

Version

March 2012

Add-on Module

TOWER

**Generation of Tower Structures
Including Equipment, Effective
Lengths and Loading**

Program Description

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically,
electronically, or by any other means, including photocopying –
without written permission of DLUBAL ENGINEERING SOFTWARE.

© Ing.-Software Dlubal
Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

Contents		Page	Contents		Page
1.	TOWER Structure	5	6.12	Export	42
1.1	Introduction	5	7.	General Functions	43
1.2	TOWER Team	6	7.1	Delete a TOWER Case	43
1.3	Using the Manual	6	7.2	Units and Decimal Places	43
1.4	Open the Add-on Module TOWER	7	8.	TOWER Effective Lengths	44
2.	Input Data	8	8.1	Introduction	44
2.1	Tower Type	8	8.2	Start the Add-on Module	44
2.2	Cross-sections	10	9.	Input Data	46
2.3	Tower Segments	13	9.1	General Data	46
2.4	Vertical Bracings	15	9.2	Details	47
2.4.1	User-defined Bracing Types	17	10.	Generated Data	50
2.5	Horizontal Girts	18	10.1	Effective Lengths - Truss Members	50
2.6	Horizontal Bracing	19	10.2	Effective Lengths - Non-Truss Members	51
2.7	Inner Bracings	20	10.3	Export of Results	51
2.8	Cross Arms	21	11.	General Functions	52
3.	Generated Data	23	11.1	Delete a TOWER Case	52
3.1	Member Releases	23	11.2	Units and Decimal Places	52
3.2	Member Rotations	24	12.	TOWER Loading	53
3.3	Parts List	25	12.1	Introduction	53
4.	General Functions	26	12.2	Open the Add-on Module	53
4.1	Delete a TOWER Case	26	13.	Input Data	55
4.2	Units and Decimal Places	26	13.1	General Data	55
5.	TOWER Equipment	27	13.2	Dead Load	56
5.1	Introduction	27	13.3	Wind Load - Part 1	57
5.2	Open the Add-on Module	27	13.4	Wind Load - Part 2	59
6.	Input Data	29	13.5	Determination of Gust Response Factor	60
6.1	General Data	29	13.6	Shielding	61
6.2	Platforms	30	13.7	Ice Load - Icing Class G	62
6.3	Creating User-defined Platforms	31	13.8	Ice Load - Icing Class R	63
6.4	Tubular Extension	34	13.9	Details	64
6.5	Antenna Brackets	35	13.10	Variable Loads	64
6.6	Antenna Groups	36	14.	Results	65
6.7	Antennas	37	14.1	Load Cases	65
6.8	Antenna Spare Areas	38	14.2	Self-weight and Ice Weight Loads	66
6.9	Inner Ducts	39	14.3	Wind Loads - Gust Reaction Factor	66
6.10	Cable Ducts	40	14.4	Wind Loads - Tower	67
6.11	Ladders	41			

Contents

	Contents	Page		Contents	Page
14.5	Wind Loads - Horizontal Bracings	68			
14.6	Wind Loads - Restrictions	69			
14.7	Export of Results	69			
15.	General Functions	70			
15.1	Delete a TOWER Case	70			
15.2	Units and Decimal Places	70			
A	Literature	71			
B	Index	72			

1. TOWER Structure

1.1 Introduction

DLUBAL ENGINEERING SOFTWARE offers this powerful add-on module to civil and structural engineers, providing them with a tool used to create structures for lattice towers. Take advantage of Dlubal's TOWER add-on modules to generate triangular or quadrilateral tower constructions which can also be spatially stiffened.

With its clear structure and its intuitive input tables, the program represents an innovative software tool making structural analysis easier. In RSTAB or RFEM, it is possible to create complex geometries of 3D tower structures in no time, but you can also use the TOWER add-on module to modify an already existing structure without problems. The manual describes the individual tables by presenting examples.

Like other add-on modules, TOWER is completely integrated in RSTAB or RFEM.

We hope you will enjoy working with the TOWER programs.

