point loads, line loads, surface loads) (0,981kN -> 100kg) using the correlation coefficient φ (and also multiplied by the combination factor ψ_2 for what EN 1990-1-1 is concerned).

Diamonds will set the correlation coefficient to 1 for all load groups, except for temperature loads an dynamic loads (set to 0). For standard in which no ψ_2 is mentioned, the correlation coefficient of only the self weight and the dead loads is set to 1.

2.5. Load combination generation



Due to the presence of seismic loads, you can now generate a combination ULS SC – ultimate limit state seismic combination.

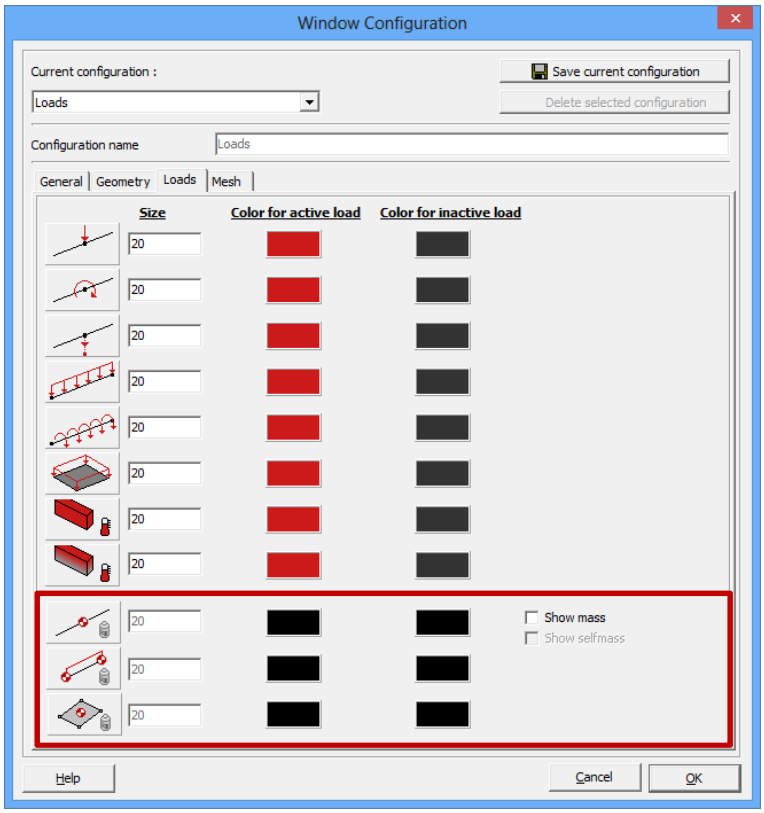
2.6. Extended load configuration

In this tab you enter (like the previous versions of Diamonds):

- how large the different load types should be represented.
- in which color the different load types should be represented.

Below you'll see that 3 load types have been added, because of the dynamic/seismic module in Diamonds 2013, namely:

- concentrated mass in a point
- distributed mass on a line
- distributed mass on a surface


Window Configuration
Current configuration :
Loads
Save current configuration
Delete selected configuration
Configuration name
Loads
General
Geometry
Loads
Mesh
Size
Color for active load
Color for inactive load
20
Show mass
Show selfmass
Help
Cancel
OK

3. Elastic analysis

3.1. New calculation core

Diamonds 2013 comes with a new more efficient calculation core. We review:

- Capable of using all the available cores of your system.
- Results will no longer be held in the RAM memory, but written to the hard drive.
- Use 64 bit processes when the operating system supports this.

Remarks:

- Installation of .NET Framework 4.0 and CodeMeter Control Center version 5.0 is required.
- A dialog window will show you the analysis progress. This window will disappear automatically when the analysis is completed without errors or warnings. If there were errors or warning during analysis, this dialog box remains open on the screen.
  With the button 'Details' you get an overview of the calculation processes and errors. After you take note of the error, you can close the window by clicking 'Close'.
- A file calculated and saved in Diamonds 2013 will contain 3 parts:
  - A *.bsf-file with geometry, loads and mesh.
  - A *bs!-file which is a copy of the *bsf-file (= a back up)
  - A folder with the same name as the .bsf-file, where results are stored.

3.2. Modified window for elastic analysis

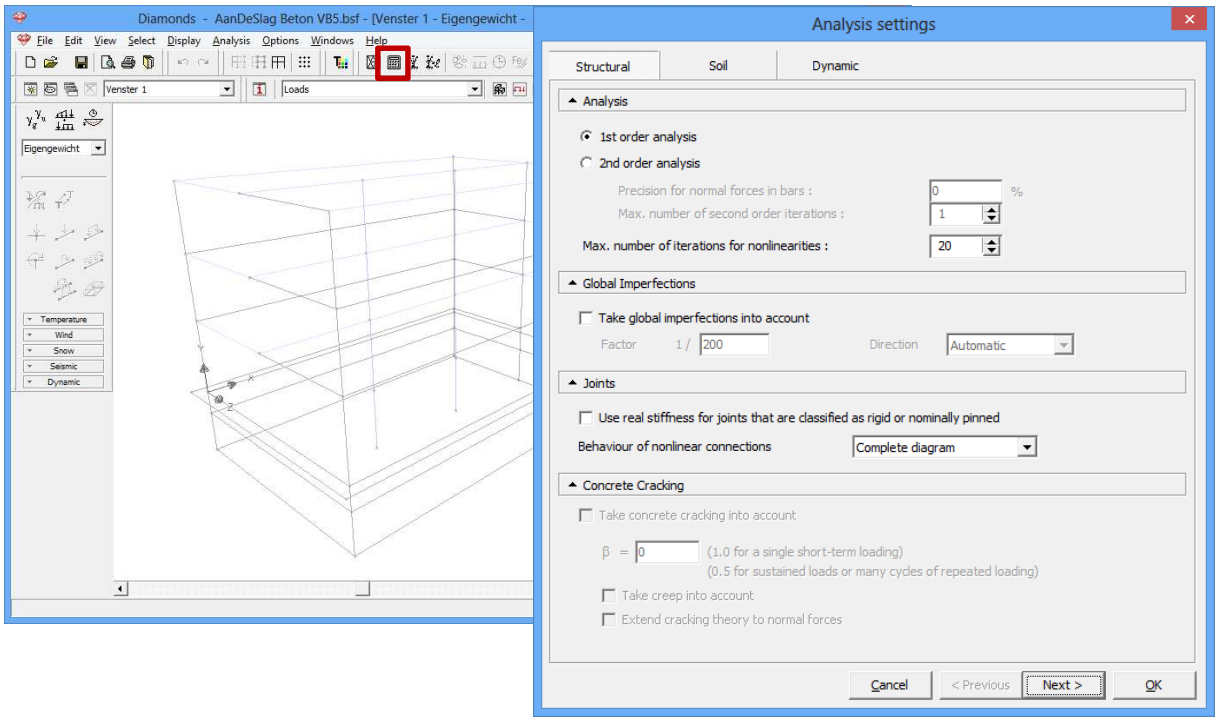
The elastic analysis window has 3 tab pages as from Diamonds 2013:

- Structural
- Soil
- Dynamic

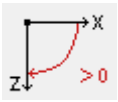
Depending on the geometry, one or more tab pages will be active. You need to go over all active tabs.

