WL -Windloads

Table of contents

Further information and descriptions can be found in the relevant documentation:

WL Parameter   Wind load parameters when selecting one of the implemented Euro
                                standards

Graphical input   Graphical input for the programs GEO, PLT, SCN and WL

Basic documentation - overview

In addition to the individual program manuals, you will find basic explanations on how to use the programs on our homepage www.frilo.eu in the download area (Manuals).

Tip: To go back - e.g. after a link to another chapter/document - in the PDF, use the key combination "ALT" + "left arrow key"

Possible applications

The program determines the distribution of horizontal forces acting on a building among its stiffening structural elements. Wind loads in the X or Y direction can be determined automatically according to DIN 1055-4, Edition 1986 or Edition 2005-03 (including amendment 1 3-2006).

Other standards:

EN 1991-1-4 with NA Germany, Austria and Great Britain

See also document WL Parameter (wind load parameters when selecting one of the implemented Euro standards)

Stiffening elements can be wall pillars or columns. Wall pillars consist of any number of straight wall sections connected to one another. Columns can have a rectangular or circular cross-section (solid or hollow cross-section).

Each column and each wall section can be assigned a material. The materials can be selected from a catalog for concrete or masonry or freely specified.

Vertical loads can be entered for columns and pillars, and the dead weight is determined automatically. For wall pillars, stresses can be determined with or without exclusion of tensile stresses for individual load cases and specified superpositions.

It is possible to read buildings entered with the Frilo Building model program via the menu item ▸ Edit ▸ FLGeo ▸ or to import floor plans entered with ISB-CAD.

To estimate the necessity of a calculation according to the second order theory, the lability numbers according to Brandt or, when using concrete according to DIN 1045 (07-2001), compliance with the stiffening criteria according to DIN 1045-1 and EN 1992-1-1 (NA D, A, GB) can be determined.

Calculation basis

Distribution of horizontal forces

The ceiling panels are assumed to be infinitely rigid in their plane. This assumption is generally correct, but in the case of elongated floor plans with large wall distances, deviations can arise in the lower floors, as the load distribution then approaches that of a rigidly supported continuous beam, i.e. the inner panels receive too small a load share under these boundary conditions.

Only the wall pillars and columns that have been defined as "horizontally rigid" are included in the calculation ⇒ see Walls, Columns.

The horizontal load distribution is based on the elasticity theory for oblique bending of thin-walled sections, considering the shear centers of the wall pillars (standard setting). The shear deformation and the torsional stiffness of the stiffening structural elements (according to St. Venant and warping force) are neglected, external torsional moments are absorbed by the spreading of the wall pillars. This corresponds to the total bar method according to Beck for Kappa = 0 (Bauingenieur 1969 p.80 ff.). Closed cores with considerable inherent torsional stiffness receive load shares that are too small with this approximation.

If all wall axes intersect at one point, the system is unstable for general loads and should therefore not be used.

In the case of wall pillars made up of several wall sections, it must be ensured that the shear forces can be absorbed in the vertical joints. The effective width must be considered.

```
LF Wind in X-RI,  Floor 3    OG 1  Upper edge ceiling   9.00 m

HORIZONTAL LOADS FLOOR    added loads of the
                                  above ceiling
Px =   33.05 kN   y =   5.00 m ,      Px1 =   170.50 kN
Py =    0.00 kN   x =   0.00 m ,      Py1 =     0.00 kN
Mz =    0.00 kNm                ,     Mz1 =   852.52 kNm

HORIZONTAL LOADS WALL PIERS (I,II = Principal axis)
(determined with the shear center of the wall pillars)
   WP      Fx       Fy       F I      F II     Alpha
  No.     [kN]     [kN]     [kN]     [kN]     [Grad]

    1   35.182   40.611   40.611  -35.182      90.0
    2  111.303  -59.300  120.634   36.771     -45.0
    3    0.300   18.589   18.589   -0.300      90.0
    4   56.770    0.100   56.770    0.100       0.0

Sum.  203.556   -0.000
```

Image: Output of horizontal load table

Wall pillars made of walls made of different materials are not permitted according to DIN 1053 Part 1, Section 6.8. If a calculation is to be carried out anyway, the shear center of the pillar is determined by ignoring the material differences. In this case, a calculation using the elastic center of gravity of the wall pillar may produce more realistic results. To do this, activate the corresponding setting in the Options menu.

Options for H-load distribution

▪ Neglect the stiffness of the weak axes:

For separate walls or for wall pillars consisting only of wall segments of one orientation, the stiffness in the weak axis is neglected.
This approach is common for masonry walls.

▪ H-load distribution is based on the wall surfaces:

The H load distribution is not based on stiffness, but on the wall area, if it is a case of separate walls or wall pillars consisting only of wall segments of one orientation.

This procedure is used, for example, for wooden panel walls.

Attention:

In the case of mixed systems consisting of separate walls and wall pillars made of walls of different orientations, the H-load distribution is recommended, considering the bending stiffnesses of the strong and weak axes and the shear centers of the wall pillars.
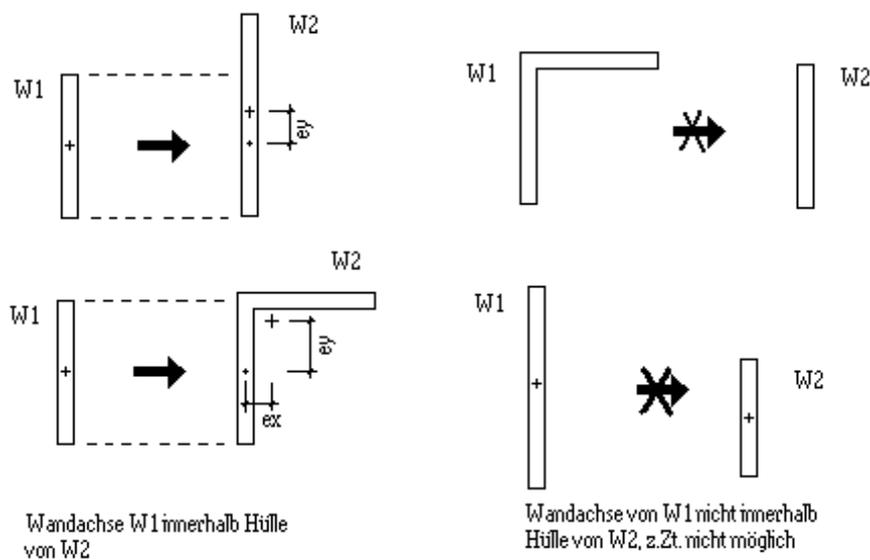
Load transfer

It is assumed that the wall pillars of a floor are rigidly clamped in the ceiling slab below (in the foundation for the lowest floor).

The calculation is carried out from the top to the bottom story, with the horizontal loads determined in a ceiling slab for each structural element in the ceiling slab below being included in the load to be distributed there.
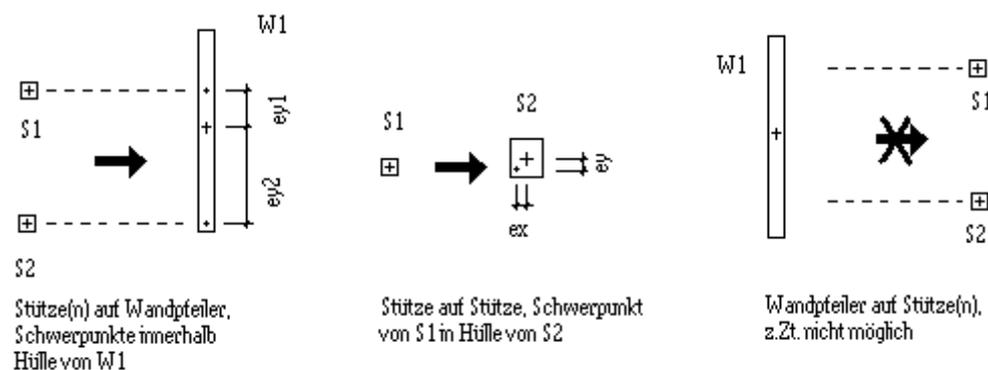
The feedback from stiffening elements that do not run through the entire building to the floors above is neglected in this procedure. In addition, in the case of non-continuous stiffening elements, the transfer of the internal forces of the end of the stiffening element via the ceiling slab must be ensured.

In order to transfer vertical loads and moments from a wall pillar to a wall pillar below, the wall axes of the upper wall pillar must lie in the envelope of the wall pillar below.

Wandachse W1 innerhalb Hülle von W2

Wandachse von W1 nicht innerhalb Hülle von W2, z.Zt. nicht möglich

1. Wall axis W1 within envelope of W2
2. Wall axis of W1 not within envelope of W2, currently not possible

For load transfer from a column to a wall pillar or to a column, the center of gravity of the upper column must lie within the envelope of the underlying wall pillar or column.

Stütze(n) auf Wandpfeiler, Schwerpunkte innerhalb Hülle von W1

Stütze auf Stütze, Schwerpunkt von S1 in Hülle von S2

Wandpfeiler auf Stütze(n), z.Zt. nicht möglich

1. Support(s) on wall pillars, center of gravity within envelope of W1
2. Column on column, center of gravity of S1 in envelope of S2
3. Wall pillars on column(s) currently not possible

If the conditions for load transfer are not met, this is shown in the output table for the internal forces.

In these cases, a load transfer specified by the user (Wall pillar table) is possible.

If moments caused by eccentricity during load transfer (distance between the centers of gravity of the vertical structural elements positioned one above the other) are to be redistributed via a force couple in the ceiling slabs, the "centered load transfer" option (menu item Options) must be activated.

Since the loads then introduced into the ceiling slabs are not considered in the calculation, it should be ensured that they remain negligibly small due to the largely symmetrical arrangement of the vertical structural elements.

Stress analysis

The stress analysis is carried out floor by floor, according to the division defined in the Output section of the load case or the superposition.

Stress determination is not possible for wall pillars with walls made of different materials.