Your team from DLUBAL ENGINEERING SOFTWARE

1.2 TOWER Team

The following people were involved in the development of TOWER:

Program coordination

Dipl.-Ing. Georg Dlubal
Ing. Evžen Haluzík
Ing. Vladimír Pátý

Dipl.-Ing. (FH) Younes El Frem
Ing. Pavol Červeňák
Dipl.-Ing. (FH) Wieland Götzler

Programming

David Schweiner
Petr Oulehle
Ing. Ph.D. Jaromír Křížek

Ing. Martin Budáč
Ing. Vladimír Pátý

Cross-section and material database

Ing. Ph.D. Jan Rybín

Program design, dialog figures and icons

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch

Program supervision

Ing. Evžen Haluzík
Ing. Jakub Harazín

Dipl.-Ing. (FH) René Flori
Jozef Krčmárik

Manual, help system and translation

Dipl.-Ing. (FH) René Flori
Ing. Ladislav Kábrt

Mgr. Ing. Hana Macková
Dipl.-Ü. Gundel Pietzcker

Technical support and quality management

Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) Wieland Götzler
Dipl.-Ing. (FH) René Flori

M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

Topics like installation, graphical user interface, results evaluation and printout are described in detail in the manual of the main programs RSTAB and RFEM. The present manual focuses on typical features of the TOWER add-on modules.

The descriptions in this manual follow the sequence of the single input and results tables as well as their structure. The different add-on modules are presented in the following order: **TOWER Structure**, **TOWER Equipment**, **TOWER Effective Lengths** and **TOWER Loading**.



The text of the manual shows the described **buttons** in square brackets, such as [Apply]. At the same time, they are pictured on the left. In addition, **expressions** used in dialog boxes, tables and menus are set in *italics* to clarify the explanations.

Finally, you find an index at the end of the manual. However, if you don't find what you are looking for, please check our website www.dlubal.com where you can go through our *FAQ pages*.

1.4 Open the Add-on Module TOWER

RSTAB and RFEM provide the following options to start the add-on module **TOWER Structure**.

Menu

To start the program in the menu bar,

point to **Towers** on the **Add-on Modules** menu, and then select **TOWER Structure**.

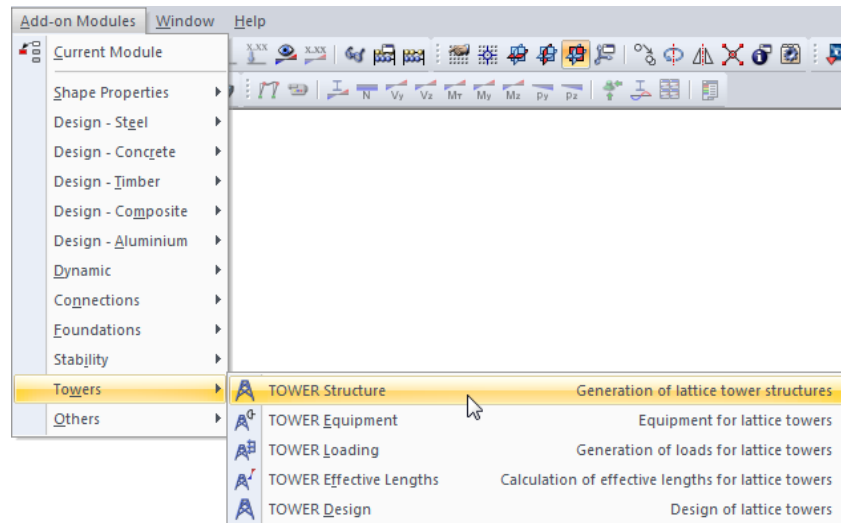


Figure 1.1: Menu: *Add-on Modules* → *Towers* → *TOWER Structure*

Navigator

To start TOWER Structure in the *Data* navigator,

select **TOWER Structure** in the **Additional Modules** folder.