3.2.1. Structural

The tab 'Structural' looks like this:

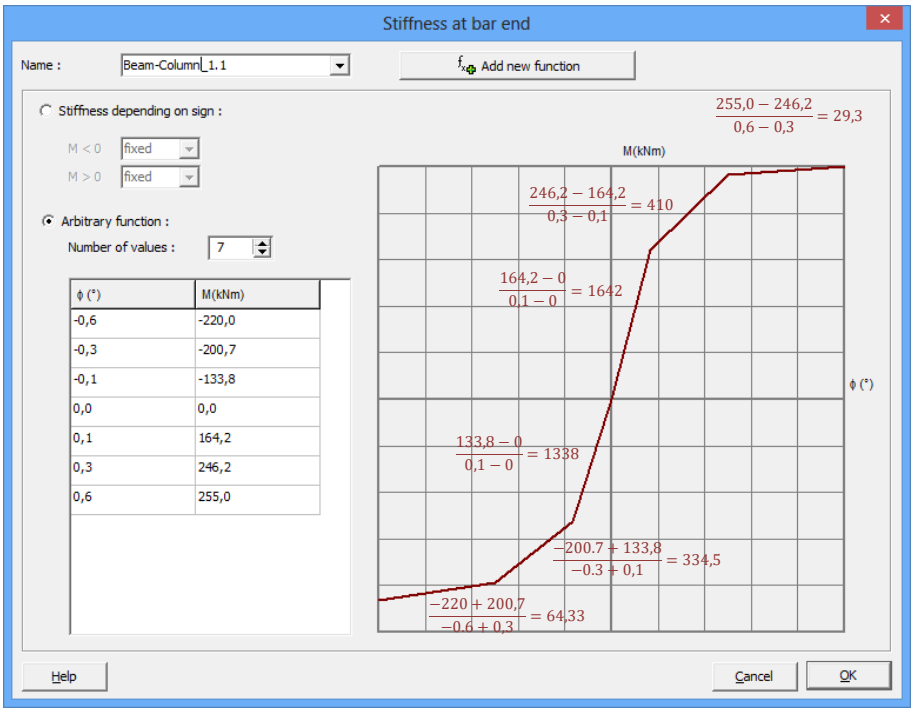


- Analysis
  New here is the possibility to impose a maximum number of iterations for non-linearities. The other options are the same as before.
- Global imperfections
  You can take the global imperfections into account by checking this box. You need to enter the factor and direction. The direction option is new and can be:
  - Automatically set by Diamonds – this will be the direction with the greatest vectorial horizontal displacement.
  - Global X
  - Global Z
  - Custom – you need to enter an angle in the horizontal plane

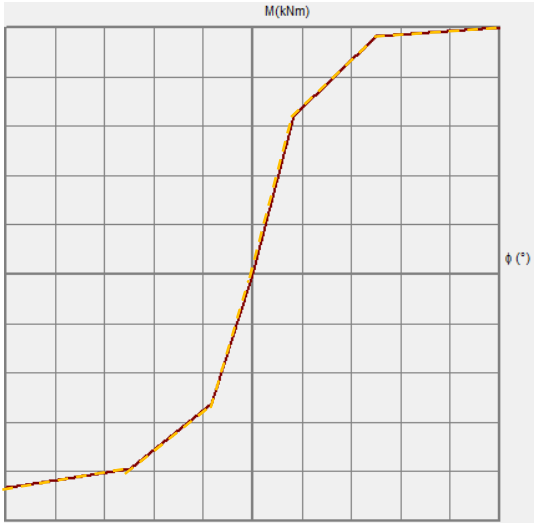
- Connections
  Here you can decide whether the analysis should be done based on the connection's classification or on the connection's real stiffness.
  New in Diamonds 2013 is the choice of the behavior of non-linear connections:
  - Take into account the full stiffness diagram.
  - Avoid plasticity (the plastic part of the stiffness diagram)
  - As limit value you enter the slope [-] of the part of the diagram from which you want to have elastic behavior again. With a negative slope value, Diamonds will take into account all plastic parts of the diagram.
  - Take into account only the initial stiffness S_j
  - Take into account only the initial stiffness $S_{j,ini}$

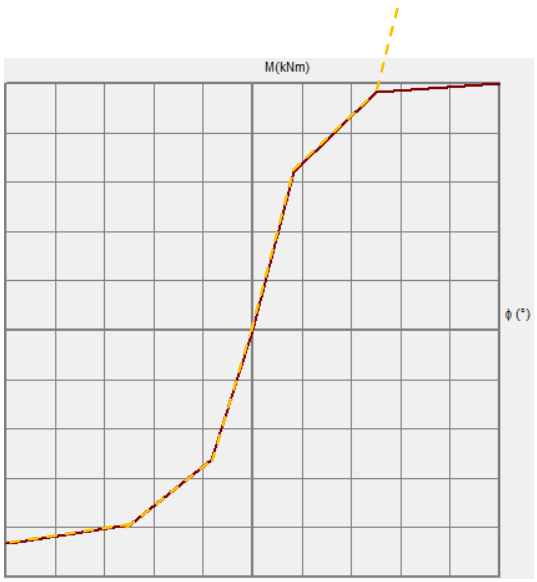
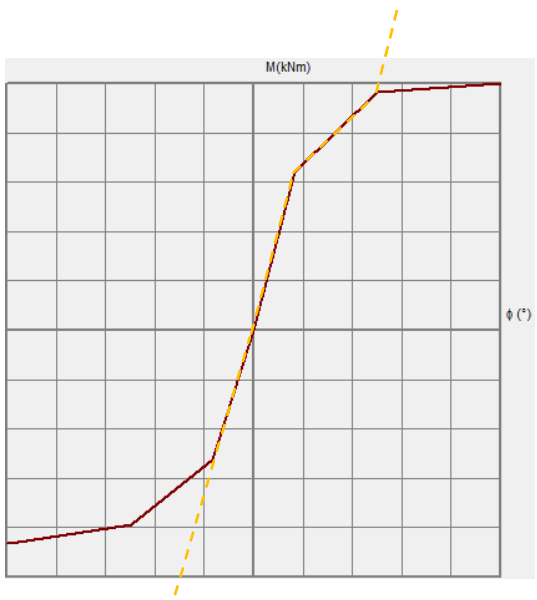
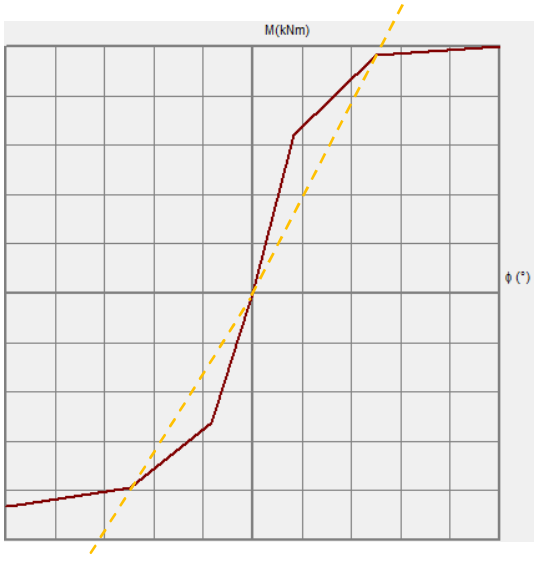
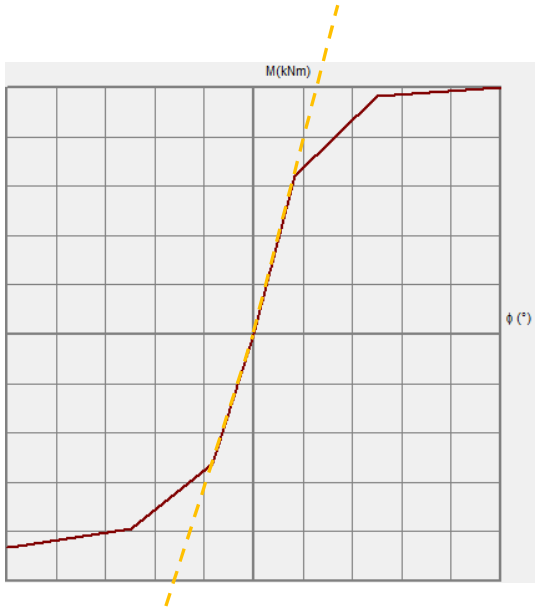
  Example: for the stiffness diagram below, we calculated all the slopes for each part of the diagram, based on the table values on the left.