For wall pillars, stress analysis can optionally be carried out with or without tensile stresses ⇒ see Wall pillars

When determining stresses excluding tensile stresses, linear-elastic material behavior of the compressed cross-sectional area is assumed. If the resulting vertical force lies outside the tilting edges, stress analysis excluding tensile stresses is not possible.

```
SECTION FORCES AND STRESSES WALL PIERS  EG
WP Dist.Foot-       Nz      Mx       My    maxSig   minSig
No    point[m]     [kN]    [kNm]    [kNm]  [N/mm2]  [N/mm2]Procedure

 1      0.00      -0.0   -695.2    602.3    1.004   -0.308 with tens.

 2      0.00      -0.0   1015.2   1905.5    0.684   -1.583 with tens.

 3      0.00      -0.0   -321.9     80.5    1.744   -1.744 with tens.

 4      0.00      -0.0     -1.7    972.0    1.244   -1.244 with tens.
```

Image: Output stress table

Stiffening criterion for neglecting the effects of second order theory

Compliance with the stiffening criteria according to DIN 1045-1

To assess the rotational and lateral stiffness of the building, stiffening criteria must be observed so that a more precise verification according to second order theory can be omitted.

Lateral stiffness in Y-direction:

$$\frac{1}{h_{ges}} \cdot \sqrt{\frac{EI_x}{F_{Ed}}} \quad \geq \frac{1}{0,2 + 0,1 \cdot m} \quad \text{für } m \leq 3$$

$$\geq \frac{1}{0,6} \quad \text{für } m > 3$$

Lateral stiffness in X-direction:

$$\frac{1}{h_{ges}} \cdot \sqrt{\frac{EI_y}{F_{Ed}}} \quad \geq \frac{1}{0,2 + 0,1 \cdot m} \quad \text{für } m \leq 3$$

$$\geq \frac{1}{0,6} \quad \text{für } m > 3$$

Rotational stiffness:

$$\frac{1}{h_{ges}} \cdot \sqrt{\frac{EI_\omega}{\Sigma(F_{Ed,j} \cdot r_j^2)}} + \frac{1}{2,28} \cdot \sqrt{\frac{GI_t}{(F_{Ed,j} \cdot r_j^2)}} \quad \geq \frac{1}{0,2 + 0,1 \cdot m} \quad \text{für } m \leq 3$$

$$\geq \frac{1}{0,6} \quad \text{für } m > 3$$

m	Number of floors
F_{Ed}	the sum of the design values of the vertical loads with $\gamma_F = 1,0$
$F_{Ed,j}$	the design value of the vertical load of the column j with $\gamma_F = 1,0$
EI_x, EI_y	the sum of the nominal bending stiffnesses of all vertical stiffening members in the main axis directions
EI_ω	the sum of the nominal warping constants
GI_t	the sum of the torsional stiffnesses (St. Vernan)

The decisive bending and torsional stiffnesses of the stiffening members combined to form the overall bar are initially those of the lowest floor. If the bending stiffnesses vary over the height of the building, an equivalent stiffness is determined with the same head deformation as a result of a unit load in the respective direction (FaxX, FakY). Stiffness jumps in the torsional stiffness cannot be considered. When determining the stiffnesses, state I is assumed.

When determining the torsional stiffness, it is also assumed that the vertical loads are evenly distributed in the floor plan (resulting load application point = floor plan center).

This results in $\Sigma(F_{Ed,j} \cdot r_j^2) = F_{Ed} \cdot i_n^2$ mit $i_n^2 = i_p^2 + c^2$

i_p	polar radius of gyration of the floor plan
c	Distance ceiling pivot point – floor plan center

Under the above assumptions, a favorable torsional stiffness is achieved by keeping the distance between the center of gravity of the floor plan and the ceiling pivot point as small as possible.

Determination of the lability numbers according to Brandt

(Concrete and reinforced concrete construction 7/76, 3/77)

In order to estimate the rotational and lateral stiffness of the building, instability numbers are determined. If these are complied with, a more precise proof according to second order theory can be omitted.

Lateral stiffness in Y-direction:

$$\alpha_x = h_{tot} \cdot \sqrt{\frac{F_v}{\Sigma EI_x}} \quad \le 0{,}6 \qquad n \ge 4$$

$$\le 0{,}2 + 0{,}1 \cdot n \quad n < 4$$

Lateral stiffness in X-direction:

$$\alpha_y = h_{tot} \cdot \sqrt{\frac{F_v}{\Sigma EI_y}} \quad \le 0{,}6 \qquad n \ge 4$$

$$\le 0{,}2 + 0{,}1 \cdot n \quad n < 4$$

Rotational stiffness:

$$\kappa = h_{tot} \cdot \sqrt{\frac{\Sigma GI_t}{\Sigma EI_\omega}}$$

wenn $\kappa < 10$

$$\alpha_\psi = \Phi \cdot h_{tot} \cdot \sqrt{\frac{F_v \cdot i_n^2}{EI_\omega}} \quad \le 0{,}6 \qquad n \ge 4$$

$$\le 0{,}2 + 0{,}1 \cdot n \quad n < 4$$

wenn $\kappa > 10$

$$\alpha_\psi = 2{,}28 \cdot \sqrt{\frac{F_v \cdot i_n^2}{\Sigma GI_t}} \quad \le 0{,}6 \qquad n \ge 4$$

$$\le 0{,}2 + 0{,}1 \cdot n \quad n < 4$$

n	Number of floors
F_y	the sum of the vertical loads
EI_x, EI_y	the sum of the nominal bending stiffnesses of all vertical stiffening members in the main axis directions
EI_ω	the sum of the nominal warping constants
GI_t	the sum of the torsional stiffnesses (St.Vernant)

The decisive bending and torsional stiffnesses of the stiffening components combined to form the overall bar are initially those of the lowest storey. If the bending stiffnesses vary over the height of the building, an equivalent stiffness is determined with the same head deformation as a result of a unit load in the respective direction (FaxX, FakY). Stiffness jumps in the torsional stiffness cannot be taken into account. When determining the stiffnesses, state I is assumed.

When determining the torsional stiffness, it is also assumed that the vertical loads are evenly distributed in the floor plan (resulting load application point = floor plan center).

This results in $i_n^2 = u\, i_p^2 + c^2$

i_p polar radius of gyration of the floor plan

c   distance ceiling pivot point − floor plan center

Under the above assumptions, a favorable torsional stiffness is achieved by keeping the distance between the center of gravity of the floor plan and the ceiling pivot point as small as possible.

Stiffening criterion according to Eurocode EN 1992-1-1

If a stiffness criterion is met, the consideration of second order effects can be dispensed with.

Based on a work by Westerberg /2/, equation 5.18 formulates a stiffening criterion for the immovability of the entire structure, which applies under the following boundary conditions:

(1)   sufficient torsional resistance, symmetrical load-bearing system

           See also stiffening criterion for torsion according to NA_D

(2)   Shear deformations negligible

(3)   Rigid foundation of the stiffening members

(4)   The stiffening members have a stiffness that is almost constant over the height

           The program may calculate a deformation-equivalent average stiffness

(5)   Uniform increase in vertical force per floor

$$F_{v,Ed} \leq K_1 \cdot \frac{n_s}{(n_s+1{,}6)} \cdot \frac{\Sigma(E_{cd} \cdot I_c)}{L^2} \qquad \text{Eq. 5.18}$$

n_s Number of floors

$F_{v,Ed}$ the sum of the design values of all vertical loads of a building

           NA_D: Design value with $\gamma F = 1.0$

$\Sigma(E_{cd}*I_c)$ Sum of the bending stiffnesses of all stiffening elements (total bar) with

         I_c: smaller value of the moments of inertia Ix and Iy in the main axis direction

         $E_{cd} = Ecm / \gamma_{cE}$ Calculated value of the elastic modulus of concrete

           Ecm   Elastic modulus of concrete according to Table 3.1

           γ_{cE} Safety factor (NDP) according to 5.8.6 (3)

         The decisive bending stiffnesses of the stiffening members combined to form the overall bar are initially those of the lowest floor. If the bending stiffnesses vary over the height of the building, an equivalent stiffness is determined with the same head deformation due to a unit load in the respective direction (FaxX, FakY).

K1:   Factor to consider crack formation

         If it is verified that the members are not cracked, K1= K2

NDP	γ_{cE}	K1	K2
EN	1.2	0.31	0.62
NA_D	=EN	=EN	=EN
NA_A	=EN	=EN	=EN
NA_GB	=EN	=EN	=EN

NA_D: Eq. 5.18 is converted according to the verification format previously used in Germany as follows:

$$\frac{F_{v,Ed} \cdot L^2}{\Sigma(E_{cd} \cdot I_c)} \leq K_1 \cdot \frac{n_s}{(n_s + 1,6)}$$

In addition, a stiffening criterion for the torsion must also be considered:

$$\frac{1}{\left(\frac{1}{L} \cdot \sqrt{\frac{E_{cd} \cdot I_\omega}{\Sigma(F_{v,Ed,j} \cdot r_j^2)}} + \frac{1}{2.28} \cdot \sqrt{\frac{G_{cd} \cdot I_T}{\Sigma(F_{v,Ed,j} \cdot r_j^2)}}\right)^2} \leq K_1 \cdot \frac{n_s}{n_s + 1,6} \quad Gl.5.18.1DE$$

$E_{cd} \cdot I_\omega$ Design value of the sum of the nominal warping constant of all stiffening elements

$$I_\omega = \Sigma(I_{x,i} \cdot x_{Mm,i}^2 + I_{y,i} \cdot y_{Mm,i}^2)$$

$G_{cd} \cdot I_T$ Design value of the sum of the torsional stiffnesses (St. Vernan) of all stiffening elements

$$G_{cd} = \frac{E_{cd}}{2 \cdot (1+\mu)}$$

μ Poisson's ratio, for concrete usually $\mu = 0.2$

I_T St. Vernan torsional moment of inertia

For a wall pillar consisting of several partial walls:

open cross-section: $I_T = \Sigma(\frac{1}{3} \cdot h_i^3 \cdot l_i)$

closed cross-section: according to Petersen /1/ p.1309 ff.

Stiffness jumps in torsional stiffness over the building height cannot be considered. When determining the torsional stiffness, it is also assumed that the vertical loads are evenly distributed in the floor plan (resulting load application point = floor plan center).

This results in $\Sigma(F_{V,Ed,j} \cdot r_j^2) = F_{v,Ed} \cdot i_n^2$ mit $i_n^2 = i_p^2 + c^2$

i_p polar radius of gyration of the floor plan

c distance ceiling pivot point - floor plan center

Under the above assumptions, a favorable torsional stiffness is achieved by keeping the distance between the center of gravity of the floor plan and the ceiling pivot point as small as possible.