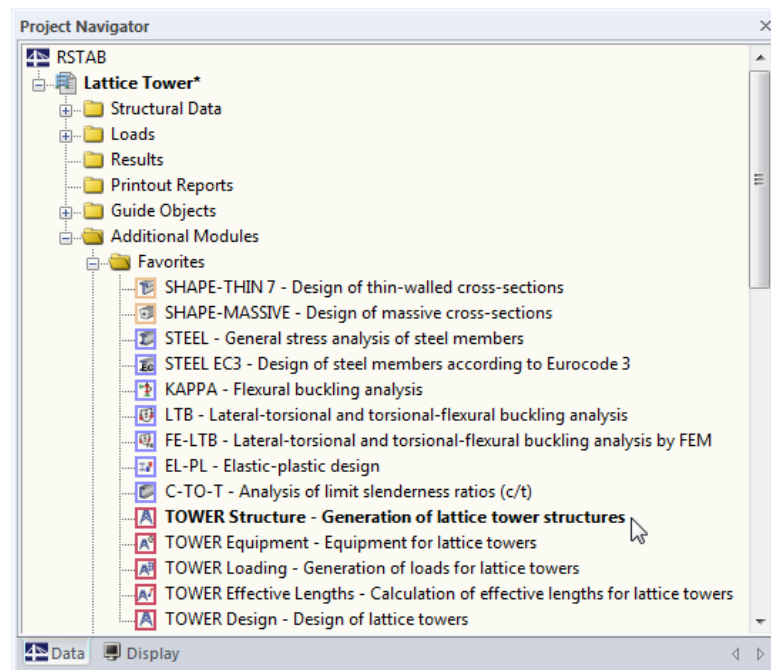


Figure 1.2: Data navigator: *Additional Modules* → *TOWER Structure*

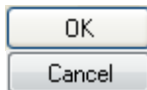
2. Input Data

The input for defining structural data is entered in tables.

When you have started the TOWER add-on module, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently.



To select a table, click the corresponding entry in the TOWER navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.



To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the module but without saving the data.

2.1 Tower Type

In table 1.1 *Tower Type*, you enter basic data required to define the shape of the tower structure.

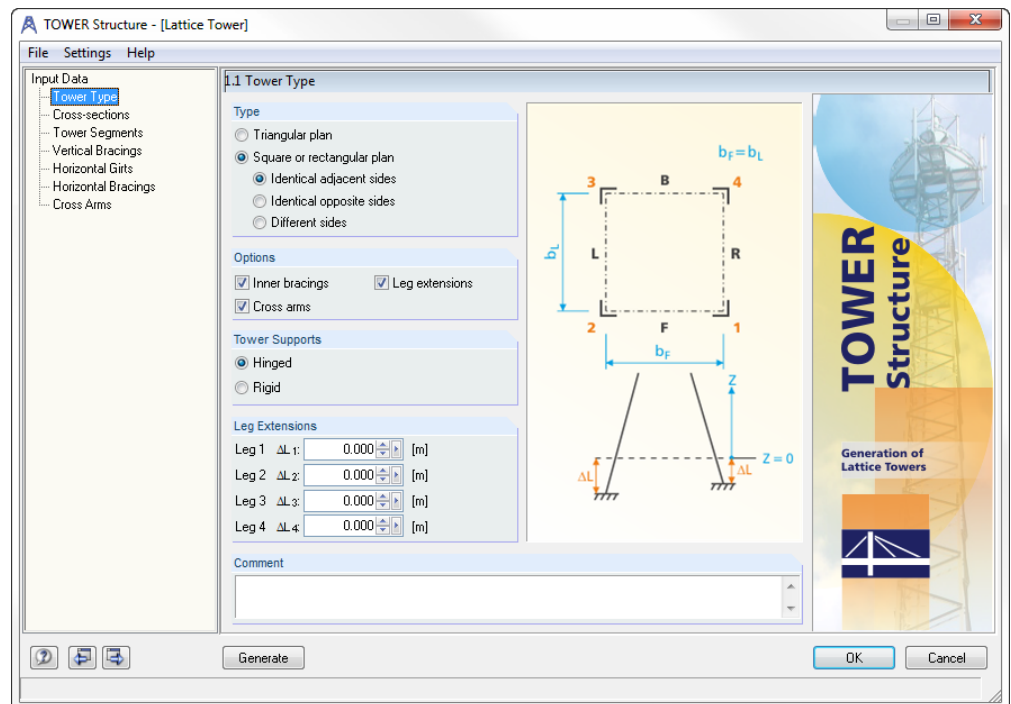


Figure 2.1: Table 1.1 *Tower Type*

Type

In this dialog section, you define the number of sides of the lattice tower. If the tower has a *Square or rectangular plan*, you also have to specify the width-to-height ratios. When you select the option *Identical adjacent sides*, TOWER Structure creates a tower whose horizontal planes have the shape of a square. If the sides are not of the same lengths, input table 1.5 appears additionally in the navigator. You can use this table to define additional vertical bracings.

It is also possible to generate four different tower sides in relation to their bracing. By activating the option *Different sides*, tables 1.6 and 1.7 are enabled. Tables 1.4 to 1.7 are described in chapter 2.4, page 15.

Options

When you want the vertical bracings to be additionally supported, you can activate the option *Inner bracings*. The corresponding table is described in chapter 2.7, page 20.

For calculations of transmission towers or towers for power lines you can also select *Cross arms*. When the check box is ticked, an additional input table becomes visible. The input options are described in chapter 2.8 on page 21.