When we choose one of following options, Diamonds will take the orange dashed stiffness diagram into account, instead of the original red one.

Full stiffness diagram



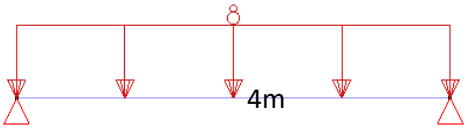
Avoid plasticity – **limit 30**	Avoid plasticity – **limit 350**
	
The stiffness S_j	The initial stiffness $S_{j,ini}$
	

- Concrete Cracking
  The parameters keep their known functions.

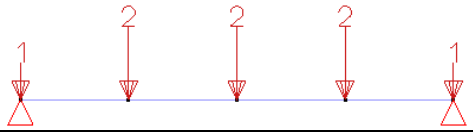
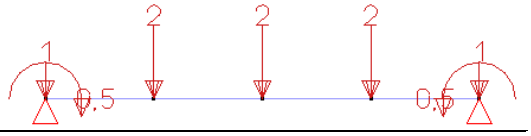
Remarks:

- The 1st order analysis as from Diamonds 2013 can produce a slightly changed result, in comparison with previous versions.

Example



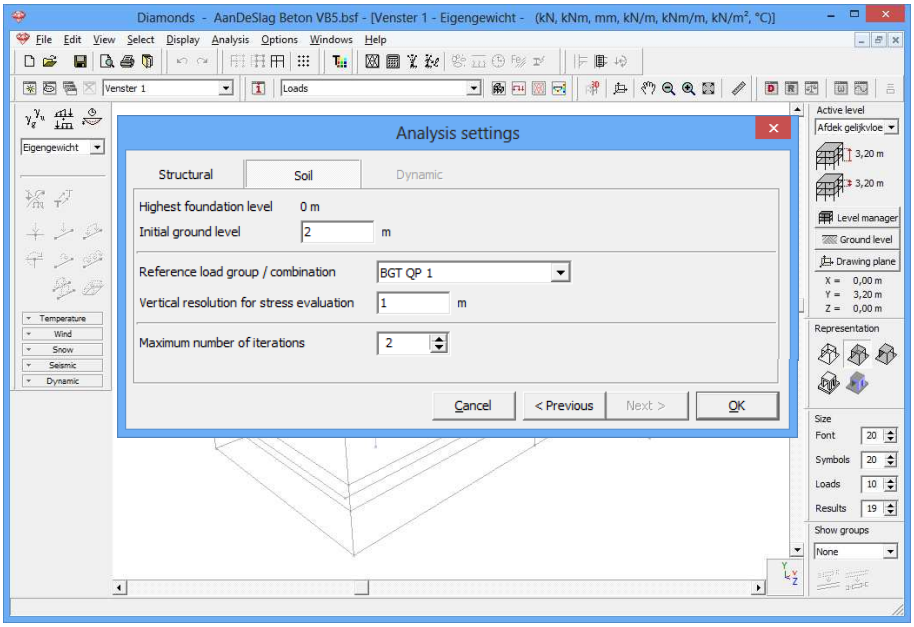
Diamonds makes the FEM mesh (let's assume 4 internal bar divisions) and will recalculate the line load to point loads and put them in the mesh nodes (= so called 'discretization'). This schematization will be used by Diamonds to calculate the model and looks like this in the different versions:

Up to Diamonds 2012 R02	As from Diamonds 2013
	
The points loads of '1kN' do not cause any deformation anymore. Therefore, previous versions of Diamonds could have a small deviation in deformation (and in the consequently internal forces).	In the new Diamonds we internally add correcting moment to fix this 'bug' as a result of using the Finite Element Method.

- Also the 2nd order analysis can produce another result as from Diamonds 2013, compared to the previous Diamonds versions. The 2nd order analysis is extended in a way that:
  - Moments are taken into account in the stiffness matrix.
  - The internal forces are calculated based on the deformed state of the structure.

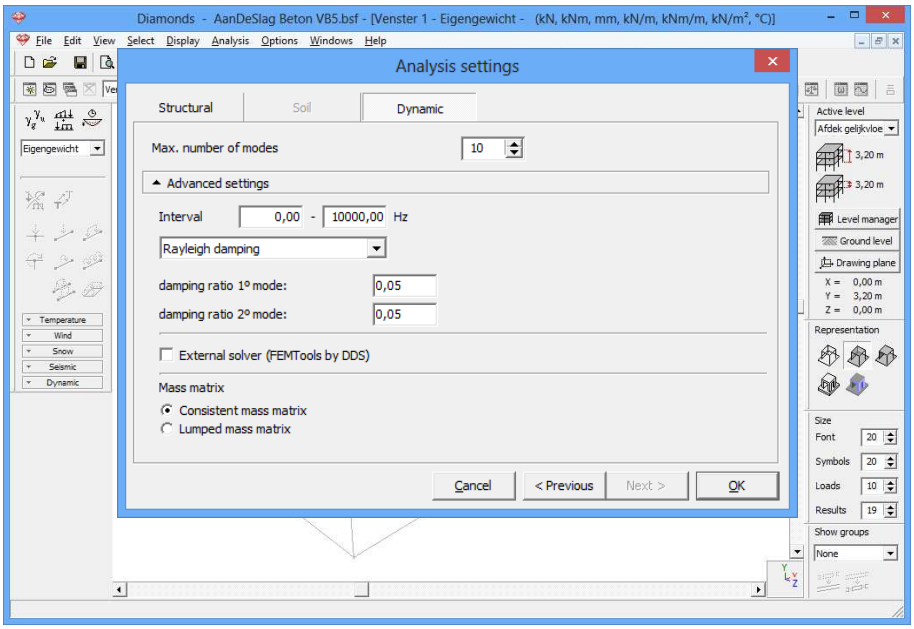
3.2.2. Soil

The parameters of this tab were previous shown in a separate dialog window after launching the elastic analysis. The meaning of these parameters remain identical.



3.2.1. Dynamic

The tab page ‘Dynamic’ will only be active when the model contains a dynamic or seismic load.