Wind load determination for structures not susceptible to vibration

The wind load on a ceiling is determined from sections above and below the ceiling, each corresponding to half a floor. If the wind pressure at the lower edge and upper edge of a section is different, the section is divided again internally.

Wind load in section j:

$Fwj = cfj \cdot q(zej) \cdot Aj$

cfj: Aerodynamic coefficient according to input - applies to the entire building. Sloping roofs are not considered separately.

q(zej) Wind pressure ordinate at the top of the section - see below.

Aj Wind area of the section.

Determination of the wind pressure ordinate

DIN 1055-4 (1986): EN 1991-1-4: - see WL Parameter

height-dependent gradation according to Table 1

0 m <	h < 8 m:	q = 0,5 kN/m²
8 m <	h < 20 m	q = 0,8 kN/m²
20m <	h < 100 m	q = 1,1 kN/m²
	h > 100 m	q = 1,3 kN/m²
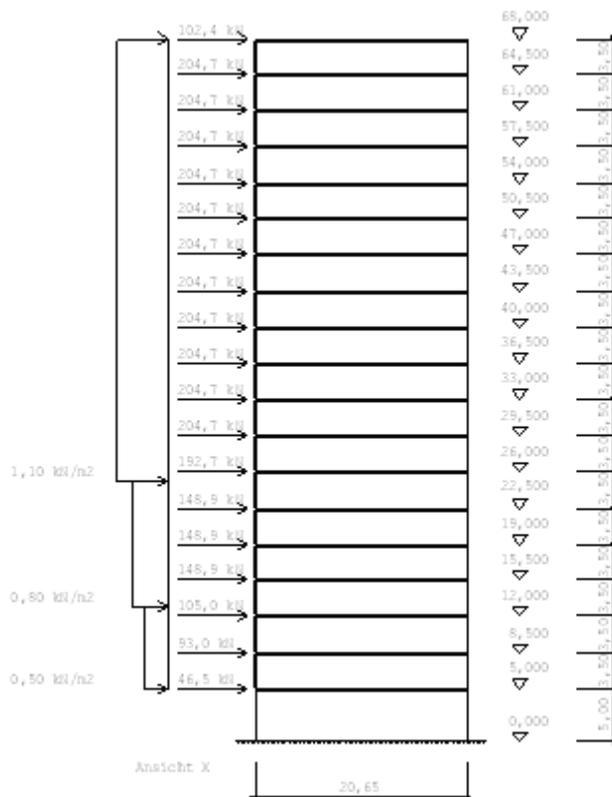
<u>DIN 1055-4 (2005):</u>

Gust velocity pressure $q = k \cdot q_{ref} \cdot (z/10)^a$

z:	Distance of ordinate from lower edge of exposure to wind
qref:	Reference wind pressure according to the wind zone (Appendix A, Figure A1)
k:	Coefficient depending on the terrain category (Appendix B, Table B2 and 10.3 (4) Eq. 10 - 17)
a:	Exponent depending on the terrain category (Appendix B, Table B2 and 10.3 (4) Eq. 10 - 17)

According to Figure 3, the following qualitative progression applies to buildings with rectangular floor plans:

Buildings wit $h \le b$

$q = q(z = h)$ const.

Buildings with $b < h \le 2b$

$z > b$ $q = q(z = h)$ const.

$z \le b$ $q = q(z = b)$ const.

Buildings with $h > 2b$

$z > h - b$ $q = q(z = h)$ const.

$z \le b$ $q = q(z = b)$ const.

$b < z < h-b$ $q = q(z = zj)$ const. per partial rectangle

In the area z > b, the wind pressure curve is approximated using partial rectangles whose upper edges are at a distance zj from the lower edge of the wind exposure. A division into rectangles with hj approx. 5 m has proven to be sufficiently accurate.

The width b is determined from the extension of the outer contour of the lowest floor perpendicular to the wind direction.

This represents an approximation for buildings with floor plans that deviate from a rectangle or with floor plans that vary from floor to floor.
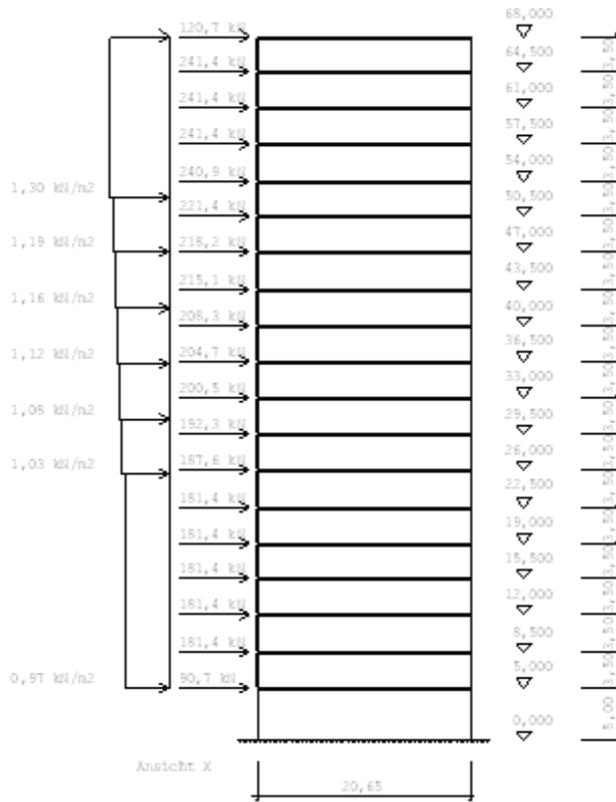
WIND IN Y-DIRECTION AUTOMATICALLY GENERATED: (DIN 1055-4 (2005-03))

Wind zone   2   qref   =   0.39 kN/m2

Altitude above sea level 5 m   Altitude factor = 1.00

Terrain category II (isolated buildings, trees and hedges)

Height for lower edge wind exposure hUkWi= 0.00 m

Force coefficient Cf= 1.30 Eccentricity e= 0.00 m

Area exposed to wind automatically determined

			Area exposed to wind				Wind load	
Gesch	htot	un/	h	xa	xe	q	W	xw
Nr.	[m]	ob	[m]	[m]	[m]	[kN/m2]	[kN]	[m]
1	5.00	un	2.50	-0.20	40.70	0.97	129.56	20.25
		ob	1.75	-0.20	40.70	0.97	90.69	20.25
2	8.50	un	1.75	-0.20	40.70	0.97	90.69	20.25
		ob	1.75	-0.20	40.70	0.97	90.69	20.25
3	12.00	un	1.75	-0.20	40.70	0.97	90.69	20.25
		ob	1.75	-0.20	40.70	0.97	90.69	20.25
4	15.50	un	1.75	-0.20	40.70	0.97	90.69	20.25
		ob	1.75	-0.20	40.70	0.97	90.69	20.25

```
 5  19.00 un    1.75  -0.20  40.70   0.97  90.69  20.25
        ob     1.65  -0.20  40.70   0.97  85.51  20.25
        ob 1   0.10  -0.20  40.70   1.03   5.48  20.25
 6  22.50 un    1.75  -0.20  40.70   1.03  95.84  20.25
        ob     1.75  -0.20  40.70   1.03  95.84  20.25
 7  26.00 un    1.74  -0.20  40.70   1.03  95.29  20.25
        un 1   0.01  -0.20  40.70   1.08   0.57  20.25
        ob     1.75  -0.20  40.70   1.08 100.23  20.25
 8  29.50 un    1.75  -0.20  40.70   1.08 100.23  20.25
        ob     1.75  -0.20  40.70   1.08 100.23  20.25
 9  33.00 un    0.08  -0.20  40.70   1.08   4.58  20.25
        un 1   1.67  -0.20  40.70   1.12  99.33  20.25
        ob     1.75  -0.20  40.70   1.12 104.09  20.25
10  36.50 un    1.75  -0.20  40.70   1.12 104.09  20.25
        ob     0.17  -0.20  40.70   1.12  10.11  20.25
        ob 1   1.58  -0.20  40.70   1.16  97.10  20.25
11  40.00 un    1.75  -0.20  40.70   1.16 107.55  20.25
        ob     1.75  -0.20  40.70   1.16 107.55  20.25
12  43.50 un    0.26  -0.20  40.70   1.16  15.98  20.25
        un 1   1.49  -0.20  40.70   1.19  94.24  20.25
        ob     1.75  -0.20  40.70   1.19 110.68  20.25
13  47.00 un    1.75  -0.20  40.70   1.19 110.68  20.25
        ob     0.35  -0.20  40.70   1.19  22.14  20.25
        ob 1   1.40  -0.20  40.70   1.30  96.58  20.25
14  50.50 un    1.75  -0.20  40.70   1.30 120.72  20.25
        ob     1.75  -0.20  40.70   1.30 120.72  20.25
15  54.00 un    1.75  -0.20  40.70   1.30 120.72  20.25
        ob     1.75  -0.20  40.70   1.30 120.72  20.25
16  57.50 un    1.75  -0.20  40.70   1.30 120.72  20.25
        ob     1.75  -0.20  40.70   1.30 120.72  20.25
17  61.00 un    1.75  -0.20  40.70   1.30 120.72  20.25
        ob     1.75  -0.20  40.70   1.30 120.72  20.25
18  64.50 un    1.75  -0.20  40.70   1.30 120.72  20.25
        ob     1.75  -0.20  40.70   1.30 120.72  20.25
19  68.00 un    1.75  -0.20  40.70   1.30 120.72  20.25
        ob     0.00  -0.20  40.70   1.30   0.00  20.25
```

System input

The system is described by its floors, the corresponding external contours, wall pillars, supports and the materials used.

These values can be entered both graphically and in tabular form.

Tabular input

Call up the tabular input via the left menu tree ▸ System tab.

See chapter Tabular input.

Graphical input

The graphical input is described in the document "Graphical input.pdf".

DXF file as input aid

To support graphic input, you can use a DXF file by loading it via File ▸ Import ▸ DXF. If the unit used is not [cm], enter an appropriate scale. Hide all layers that are not required. You can now carry out graphic input in a convenient way. You can delete the DXF layer again using the "Grf Options" menu item.