Tower Supports

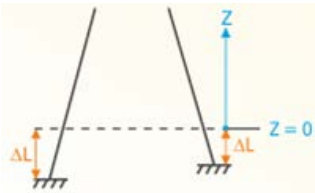
With the settings in this dialog section you define the support type that will be generated later in RSTAB. You can select hinged supports or a full restraint.

Leg Extensions

Often, tower structures are built in areas where top edges are not uniform for all three or four legs. Therefore, TOWER Structure provides the option *Leg Extensions* that you can use to specify an offset height ΔL for each leg.

Comment

In this input field, you can enter user-defined notes describing in detail, for example, the current tower structure.



2.2 Cross-sections

In table 1.2 *Cross-Sections*, you define the cross-sections and materials that you want to use. The entered cross-sections will be available in all subsequent tables.

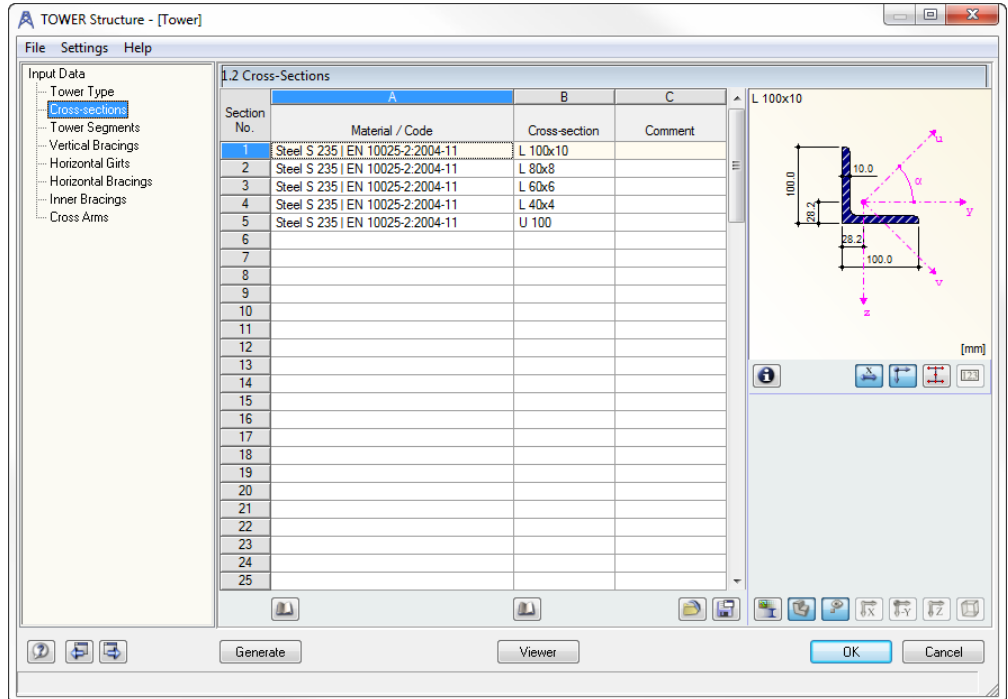


Figure 2.2: Table 1.2 *Cross-Sections*

To modify a cross-section, enter the new cross-section description directly into the corresponding table row. You can also select the new cross-section from the library. To open the library, use the button [Import Cross-section from Library] below the table. Alternatively, place the pointer in the respective table row and click the [...] button, or use the function key [F7]. The library's cross-section table that you already know from RSTAB appears.

The cross-section table that you define in input table 1.2 can be saved as data record so that you can import it, if necessary, to any other TOWER case.

The selection of cross-sections from the library is described in detail in chapter 5.3 of the RSTAB manual, or chapter 5.13 in the manual for RFEM.