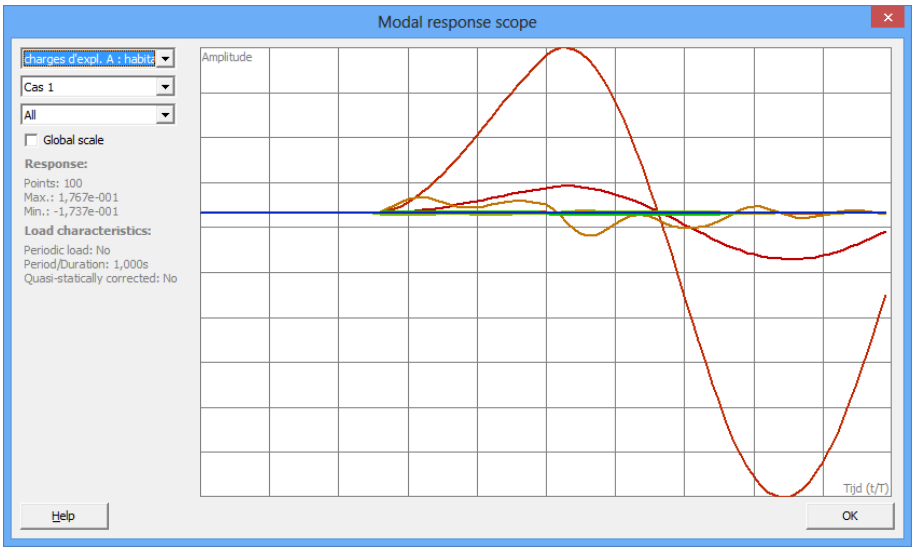
This part has following parameters:

- Specify the number of eigen modes (and accompanying eigen frequency) you want to calculate.
- Also select based on which reference load combination you want Diamonds to determine the stiffness matrix. A good choice here is ‘Self weight’.
- Advanced settings:
  - The ‘Interval’ option lets you set the frequency range within the eigen frequencies should be calculated..
  - You can choose between 2 types of damping:
    - Rayleigh damping: enter the damping ratio ξ of the first and second eigen model.
    - Or you enter the measured dampingsratios for each eigen modes separately.
  - The parameter ‘Maximum sub iterations’ can be left on ‘auto’. In case of a convergence problem you may obtain convergence by increasing this parameter.
  - If you have the Diamonds ‘Linear Dynamic Analysis’ module, you can choose here between the internal Diamonds solver or the efficient external DDS solver (recommended).
  - Inidicate whether you would like to you a consistant mass matrix or one based otn eh principle of lumped mass.

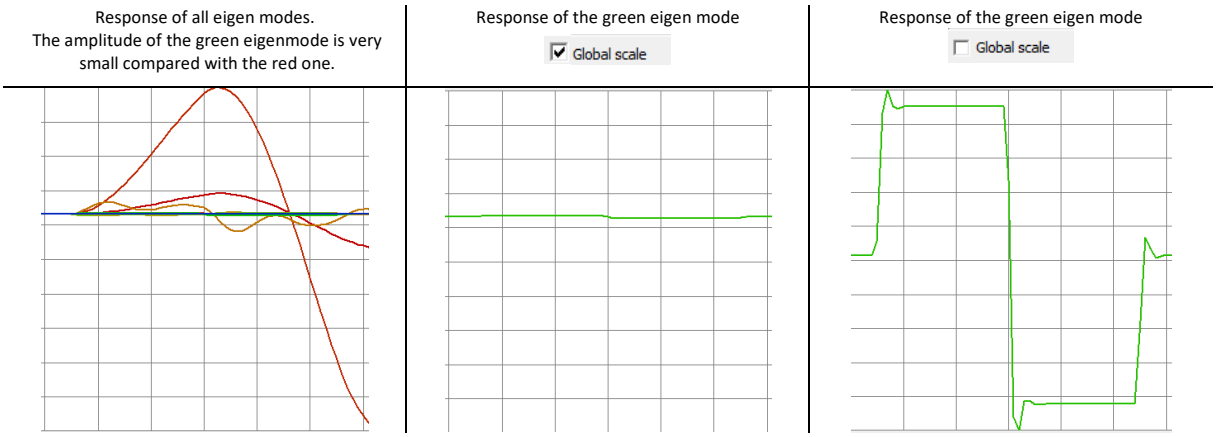
After the dynamic analysis you can explore different results:

- For each eigen frequency the accompanying eigen mode and results (method see §0).
- A table with the modal information (see §0).

- For each eigen mode the response on the dynamic loading .



- In the **first pull down menu**, you choose the dynamic load case for which you would like to see the response of the structure.
- If this dynamic load case has sub-load cases, you can select the appropriate sub-load case in the **second pull down menu**.
- In the **third pull down menu** you choose the eigen modes to be displayed. You can also show the responses of all eigen modes in 1 graph.
- When you select the option **'Global scale'**, the amplitude of the eigen mode will be rescaled with respect to the maximum amplitude. If you uncheck this, the response of eigen mode will be displayed as large as possible. Example:



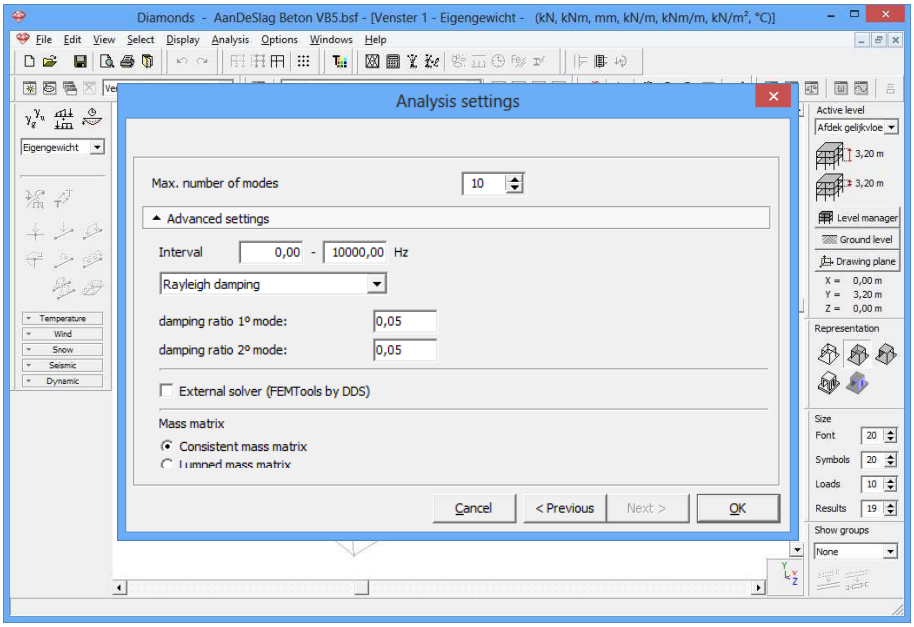
This window helps you to

- Determine which eigen mode reacts a lot or not to a certain dynamic loading.
- Decide whether a quasi-static correction is justified or not.
  If the eigen frequency of the mode is much larger than the frequency of the imposed loading, the mode will react in a 'quasi statical' way – the amplitude has the same shape of the defined loading. In that case, you can use the quasi-static correction.

The response's time frame is identical to the period you have filled out in (see §0.