Take over data set from ISB- Glaser

If there are floor plans entered with ISB-CAD from Glaser, these can be imported as follows. First define the floors with the floor heights and designations and then import the floor plans onto the predefined floors (File ▸ Import ▸ GLASER -isb cad-).
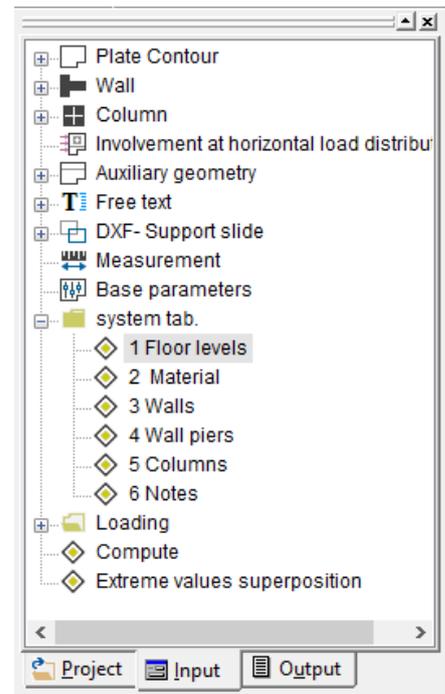
Take over ASCII data set from "Frilo Building Model"

Create a new position and import the corresponding ASCII data set via File ▸ Import ▸ ASCII.

The vertical loads on pillars and columns of the ceiling load cases G and P calculated with the Frilo Building model program can be transferred to the wind loads program via the ASCII interface.

A load case G is generated from the ceiling load cases G. The ceiling load cases P are combined to form a load case P.

Testing the input data

You can check that the input is correct in the info window (bottom left). In the text view ("Text output" tab), a detailed error log is output after the input data has been displayed.

See chapter Testing the input data.

Tabular input

The tabular system input is a multi-page input dialog controlled by tabs for entering floors (including external contours and wind surfaces), wall pillars, columns and materials.

| 1 Floor levels | 2 Material | 3 Walls | 4 Wall piers | 5 Columns | 6 Notes |
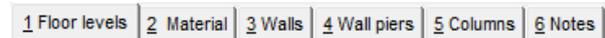
Floor levels

This table shows all the floors belonging to the building. New floors can be created and existing floors removed. The respective outer contour and the wind area can be edited.

Floor number

The floor number is assigned automatically when the floor is created and is used for identification.

For a floor attached at the top, the number is increased by one, and for a floor attached at the bottom, it is decreased by one.

Floor height

If the floor height is changed on one floor, the height of the top edge of the ceiling changes for all floors below it, while the floors above remain unchanged.

An alternative editing option is available in the Basic parameters dialog for graphical input.

Top edge of ceiling

If the height of the ceiling top edge is changed on one floor, the heights for the ceiling top edge on all floors will be adjusted accordingly.

An alternative editing option is available in the Basic parameters dialog for graphical input.

Floor designation

A name of up to 10 characters can be edited. An alternative editing option is available in the Basic parameters Floor dialog of the graphical input.
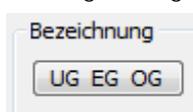
Automatic naming is possible using the "LF GF UF" button, whereby the floor specified in a selection dialog is designated "GF" (ground floor), the lower floors are designated "LF1, 2, ..." in descending order, and the upper floors are designated "OUF1, 2, ..." in ascending order.

Create floor

	Create the floor below
	Create floor above

When creating a floor, it is possible to adopt the data from the neighboring floor.

Floors can also be created using the <u>Floor selection</u>.

Delete floor

When you delete a floor, all floor data is deleted.

Copy floor
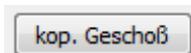
kop. Geschoß   Copy the current floor to any number of floors.
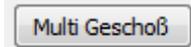
Multi Geschoß   Attach any number of similar floors to the top or bottom floor.

Edit the outer contour of a floor slab

▸ System tabular ▸ Floor levels ▸ Button outer contour

The segments of the outer contour polygon can be edited in the table. The starting point of the segment is the starting point entered in the current line. The end point is the starting point of the following line (if current = last line, it is the first line).

Arched polygon segments can be created by entering another arch point.

The outer contour of another floor can be adopted using the "same as floor" button.

The outer contour of the current floor can be copied to other floors using the "Copy" button. The floors in question must be marked in a selection dialog that appears (simply click on the desired floor).

Außenkontur Geschoss 1					
	xa [m]	ya[m]	Bogen	xb[m]	yb[m]
1	-5,00	-5,00	☐		
2	3,50	-5,00	☐		
3	3,50	-3,00	☐		
4	5,00	-1,50	☐		

Edit the wind area of a floor

▸ System tabular ▸ Floor levels ▸ Wind area button

The wind area describes the area of the building envelope belonging to a floor on which wind could be exposed. The height of the wind area below the ceiling level is determined by half the height of the current floor, the height of the wind area above the ceiling level is determined by half the height of the floor above.

If the wind can act on the building unhindered, the extension in X and Y directions results directly from the enclosing rectangle of the outer contour. In this case, the wind area can be determined automatically ⟹ see Generate wind load automatically.

If these assumptions do not apply, e.g. if neighboring buildings are to be considered or the outer contour extends from the upper to the lower floor, a differentiated input of the wind areas is also possible. For wind attack in the X direction, a start coordinate YA and an end coordinate YE must be specified, and for wind exposure in the Y direction, a start coordinate XA and an end coordinate XE must be specified. These must not lie outside the enclosing rectangle of the outer contour.
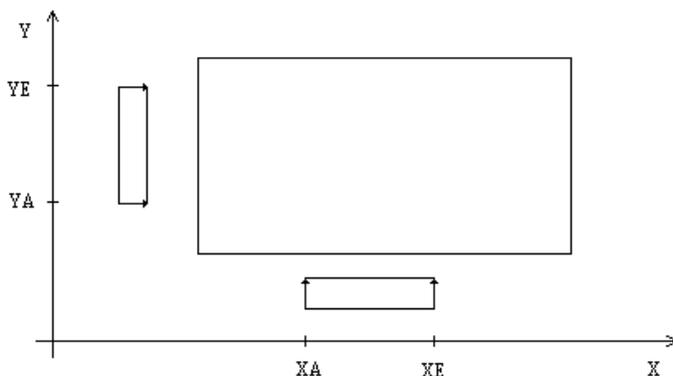
The wind area entered here will only be considered if you have not set automatic wind area determination when automatically generating the wind load.

Material

This table provides the materials that can be used when entering walls and columns.
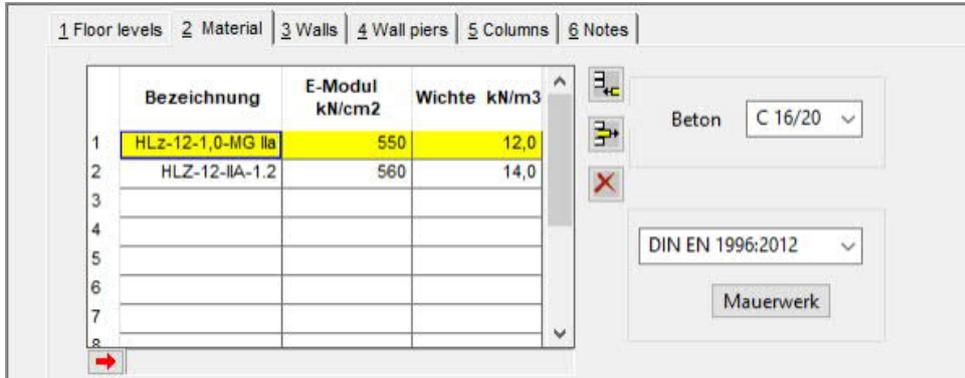
The elastic modulus (determination of element stiffness), the specific gravity (determination of dead weight) and a designation must be entered. Concrete classes according to DIN 1045 7/88 and

DIN 1045-1 as well as masonry classes according to DIN 1053 can be selected using selection lists.

A material may only be deleted if it is no longer used in any component.

Button / Dialog Masonry Selection

Selection of masonry type and type (brick, sand-lime brick, lightweight concrete block, normal concrete block, aerated concrete block), strength class, density class and mortar group.

In addition, products from different manufacturers can be selected "… after approval" (Wienerberger, Schlagmann Poroton, UNIPOR, Kellerer ZMK Ziegelsysteme, BV Leichtbeton).

You can also define the material yourself using "… user-defined".

See also the description in the masonry programs - Material dialog.
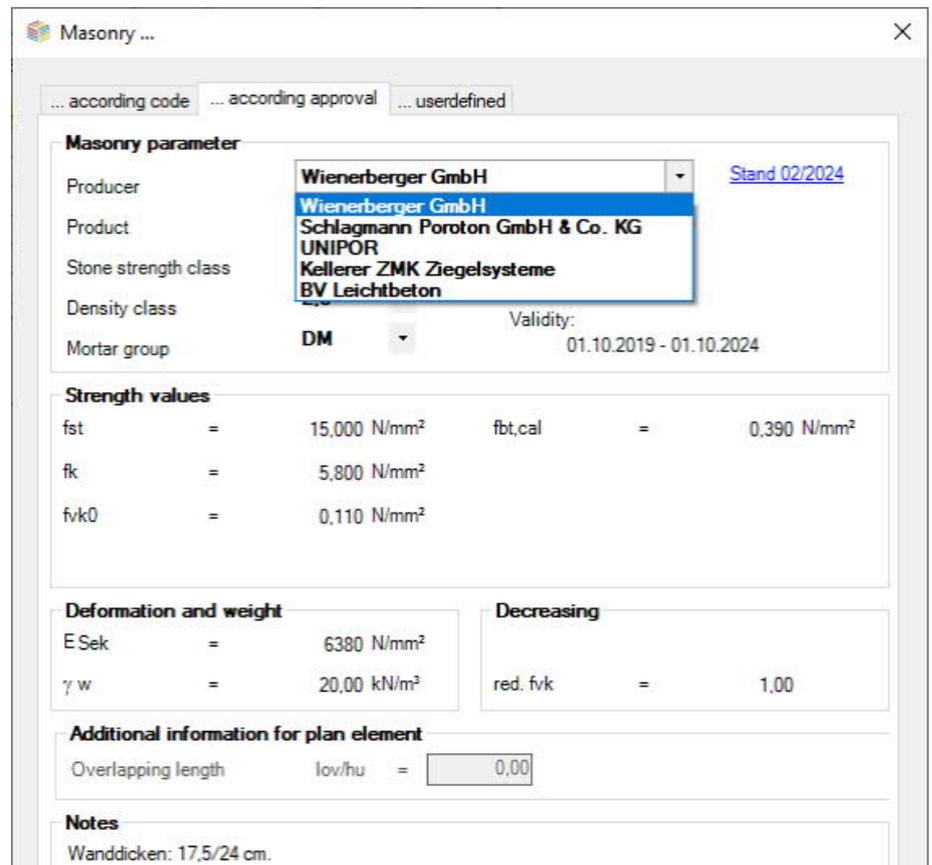
Walls

The input and editing of the wall pillars are done via a table which contains the walls of the wall pillars of the current floor.