3.3. Modal analysis

A modal analysis will calculate the eigen frequencies and eigen modes of a structure. In order to start this analysis, you click the button. Diamonds shows you a dialog window where you can set some analysis options:

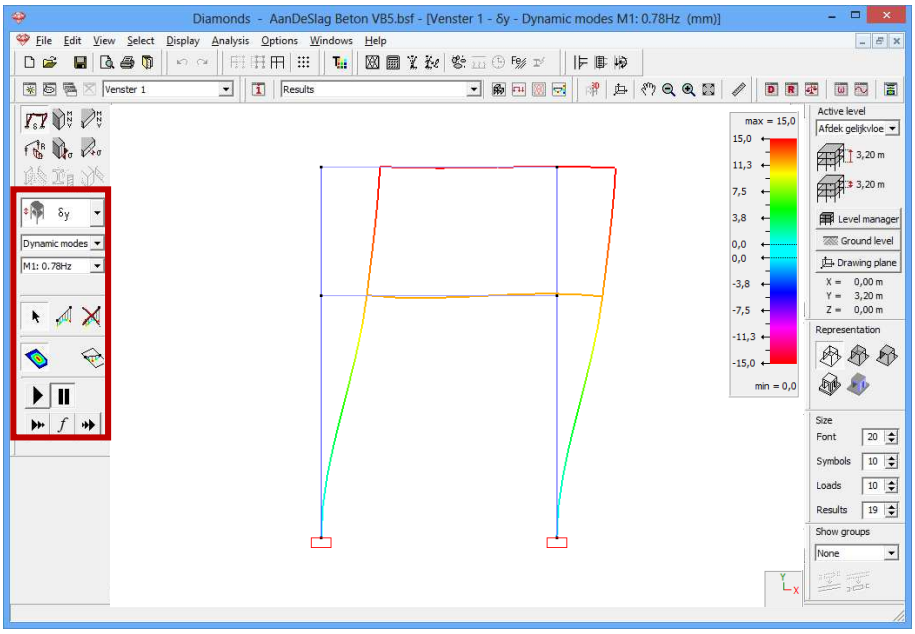


This window has following parameters:

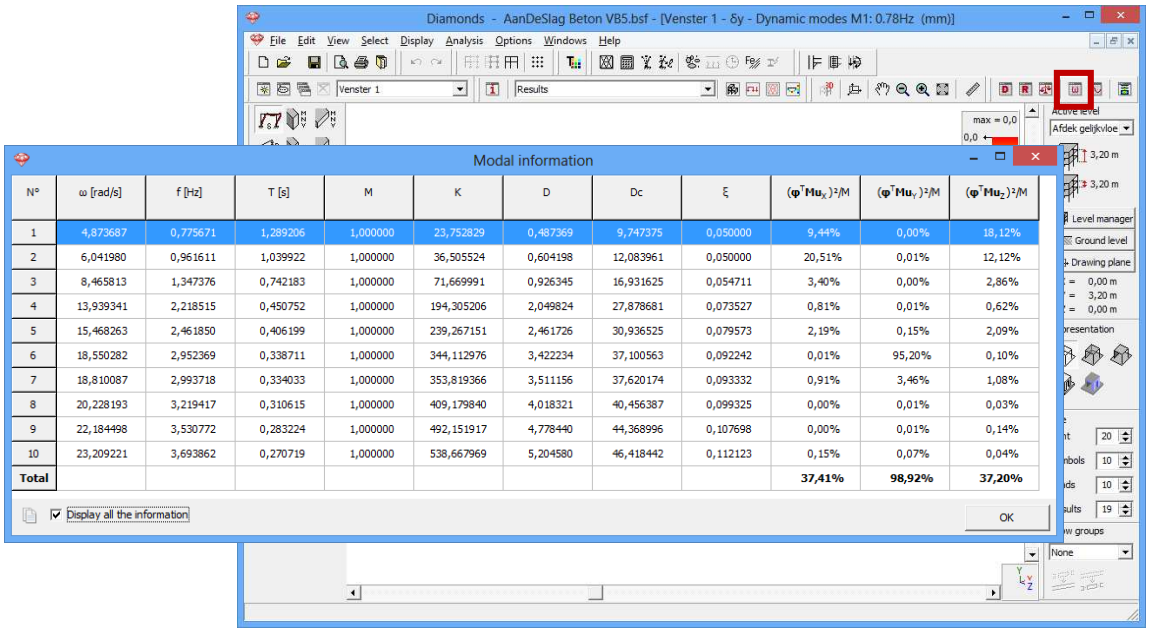
- Specify the number of eigen modes (and accompanying eigen frequency) you want to calculate.
- Advanced settings:
  - The 'Interval' option lets you set the frequency range within the eigen frequencies should be calculated..
  - You can choose between 2 types of damping:
    - Rayleigh damping: enter the dampingsratio ξ of the first and second eigen model.
    - Or you enter the measured dampingsratios for each eigenmodes separately.
  - The parameter 'Maximum sub iteraties' can be left on 'auto'. In case of a convergence problem you may obtain convergence by increasing this parameter.
  - If you have the Diamonds 'Linear Dynamic Analysis' module, you can choose here between the internal Diamonds solver or the efficient external DDS solver (recommended).
  - Inidicate whether you would like to you a consistant mass matrix or one based otn eh principle of lumped mass.

After having performed a modal analysis, you can look at the results per eigen frequency (and accompanying eigen mode). Go to the 'Results' window and select for example the deformations . In the pull down menu below, you can select one of the eigen frequencies. You can animate the movement of the structure with :

- : play
- : pauze
- : speed up
- : slow down
- : animate with the speed of the eigen frequency.



With the button you access the table with modal information:



N°	ω [rad/s]	f [Hz]	T [s]	M	K	D	Dc	ξ	$(\varphi^T Mu_X)^2/M$	$(\varphi^T Mu_Y)^2/M$	$(\varphi^T Mu_Z)^2/M$
1	4,873687	0,775671	1,289206	1,000000	23,752829	0,487369	9,747375	0,050000	9,44%	0,00%	18,12%
2	6,041980	0,961611	1,039922	1,000000	36,505524	0,604198	12,083961	0,050000	20,51%	0,01%	12,12%
3	8,465813	1,347376	0,742183	1,000000	71,669991	0,926345	16,931625	0,054711	3,40%	0,00%	2,86%
4	13,939341	2,218515	0,450752	1,000000	194,305206	2,049824	27,878681	0,073527	0,81%	0,01%	0,62%
5	15,468263	2,461850	0,406199	1,000000	239,267151	2,461726	30,936525	0,079573	2,19%	0,15%	2,09%
6	18,550282	2,952369	0,338711	1,000000	344,112976	3,422234	37,100563	0,092242	0,01%	95,20%	0,10%
7	18,810087	2,993718	0,334033	1,000000	353,819366	3,511156	37,620174	0,093332	0,91%	3,46%	1,08%
8	20,228193	3,219417	0,310615	1,000000	409,179840	4,018321	40,456387	0,099325	0,00%	0,01%	0,03%
9	22,184498	3,530772	0,283224	1,000000	492,151917	4,778440	44,368996	0,107698	0,00%	0,01%	0,14%
10	23,209221	3,693862	0,270719	1,000000	538,667969	5,204580	46,418442	0,112123	0,15%	0,07%	0,04%
Total									**37,41%**	**98,92%**	**37,20%**

You will find following parameters:

- angular velocity ω $[rad/s]$
- frequency f $[Hz]$
- period T $[s]$
- modal mass M
- modal stiffness K
- modal damping D
- critical damping D_c
- damping ratio ξ $[\%]$
- effective modal mass according to the global X-axis
- effective modal mass according to the global Y- axis
- effective modal mass according to the global Z- axis

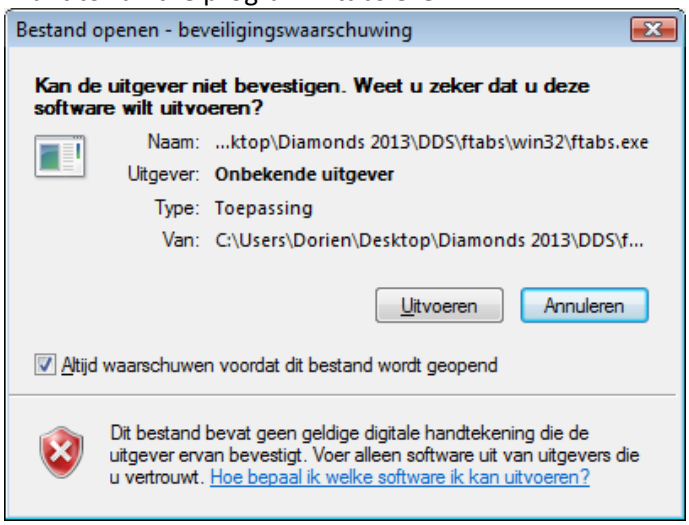
Use the button to copy the contents of this window.

Remarks:

- Diamonds has 2 solvers to perform a modal, dynamic or seismic analysis:
  - An internal solver, following the Lanczos method.
  - An external and highly efficient DDS solver. DDS (Dynamic Design Solutions) is a Belgian company specialized in dynamic solutions.

- Solver access:
  - If you have a license of the new Diamonds 'Seismics' or 'Lineair Dynamic Analysis' module OR you have separately acquired these in the past, you will have access to both solvers (both internal and external solver).
  - If you had in the past a PowerFrame license, you now can calculate eigen modes and eigen frequencies for frame structures in Diamonds (modal analysis).
  - If you have doubts about your license, feel free to contact us.

- On executing a modal, dynamic or seismic analysis with the DDS solver, Windows will ask you if you really want to run the program ‘ftabs.exe’:



  You need to click ‘Run’ in order to proceed with the analysis.

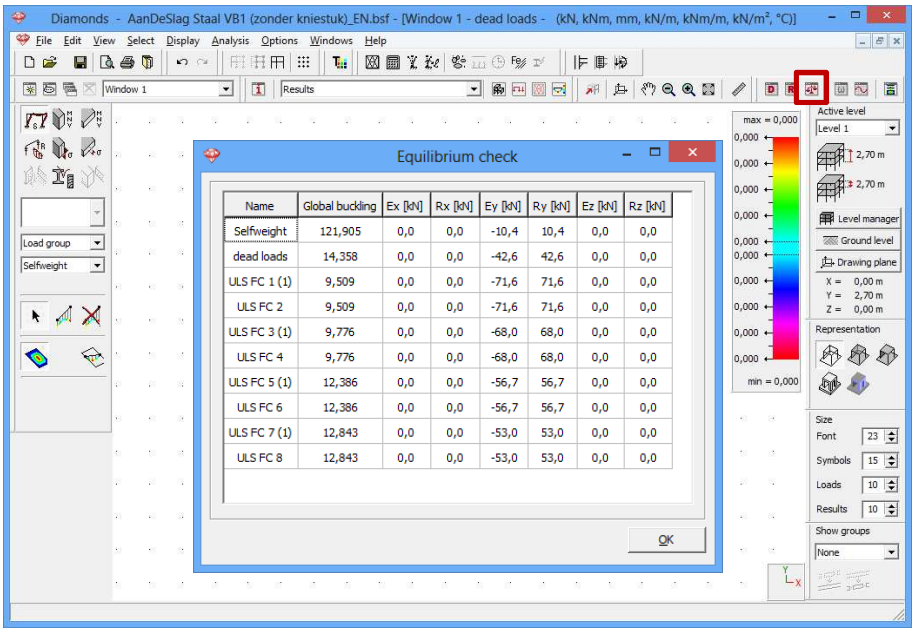
- De eigenmodes in Diamonds are mass [kg] normalized, in PowerFrame they are normalized to the mass in tonne.

3.4. Equilibrium check and global buckling factor

The button in the icon bar has both the equilibrium check and the global buckling factor, for each load combination:

- The **equilibrium check** contains for each load combination an overview of the applied horizontal a vertical loads and the consequential horizontal and vertical reaction forces..
- The **global buckling factor** is the factor by which the loads have to be multiplied in order to provoke global buckling of the structure.
  If the global buckling factor is less than ‘1’, the loading is more than the critical loading, and therefore the structure will fail (collapse) as a whole.
  The global buckling factor is only available after a second order calculation.
  The global buckling factor is independent of the standard and is not to be confused with buckling check of a single element.

Use the button to copy the contents of this table.



Remarks:

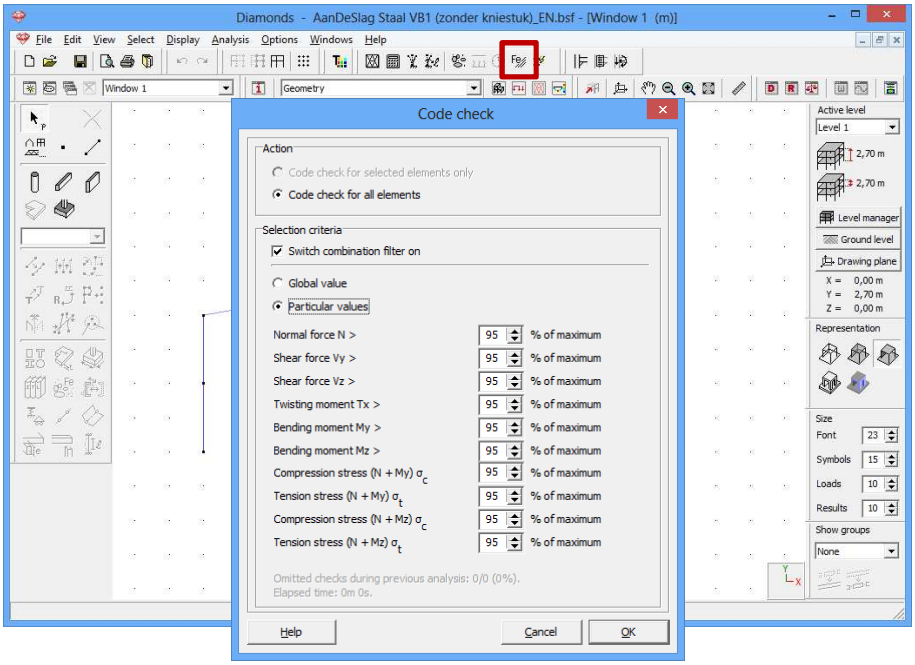
- The equilibrium check was available as from Diamonds 2012 R00.
- Global buckling is design code Independent and is not to be confused with the buckling stability of one single element.

4. Design

4.1. Extended filter on steel and timber verification

Until now, you could set a global or individual criterion for internal forces (M, V, N), in order to only take the relevant combinations into account for the steel and timber verification.