The current floor can be changed using the arrow keys next to the floor display.

Wall pillar and wall number

A wall pillar consists of a number of interconnected walls made of the same material.

Wall pillars that are identical from floor to floor should have the same pillar number. If a pillar number that does not yet exist is assigned, a new wall pillar is created.

The walls are assigned to the respective wall pillar using the pillar number. If a pillar number that does not yet exist is assigned, a new wall pillar is created.

They are identified in the wall pillar using their wall number. A wall number may only be assigned once within a wall pillar.

Wall geometry

The wall geometry is characterized by the start and end point of the wall axis and the wall width. When entered in a table, the start and end points are always related to the wall center line.

Walls with a length of 0, walls that are inside another wall, and overlapping walls in a line are not accepted.

Material

By entering a material number, a previously defined material can be assigned to a wall. The F5 key can be used to select from the list of available materials. Optionally, a material assignment for all walls of the pillar is possible.

If the material of a wall changes in pillars with multiple walls, the pillar is split.

Insert wall - new pillar

By pressing "Return" in the last occupied cell of the table or by pressing the Insert button , a new wall is created, which is initially assigned to the pillar of the previous row. A new pillar can be created by assigning a pillar number that does not yet exist.

A newly entered wall is only finally inserted into the wall pillar and becomes graphically visible after the table row has been left.

Delete wall

Deleting a wall also removes its wall connections

$\Rightarrow$ see "Removing wall connections".
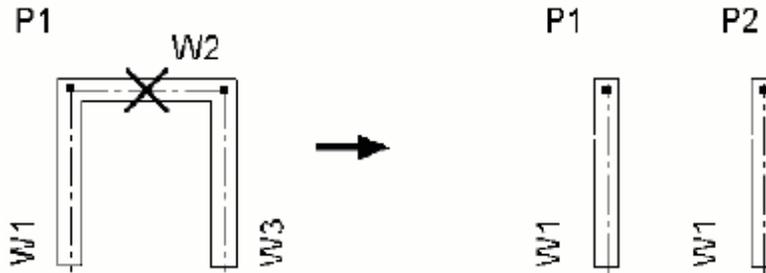
Figure 1: Deleting a wall

Remove wall from pillar

A new wall pillar is formed from the activated wall.

Combine wall with pillar

Wall is united with the selected pillar (pillar of the same material with overlaps).

Wall connections

A connection between a wall W1 and another wall W2 occurs when a wall point of W1 lies in the envelope of wall W2.

If there is a point common to both walls, this is referred to as a point-point connection.
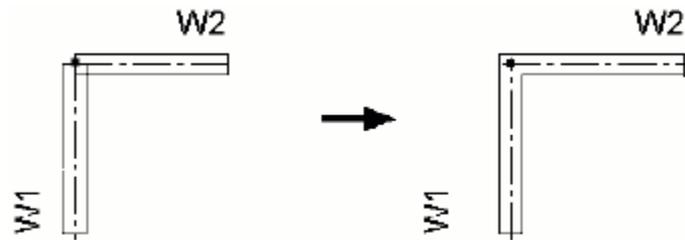
Figure 2: Point - point - connection

If there is an intersection point between the center line of W1 and the outer edge of W2, this is referred to as an outer edge point connection. If the wall point of W1 is not on the center line of W2, it is moved there.
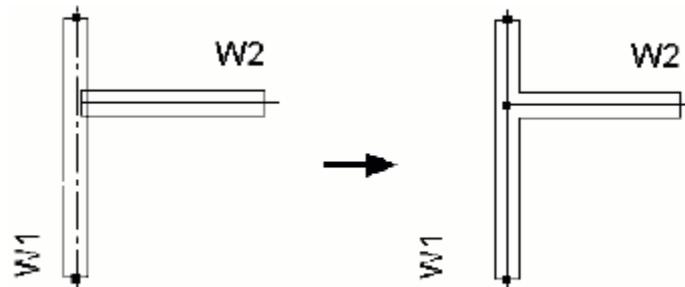
Figure 3: Outer edge - point - connection

If an outer edge of the wall W1 intersects the center line of the wall W2 just outside the axis of W1, the wall point of W1 is moved to the wall point of W2, thus creating a point-point connection.

Figure 4: Subsequent point-point connection

Crossing walls

Durch das Einfügen einer Wand oder durch Manipulation von Wandpunkten kann eine Wand entstehen, die eine oder mehrere andere Wände eines Pfeilers kreuzt. Kreuzende Wände werden automatisch wie in Bild 5 dargestellt geteilt.
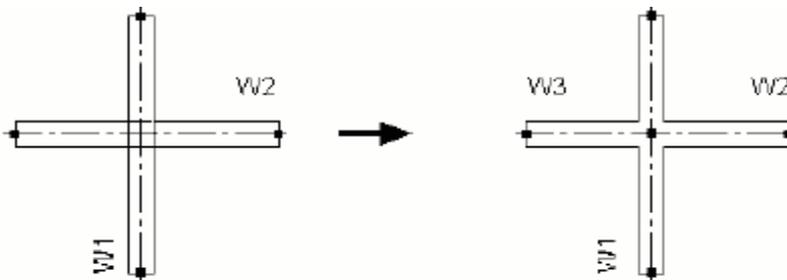
Figure 5: Crossing walls

Changing bound wall points

If a wall point is changed that is also a wall point of one or more other walls, the points of the affected walls will also be changed upon request.
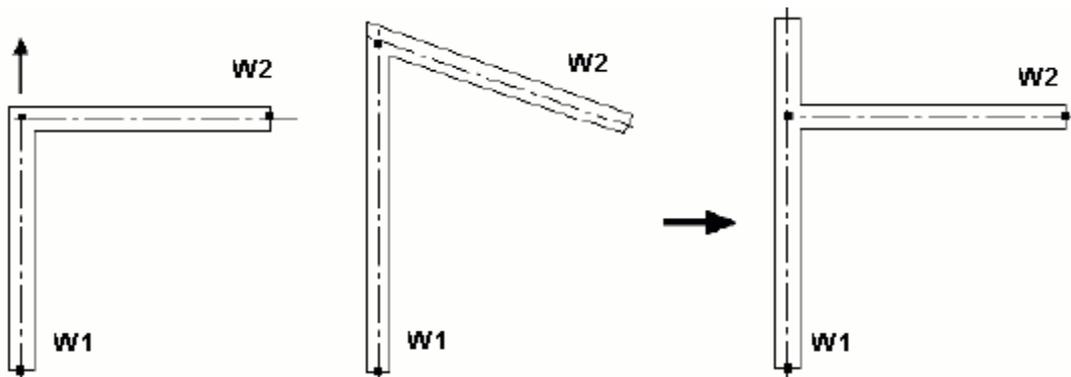
Figure 6: Bound wall points (a) Do not disconnect connections (b) Disconnect connections

Disconnecting wall connections

Disconnecting point-point connections

If walls made of different materials were connected, the primary wall will be lengthened and the secondary wall will be shortened when a point-to-point connection is removed. The primary wall is the wall with the smallest number.
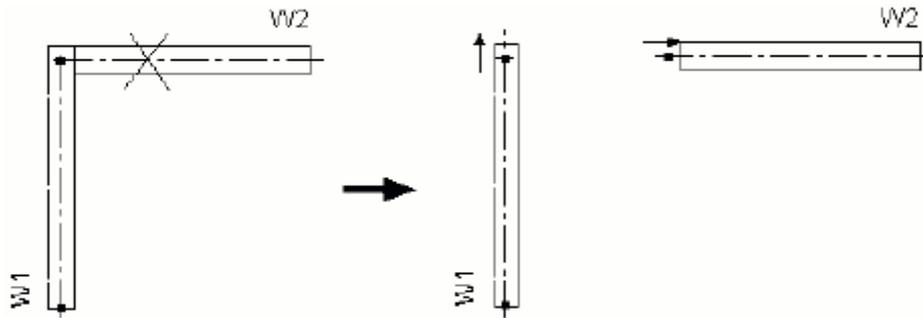
Figure 7: Diconnect point - point - connection (a) primary wall lengthens (b) secondary wall shortens

When disconnecting an outer edge-point connection, the wall point is moved from the center line to the outer edge in the direction of the wall.
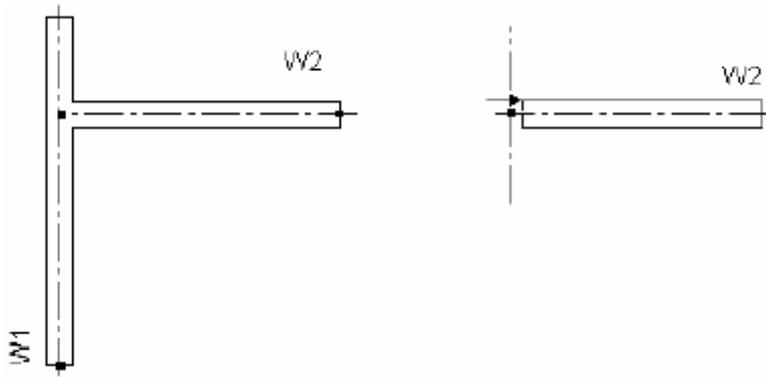
Figure 8: Disconneting outer edge - point - connections

New entry of a wall or a wall section

If the newly entered wall overlapped with an existing pillar, it was previously always assigned to this wall pillar.

Now, after entering the wall axis, a possible union can be accepted or prevented in the wall dialog.
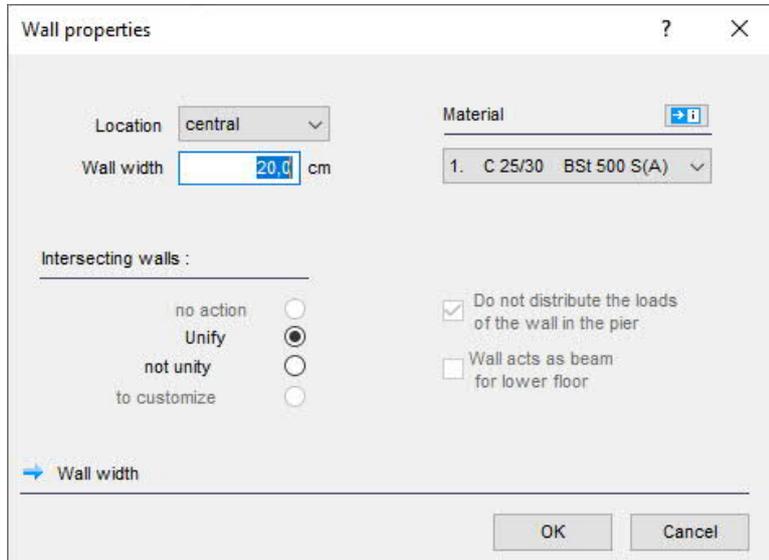
If no union takes place, the entered wall forms a new pillar. The wall axis is nevertheless extended to the wall axis of the cut pillar, whose envelope ends at the intersection with the envelope of the cut pillar.

If a new pillar crosses an existing pillar, the existing pillar is split.

Effects of changing a wall pillar on neighboring pillars

If the width of a wall changes, the walls of adjacent pillars (whose axes are extended to the intersection with the changed wall) are adjusted.

Wall pillar

In this table, the following properties that apply to the entire wall pillar can be changed:

Pillar number

A positive pillar number that has not yet been assigned to the floor can be entered.

Wall pillars that are identical from floor to floor should be given the same number.

Designation

By default, the pillar number is preceded by a "W".

Active: Participation in horizontal stiffening

By removing the marking in the "active" column, individual wall pillars (e.g. lightweight partition walls) can be excluded from the horizontal load transfer.

Stress analysis

This selection box can be used to set the method of stress analysis (with or without tensile stresses).

A global setting of the method of stress analysis - valid for all wall pillars in the building - is possible using these two buttons.

Load on pillar

In cases where automatic load transfer is not possible in the program (identified by a pillar number of -1), the user can specify a load transfer to a specific wall pillar via the "Load on pillar" column. The normal forces and moments of the pillar are transferred to the pillar mentioned.

This can be particularly useful in the following cases:

- Wall pillars above and below contain only one wall and the lower wall is shorter than the upper wall (axis is not in the envelope of the lower wall).
- Wall pillar center of gravity is not in the envelope (e.g. U-shaped wall pillars) and the center of gravity of the pillar above and below shifts due to a shorter or longer wall.

In any case, the user must check whether the conditions for such a specified load transfer are met.

A comprehensive automatic load transfer via an intermediate plate calculation is currently being worked on.

Delete pillars

All walls of a wall pillar are deleted.

Copy wall pillars

Copy or delete a wall pillar from floor a to floor b (the corresponding dialog is displayed). Existing pillars with the same number are overwritten after a corresponding warning. This switch can be used to quickly adopt input values for the same pillars on other floors.

Wall pillars only contain walls with the same material

If the material of a wall changes in a pillar with multiple walls, the pillar will be split.

Effects of changing a wall pillar on neighboring pillars

If the width of a wall changes, the walls of adjacent pillars (whose axes are extended to the intersection with the changed wall) are adjusted.

Columns

The table shows the columns present on the current floor. The current floor can be changed using the arrow keys next to the floor display.

	St Nr	Mat.	Typ	Hohl-St.	b [cm]	d [cm]	bi [cm]	di [cm]	Phi [Grd]	x [m]	y [m]	aktiv
1	1	1	K	☐	25					2,00	0,00	☑
2	2	1	K	☐	25					0,00	4,00	☑

Geschoss 1

Column number

The column number is used to identify the column in the floor and can only be assigned once.

Material

By entering a material number, a previously defined material can be assigned to a support. The F5 key can be used to select from the list of available materials.

Type

Selection of the cross-section type (rectangle or circle, optionally hollow cross-section).

Hollow cross-section

If the Hollow cross-section option is selected, the selected cross-section type is treated as a hollow cross-section.

Geometry

Depending on the selected cross-section type (rectangle or circle, optionally hollow cross-section), the width and thickness or the diameter must be entered; for hollow cross-sections, the width and thickness or the inside diameter must also be entered.
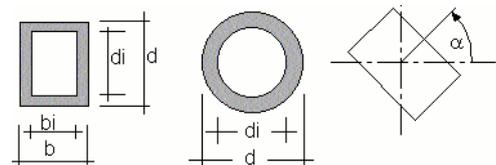
Ill. 9: Column cross-sections

Angle of rotation Phi

If the local coordinate system of the column is rotated relative to the global coordinate system of the story, the rotation angle Phi must be entered.

Arrangement on the floor

The column is placed using X-Y coordinates of its center in the global coordinate system of the floor.

Participation in horizontal stiffening

By removing the marking in the "active" column, individual columns can be excluded from horizontal load transfer.

Symbols

The state of participation in the horizontal load transfer of the current column is adopted for the other columns on the floor.

Copy wall pillars

Copy or delete a column from floor a to floor b (the corresponding dialog is displayed). Existing columns with the same number are overwritten after a corresponding warning. This switch can be used to quickly adopt input values for the same columns in other floors.

Loading

Load case

The horizontal loads acting on the building and the vertical loads assigned to the wall pillars and columns are managed in load cases.

	Lastfallname	Eigen-gewicht	Labili-täts-zahl	zus. VL [kN]	VL Tab [KN]	SumVL Zus+ Tab+G
1	Wind in X-RI	☐	☑	36480	0	36480
2	Wind in Y-Ri	☐	☑	36480	0	36480

Tabs: 1 Load case | 2 Horizontal loads | 3 VL wall piers | 4 VL columns | 5 Notes | 6 Superposition

The load case is named in the load case table. You can also set whether the dead weight of wall pillars and columns is generated automatically.

The sum of the vertical loads entered for each load case on wall pillars and columns is displayed for information.

Load cases can be added (with or without a copy of an existing load case) or deleted using the symbols displayed to the right of the table.

Lability numbers

If the "Lability number" option is selected, the lability numbers are determined according to Brandt, or when using concrete according to DIN 1045 (07-2001), the stiffening criterion according to DIN 1045-1 is determined for the corresponding load case.

Additional vertical loads

Vertical loads that were not recorded during vertical load input can be considered via the "additional VL" column.

Horizontal loads

The horizontal loads of the current load case are entered floor by floor.

The current load case can be changed using the arrow keys next to the load case display.


Px	Resulting in X-direction (positive corresponding to positive coordinate directions)
y	Point of application Px corresponding to coordinate origin
Py	Resulting in Y direction (positive corresponding to positive coordinate directions)
x	Point of application Py corresponding to coordinate origin
Mz	Additional torsional moment around the z-axis (positive clockwise)
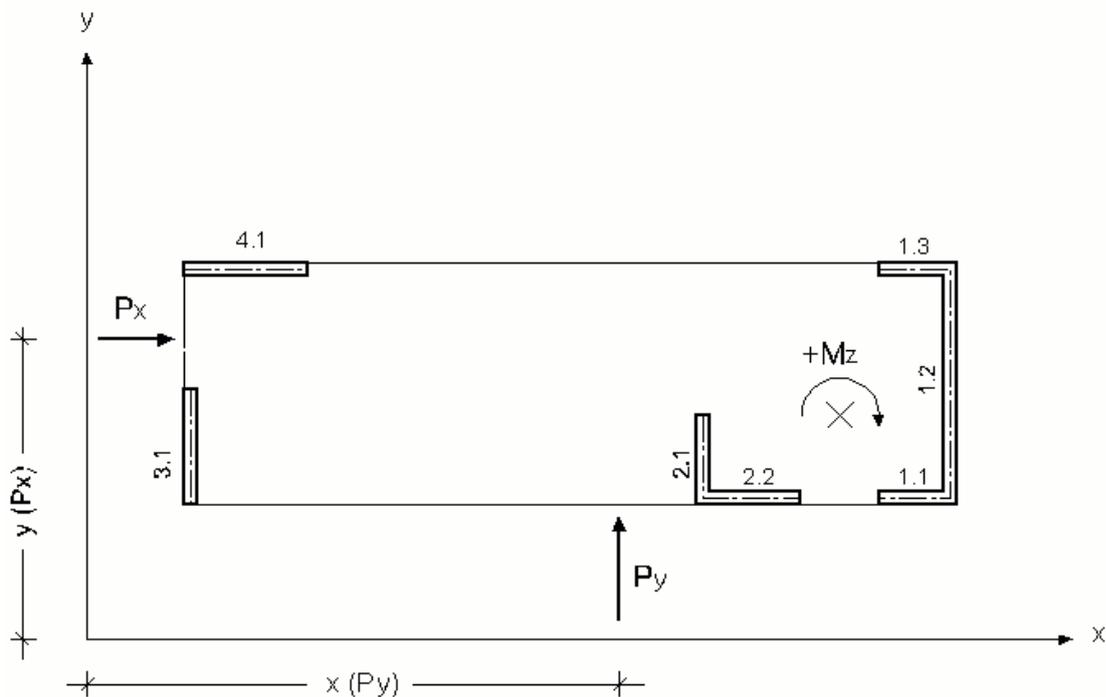
Horizontal loads can be automatically generated as wind in the X-, Y or X+Y direction using the corresponding buttons after entering the wind load parameters.
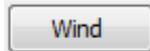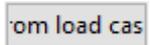
Wind	Generate wind load
om load cas	Copy function "Copy load case"
Delete all	Delete loads in the table
Copy row	Copies the current line to the floors to be selected in another dialog
Factor 1,00 Px * Fak Py * Fak	The sign of Px and/or Py can be changed for all floors using a button. The Px/Py value can be multiplied by a definable "factor".
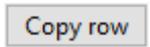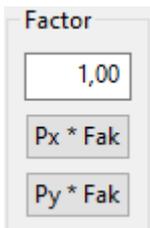
Generate wind load

Call via Loading >> Horizontal loads >> Button "Wind"
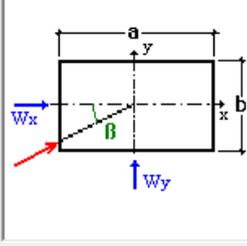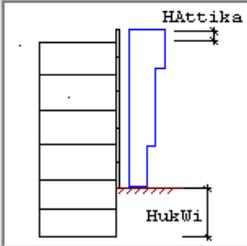
>>	Call up the Building basic data dialog. Enter standard, wind zone, terrain category. Height above sea level.
Eccentricity	The load application point of the automatically determined wind load is shifted by this distance. The entered eccentricity applies approximately to the entire building.
	DIN 1055 Part 4 requires the investigation of eccentricities of 1/10 of the extent of the side of the building exposed to the wind.
HukWi	Bottom edge of wind exposure
	Distance from the TE of the foundation to the lowest wind coordinate
	> 0: lowest wind ordinate shifted upwards, e.g. due to underground building parts
	< 0: lowest wind ordinate shifted downwards, e.g. when examining building parts cut off downwards
HAttika	Distance from the top edge of the roof to the highest wind ordinate.