This selection criterion is now extended to the elastic stresses in the bars.

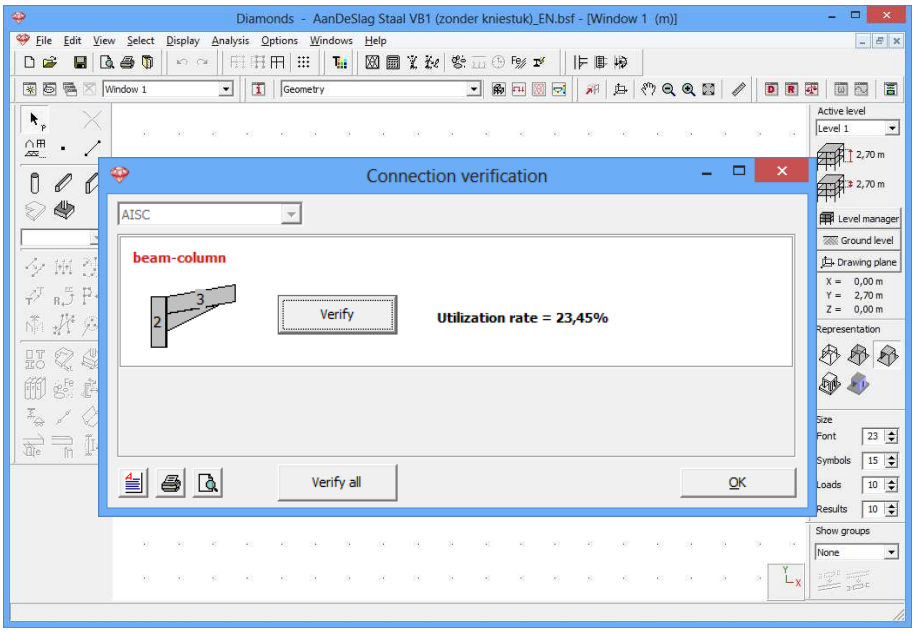


4.2. Extended connection verification (according to AISC)

On calculation connections, the stiffness diagram is transferred to Diamonds. The presence of this diagram will influence the stiffness of the global structure, and therefore also the internal forces results. Theoretically, you should then recalculate the connection in PowerConnect, with the new found internal forces.

For connections according to AISC, this recalculation is now implemented in Diamonds 2013 :

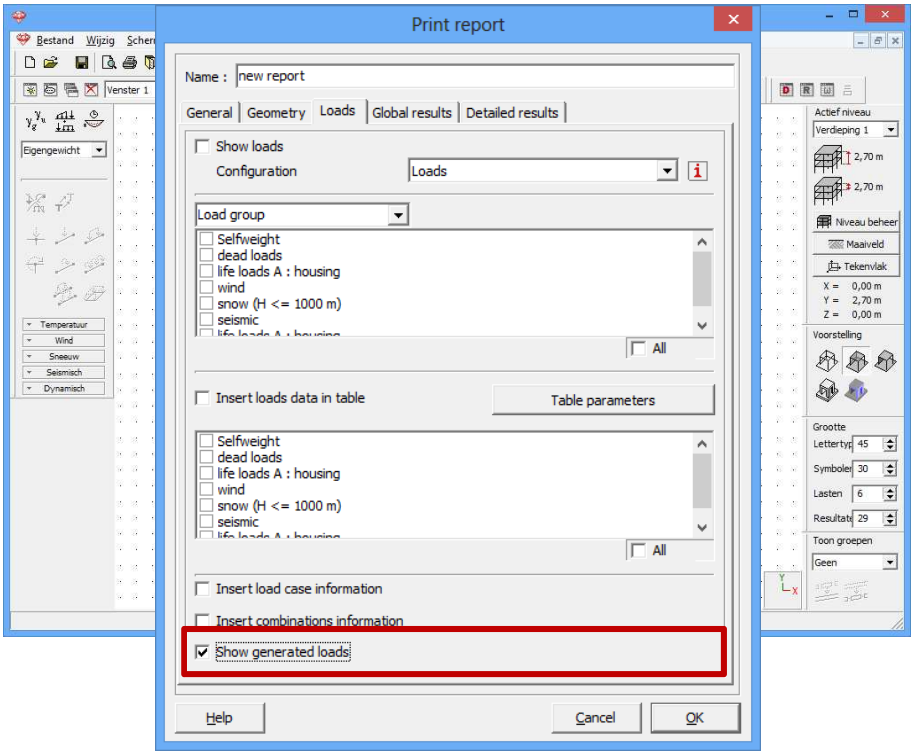
- After recalculation the structure and having the new internal forces, you click the .
- Diamonds offers you the possibility to recalculate each connection separately or all connections at once.
- Diamonds will ask you for which combinations the connection should be checked.
- After the calculation (behind the scenes in PowerConnect), the loading level [%] will be displayed on the screen for you. The connection is still sufficient if you get a percentage below 100%. If the connection is no longer ok, you should modify the connection's geometry in PowerConnect.

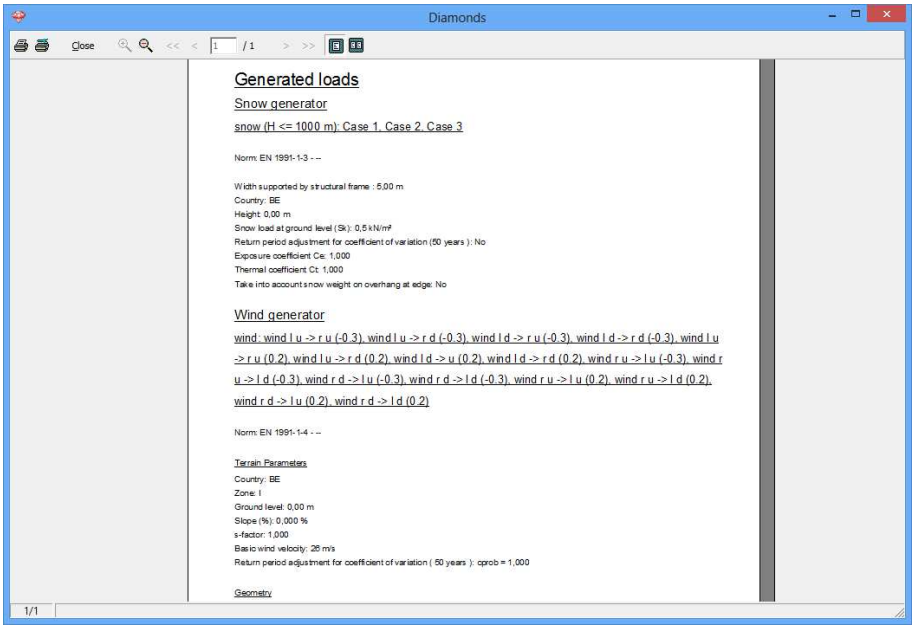

Diamonds - AanDeSlag Staal VB1 (zonder kniestuk)_EN.bsf - [Window 1 (m)]
File Edit View Select Display Analysis Options Windows Help
Window 1
Geometry
Active level
Level 1
2,70 m
2,70 m
Level manager
Ground level
Drawing plane
X = 0,00 m
Y = 2,70 m
Z = 0,00 m
Representation
Size
Font 23
Symbols 15
Loads 10
Results 10
Show groups
None
Connection verification
AISC
beam-column
2
3
Verify
Utilization rate = 23,45%
Verify all
OK

5. Report

5.1. Terrain parameters wind and snow

In Diamonds 2013 you can report the terrain parameters of the by Diamonds generated wind and snow loads. In order to do this, you need to check the box ‘Include generated loads’.