Force coefficient Aerodynamic force coefficients

DIN 1055-4 (1986):	according to Table 2
$\beta = 0°$ bzw. 90°:	cf = 1,3

DIN 1055-4 (2005)	cf = cpe,10(D)+ cpe,10(E)
	(see DBV example collection DIN 1045-1 Part 2 p. 20-8)

 according to Table 3 applies to:
 - load area A > 10 m²
 - rectangular buildings

$\beta = 0°$ bzw. 90°

h/d >= 1: cf= 1,3

h/d < =0,25   cf= 1,2

Automatically determine wind areas

The height of the wind area below the ceiling level is determined from half the floor height of the current floor, the height of the wind area above the ceiling is determined from half the floor height of the floor above.

The extension in the X and Y directions is determined directly from the enclosing rectangle of the outer contour of the current floor or the floor above.

If wind areas that differ from this are to be considered, the wind areas that can be entered in the

Edit wind areas dialog can be considered if the automatic determination is deactivated.

When you exit the dialog with "OK", the wind load is generated floor by floor using the set parameters → see also Calculation basis - Wind load determination.

Vertical loads wall pillars

The vertical loads (Pz) of the wall pillars of the current floor and the current load case can be entered in the table.

	PfNr	Pz [kN]	ex [cm]	ey [cm]
1	1	0,00	0,0	0,0
2	2	200,00	0,0	0,0

The floor or load case can be changed using the arrow keys next to the display of the current floor (Geschoss 2) or load case.

For each wall pillar, a vertical load with corresponding eccentricities (ex, ey) to the center of gravity of the wall pillar can be entered.

Using the corresponding buttons, it is possible to delete all vertical loads of a load case or to adopt the vertical loads of another load case.

Furthermore, the vertical loads of a wall pillar or of all wall pillars of a floor can be copied to other floors.
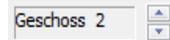
The floors into which the loads are to be copied can be marked using a selection dialog. Loads already present there will be overwritten.

Vertical loads columns

The vertical loads of the columns of the current floor and the current load case can be entered in the table.

	StNr	X [m]	Y [m]	Pz [kN]
1	1	2,00	0,00	0,00
2	2	0,00	4,00	0,00

The floor or load case can be changed using the arrow keys next to the display of the current floor

Geschoss 2

A vertical load (load application at the centre of gravity) can be entered for each column.

Using the corresponding buttons it is possible to delete all vertical loads of a load case or to adopt the vertical loads of another load case.

Furthermore, the vertical loads of a column or of all columns of a floor can be copied to other floors.

The floors into which the loads are to be copied can be marked using a selection dialog. Loads already present there will be overwritten.

Vertical load components on unrecorded structural elements (e.g. columns that are not relevant for stiffening because they do not contribute to the structure) can be recorded separately (via Vertical loads wall pillars) so that the obliquity loads can be correctly determined later.

Superposition

Predefined superpositions can be created from the entered load cases.

Each load case is assigned a factor that is initially zero.
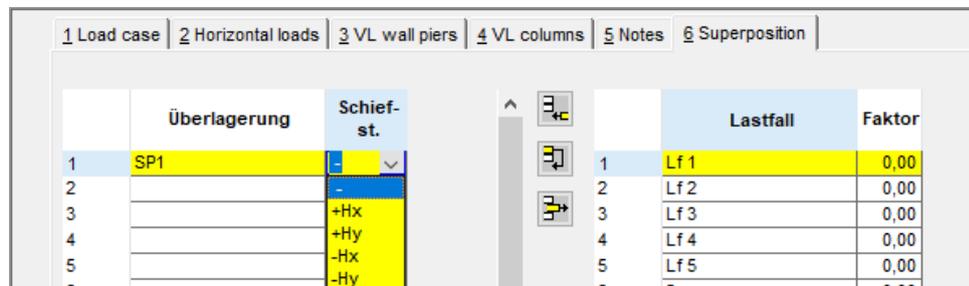
	Überlagerung	Schief-st.			Lastfall	Faktor
1	SP1	-		1	Lf 1	0,00
2		-		2	Lf 2	0,00
3		+Hx		3	Lf 3	0,00
4		+Hy		4	Lf 4	0,00
5		-Hx		5	Lf 5	0,00
		-Hy				0,00

A superposition can be assigned an obliquity in the X or Y direction.

An obliquity is generated from the vertical loads of the superposition according to DIN 1045 7/88 equation 5 or according to DIN 1045-1 equations 4 and 5.

The following standards are supported:

- DIN 1045 7/88: according to Eq. 5
- DIN 1045-1 according to Eq. 4 and 5
- EN 1992-1-1 according to Eq 5.1 and Eq. 5.4

    DIN EN 1992-1-1/NA: 2010-08

    ÖNorm B 1992-1-1:2007-02-01

    NA to BS EN 1992-1-1:2004

The results are displayed as for a normal load case. In addition, there is the option of outputting the extreme stresses of all superpositions.

The overall output of the superpositions is controlled with a separate output section, which largely corresponds to that of the load cases. It also contains a member section related to the superpositions.

Testing the input data

In the info window (under the main selection) there is a global display of a correct entry (OK) or an incorrect or incomplete entry (Error). You may be "informed" of entries that deviate from the norm but may well be intentional (see below).

In the event of an error, you will be informed of the incorrect data via corresponding symbols in the main selection.

In the text view, a detailed error log is output after the input data has been displayed.

```
Mat  = 2
Floor= 4
LC   = 6
SU   = 1

Input : ok
CsVal. : --
H-load : --
Stress : --
```
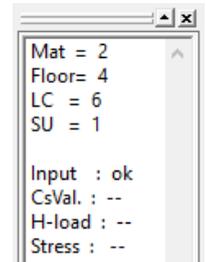
Error:

The values of a material are not allowed !

E-modulus and unit weight must assume values > 0.

An edited wind area is larger than the outer contour !

A manually edited wind area must not be larger than the outer contour.

The geometry of a wall is not permitted !

The length of the wall and its width must be > 0.

The material of a wall is not permitted!

Invalid material reference -> Input table material.

One wall lies within another wall of the pillar !

Invalid wall layout.

There are walls without connection in the pillar !

Only connected walls can be assigned to a wall pillar.

The material of a column is not permitted !

Invalid material reference -> Input table material.

The cross-section of a column is not permitted !

Check internal dimensions of hollow cross-sections.

One type of column is not permitted !

The specified column type cannot be processed by the Wind Loads program.

Bar system not permitted for Q3!

No cross-sectional values can be determined. Check the wall connections in the affected wall pillar!

Calculation with Q3 not successful I!

No cross-sectional values can be determined. Check the wall connections in the affected wall pillar !

Invalid horizontal stiffening, CM= 0 !

Shear walls arrangement does not allow absorption of a torsional moment, system unstable (e.g. shear walls in one line of action).

There must be at least 2 stiffening elements !

System unstable. Since the inherent torsional stiffness of the stiffening elements is neglected, torsional moments must be absorbed by at least two stiffening elements.

Columns intersect with columns or wall pillars !

Check the position of the column.

Notes:

No envelope casing was entered for a floor !

An automatic wind load determination for this is not possible !

There are unequal wall pillars with the same number

This is not recommended for reasons of clarity, but is tolerated.

When displaying a single wall, please note that a selected wall pillar extends over several floors !

An active wall pillar is located outside the envelope!

The wall pillar is treated in such a way that it participates in the load transfer of the horizontal forces

An active column is located outside the floor envelope!

The column is treated in such a way that it participates in the load transfer of the horizontal forces

No load transfer possible !

The conditions for automatic load transfer are not met (Calculation basis). Vertical forces and moments must be entered manually as external loads on the underlying stiffening elements. The possible need to transfer loads via the floor slab must be considered.

Wall pillars intersect with columns or wall pillars !

Check the position; overlapping wall ends may be a desired input. The cross-sectional values determined in this way are calculated using the full wall lengths.

Calculate

Main selection ▸ Calculate or icon [Rec] in the toolbar

When a new position is entered or just read, the user must trigger a "calculate" once, which signals the end of the input or a major change. Any subsequent changes to the input data are immediately recalculated.

The calculation is carried out for the current load case set in the Load case selection (toolbar).

| LC 1: Lf 1 | ∨ | Floor.1: EG | ∨ | show all | ∨ | [Rec] |

The scope of the calculation is controlled by the Output profile of the load case. If this has not yet been edited, a complete output is made.

Before the calculation, the data is checked for plausibility. In the event of an error, an Error log is output instead of the calculation results.

Note:

The Options menu item can be used to set whether the shear center or the center of gravity of the stiffening elements should be used as the basis for the distribution of the horizontal loads. The "centered load transfer" option can also be set (see Calculation basis).

Options   Layout   Window   Grf-Options

Settings -WL Wind Loads (x64)

Colors

Coordinate axis   ▸
✓ Small toolbars
Big toolbars
compute at once
✓ Compute with shear centers
centered load forwarding
Building base data
H-load distribution
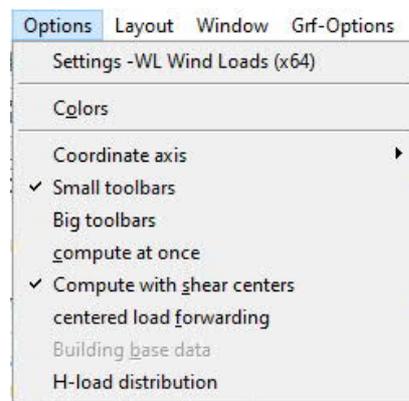
Output options

Graphic view screen

In addition to the graphical input (GrfEdit tab, see Graphical input), there is a second graphical representation on the screen, which you can access via the "Graphic" tab.

GrfEdit	Belastung	Textausgabe	Grafik	System tab.
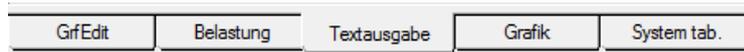
Alternatives: Window ▸ Graphic or

Main selection ▸ Output tab ▸ Screen ▸ Graphic output

Control view in the Graphic

The display of the desired graphic is controlled via the following toolbar:

or the View menu.

View in Y-direction, if necessary with representation of the selected wall pillar

View in X-direction, if necessary with representation of the selected wall pillar

Floor plan of the current floor or the selected wall pillar

Internal force and stress distribution of the selected wall pillar

Stress curve on/off (in the floor plan view, if you display the wall pillars individually. You control the display of the wall pillars via the list selection in the Toolbar.

Load display on/off

Dimension lines on/off

Text larger

Text smaller

Tooltips on/off (in the floor plan view). If the tooltips are switched on, the data of an object is displayed as soon as you position the mouse over the object (wall pillar...) for a certain (short) time (no clicking necessary).

Note: These display functions can also be found in the "View" menu item.
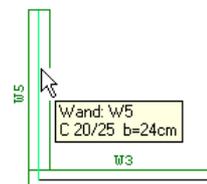
GL   3D graphic view for visual control of the entered system.

View in X or Y direction (▉ ▉)

"Show all" mode (selection list in the toolbar):

show all	▾
show all
pier 1
pier 2
pier 3

Shows the outer edges of the building envelope in the respective direction, the position of the ceilings, the floor heights, the heights from the upper surface of the subfloor to the ceiling and any existing horizontal loads at right angles to the viewing direction.

For horizontal loads that were generated as Wind loads, the course of the ordinate for the underlying wind pressure as well as the height gradation of the wind zones are shown.

The load display and dimensions can be switched on and off using the ▉ and ▉ symbols.
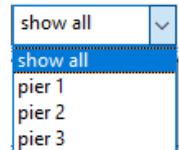
Mode for selected wall pillar or column (WdPf1 …):

Shows the envelope of the selected structural element within the overall view. Instead of the external load, the horizontal loads acting on the element and the vertical loads entered for the structural element are displayed.

In addition, the internal force and stress curve (▉) of the selected member can be displayed.

Floor plan of the current floor or selected wall pillars (▉)

"Show all" mode

show all	▾
show all
pier 1
pier 2
pier 3

Representation of the ceiling floor plan of the current floor (outer contour, wall pillars and columns, horizontal loads of the current load case).
The extent of the outer contour in the X and Y directions is dimensioned. After calculating the current load case, the ceiling pivot point is displayed and dimensioned.

If the tooltip button (▉) in the toolbar is activated, a window with the loads acting at the height of the pillar head is displayed when the mouse is touched on a wall pillar.

For horizontal loads that were generated as wind load, the qualitative course of the wind pressure acting on the floor is shown as a line load. If the upper and lower wind surfaces have different dimensions, two corresponding line loads are shown.

The load display and dimensions can be switched on and off using the symbols ▉ and ▉ .
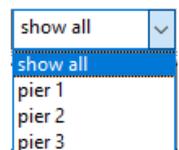
Mode for selected wall pillar or column (WdPf1 …):

The selected structural element is shown in plan view. The shear center is shown and dimensioned. If a calculation with stress analysis has already been carried out for the current load case or superposition, internal forces, maximum and minimum stress as well as a qualitative stress curve are displayed at a selected section. The section selection corresponds to the division selected in the load case output

section. The stress display and dimensioning can be switched on and off using the ▉ and ▉ symbols.

To display the entire floor plan, click "Show all".

Print

If the graphic is the currently active view, it can be printed using the printer icon in the toolbar.

Text view screen

You can access the text view via the corresponding tab.

GrfEdit	Belastung	Textausgabe	Grafik	System tab.

Alternatives: ▸ Window ▸ Text output or

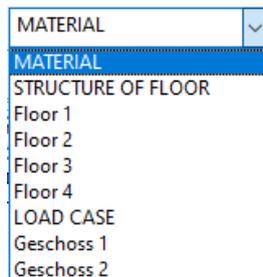   Main selection ▸ Output tab ▸ Screen ▸ Text output

Content:

The output initially includes all input data in table form. After calculating the current load case, the output is expanded to include the cross-sectional values of the stiffening structural elements, the floor plan, pivot points of the floors, the load, horizontal force distribution and stress analysis.

The output section of the system output and the output section of the current load case or overlay also control the extent of the text view on the screen.

The current load case can be selected via the load case selection.

The selection list in the standard toolbar is used to navigate through extensive output.

```
MATERIAL
MATERIAL
STRUCTURE OF FLOOR
Floor 1
Floor 2
Floor 3
Floor 4
LOAD CASE
Geschoss 1
Geschoss 2
```

Output for a selected wall pillar:

Using the selection box for wall pillars, it is possible to display a single representation of a selected wall pillar on the current floor instead of the entire output.

If a calculation with stress analysis has already been carried out for the load case or the superposition, the internal forces and a table with the stresses at all polygon points of the wall pillar are displayed for a selected section.

The section selection corresponds to the division selected in the load case or superposition output section.

A graphic of the wall pillar with the maximum and minimum stresses is shown in the graphic view.

To return to the output for the entire building, click "Show all" in the wall pillar selection box.

Print

If the text view is the currently active view, it can be printed using the printer icon in the toolbar.

Total print

Total print takes place according to the selected output section for system data and load cases.

A total print can be triggered via "Print" in the main selection, Output tab or via "Total print" in the File menu item.

Output section

The scope of the output can be defined using the output section.

You can call up the dialogs for the individual output sections using the menu item ▸ Edit ▸ Output sections or the main selection, tab ▸ Output.

System data

The definition of the output section allows a very differentiated design of the system section. To ensure that the results of the calculation are perfectly understandable in the output, the input data for all floors should be output in full.

The output section of the system data also controls the extent of the text view on the screen.

no output	all checkmarks are removed
output complete	all checkboxes are set
save in item	The marked system data is also saved in the item when saving.

Output section load cases/members

Any load case can be activated for the total print output.

At a minimum, the vertical and horizontal loading are output for each load case.

If a wind load was generated, an associated table is output that documents the wind load determination.

Optionally, an output of the internal forces and stresses of the wall pillars according to the selected division is possible.

With a division of 1, the output section is at the base of the wall pillar. With a division >= 2, the first section is at the height of the top edge of the ceiling of the floor (head of the wall pillar), all others at a distance of $z_i = i *$ floor height/(division - 1). With a division of 2 (standard), the second section is then at the base of the wall pillar, with a division of 3 it is at half the floor height.

The type of stress analysis (with or without tension) can be specified separately for the wall pillars in the Wall pillars table.

By specifying a floor number, the output can only be for a specific floor. All floors are output if "all" is entered.

The graphic is output showing the horizontal loading acting on it.

The output options for a selected wall pillar (view Y, view X, stress curves over the building height, stress curves in section, stress table in section) are part of the "Output section for building elements" (lower section of the dialog).

If -1 is entered in the "Load case" column, the element section is output for all load cases for which the "Elem. section" option was activated.

If an existing load case is selected, the output is for exactly this load case.

Output section superpositions

Any number of superpositions can be activated for the total print output.

At a minimum, the vertical and horizontal loadings are output for each superposition.

If tilting forces have been generated, an associated table is output.

Optionally, the internal forces and stresses of the wall pillars can be output according to the selected division.

With a division of 1, the output section is at the base of the wall pillar. With a division >= 2, the first section is at the height of the top edge of the ceiling of the floor (head of the wall pillar), all others at a distance of $z_i = i *$ floor height / (division - 1). With a division of 2 (standard), the second section is then at the base of the wall pillar, with a division of 3 it is at half the floor height.

The type of stress analysis (with or without tension) can be specified separately for the wall pillars in the Wall pillars table.

By specifying a floor number, the output can be made explicitly for a specific floor. All floors are output when "all" is selected.

The graphic is output showing the horizontal loading acting on it.

The output options for a selected wall pillar (view Y, view X, stress curve over the building height, stress curve in section, stress table in section) are part of the "Output section for building elements" in the lower dialog section.

If -1 is entered for the superposition, the element section is output for all superpositions for which the "Elem. section" option has been activated.

If an existing superposition is selected (e.g. UL1), the output is for exactly this superposition.

Program-specific toolbar

Use the list options in the toolbar to select the active load cases, floors or wall pillars to be displayed.

| LC 1: Lf 1 ⌄ | Floor.1: EG ⌄ | show all ⌄ | Rec |

List selection of current load case

It is used to select the current load case or the superposition when displaying the calculation results in the Text view and Graphic view.

List selection current floor

Selecting the current floor is used to navigate through the floors of a building (graphical input, graphic view of the floor plan on the screen).

In addition, new floors can be created upwards or downwards.

List selection for wall pillars

Selecting a wall pillar switches to the individual representation of this wall pillar in the Graphic view (floor plan) or Text view.

"Show all" returns to the overall representation of the floor plan (graphic view) or the building (text view).

Graphical input

The symbols for "graphical input" are explained in the documentation for Graphical input.

Additional menu items

The standard menu items are described in the document "Menu items". The program-specific (additional) menu items are explained below.

Edit

System	Menu items of the tabular system input
Loading	Menu items of the tabular load input
Output section	System output, load cases
Calculate	When a new item is entered or just read, the user must trigger a "calculation" once, which signals the end of the input or a major change. Any subsequent changes to the input data are immediately recalculated.
FLGeo	Saving and reading input data (see also Graphical input) created with the Frilo Building Model program.
Extreme values	Output of the extreme values of all superpositions.

Options

Calculating with shear centers,

centered load transfer → see chapter Calculate

Basic data building   Height above mean sea level, standard for load assumptions, wind zone.

Input, Grf Options, Tools

These menu items are described in the document "Graphical input".

Literature

/ 1 /	Petersen; „Stahlbau", Viehwegverlag 1990
/ 2 /	„Second order effects in slender concrete structures"; Betonbyggnad 2004