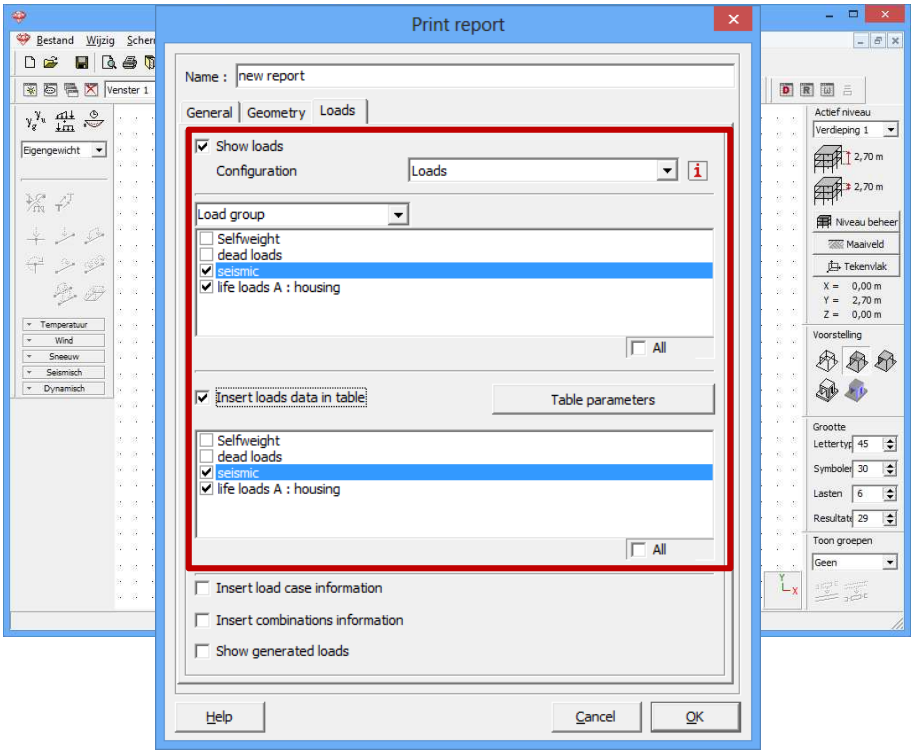


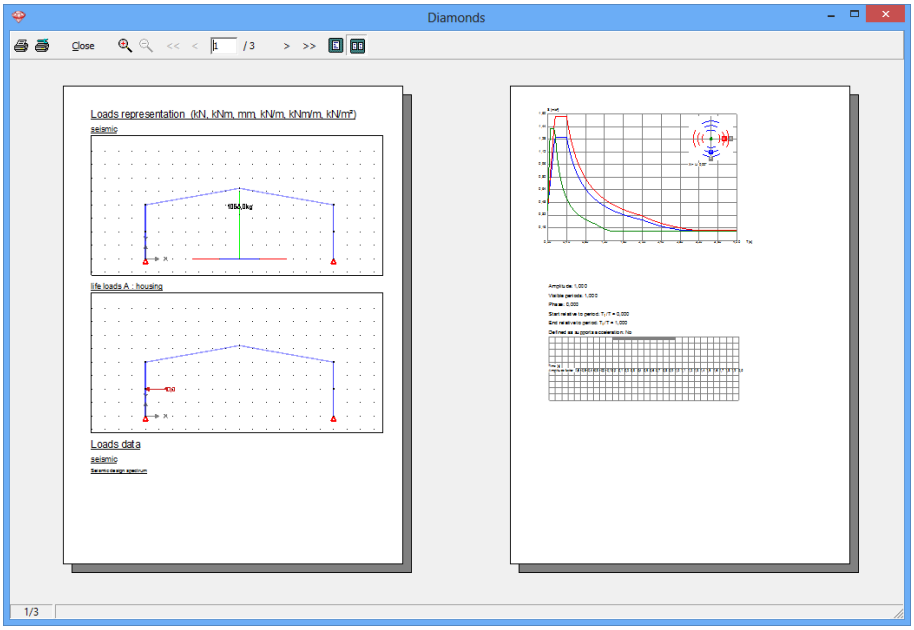
Remark:

- This option will only work for wind or snow loads generated in Diamonds 2013.

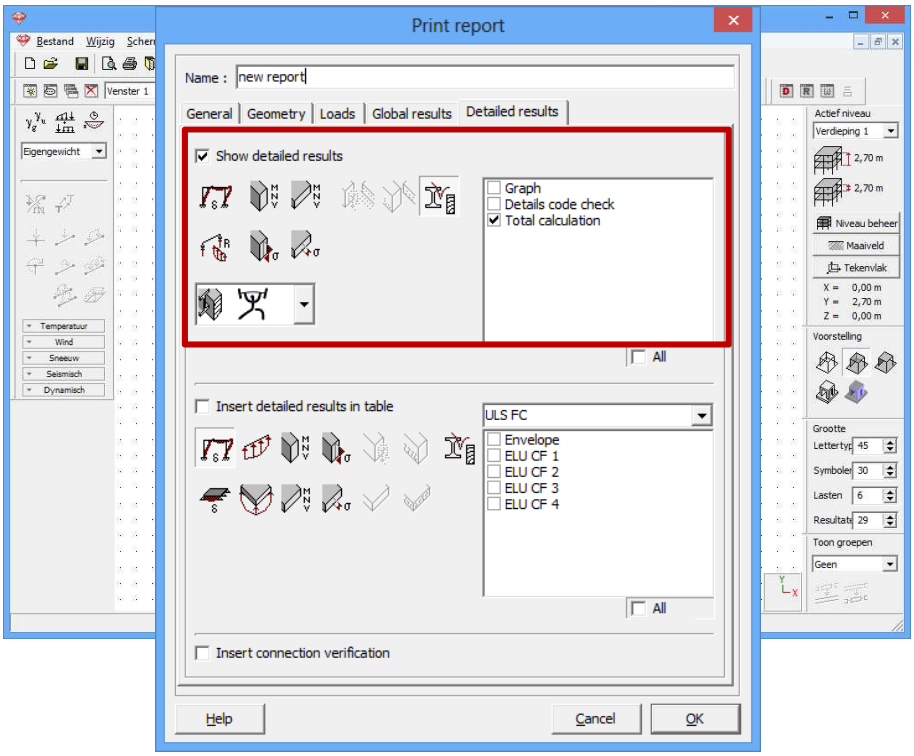
5.2. Dynamic and seismic loads

In order to include the dynamic and seismic parameters, you check the box ‘Include generated loads’.





5.3. Complete calculation of the design code verification (for steel or timber)



In Diamonds 2013 you can report the design code verification (resistance and buckling stability)

- a diagram with resistance and buckling stability graph
- the details of the critical resistance and buckling stability verification
- all the details of all the performed resistance and buckling stability verifications.

Remarks:

- The resistance verification details takes up 3 pages and the buckling stability details takes up 2 pages – just for one single bar! So, think your report through and only request the design code details for the appropriate bars.