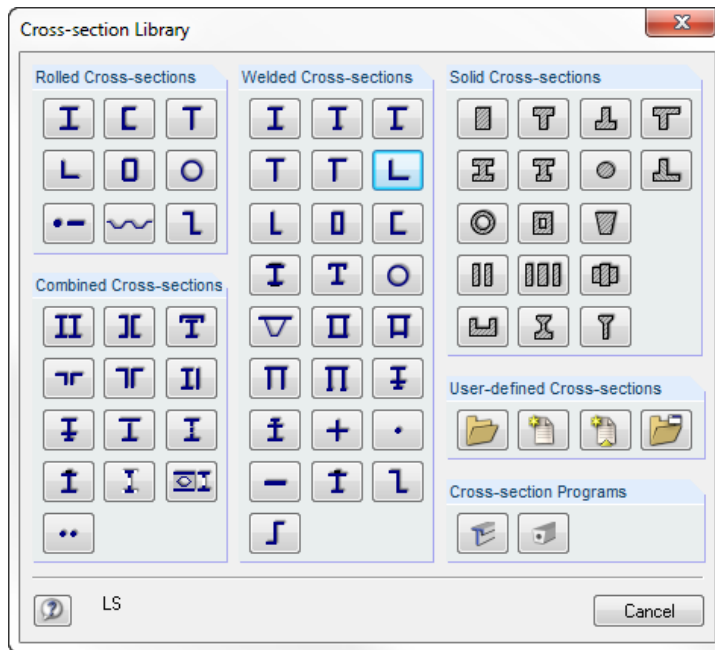


Figure 2.3: Cross-section Library

Info about cross-section



By using the [Info] button you can check additional detail information of each cross-section. A dialog box with cross-section properties appears.

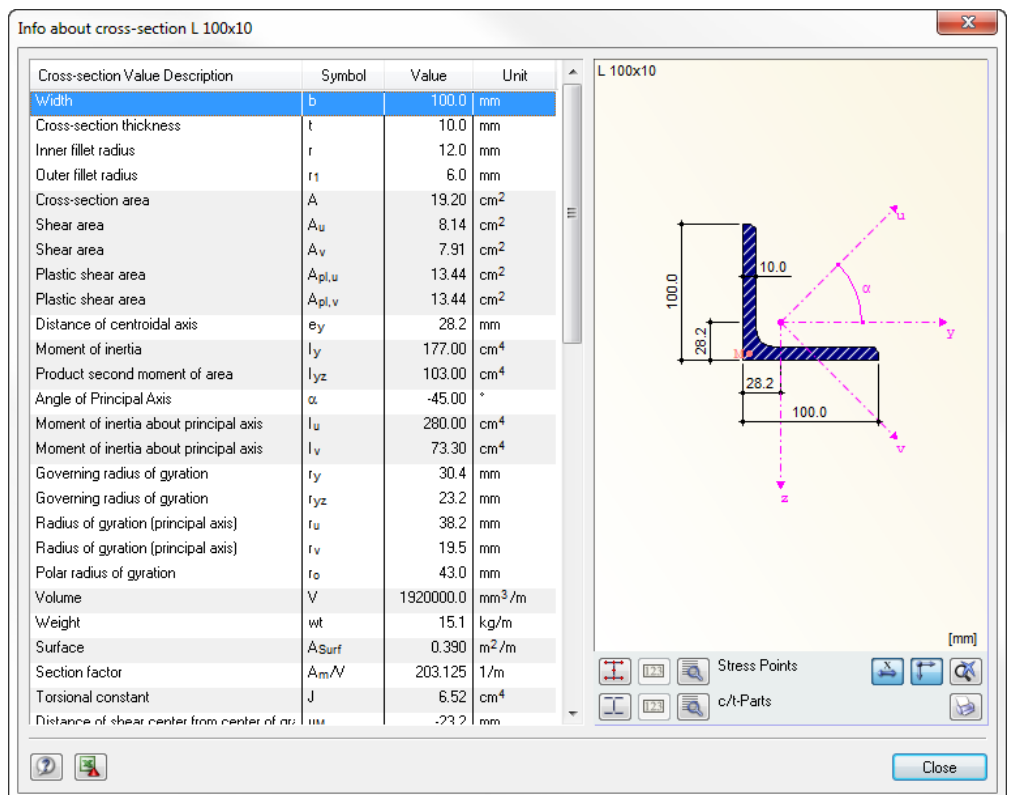


Figure 2.4: Dialog box Info about cross-section

In addition, it is possible to display *Stress Points* and *c/t-Parts* for the *c/t* design. The right part of the table shows the currently selected cross-section graphically.

The buttons below the graphic are reserved for the following functions:

Button	Function
	Displays or hides the stress points.
	Displays or hides the (c/t) cross-section parts.
	Displays or hides the numbering of stress points or (c/t) cross-section parts.
	Displays or hides the details of stress points or (c/t) cross-section parts.
	Displays or hides the dimensions of the cross-section.
	Displays or hides the principal axes of the cross-section.
	Displays the entire cross-section.
	Enables a function for printing the cross-section information.

Table 2.1: Buttons of cross-section graphic

Material / Code

In column A of table 1.2, you can select the *Material* directly by using the pull-down menu. The materials of the standard or code group specified in the library are available for selection.

Material Library

Numerous materials are available in the library. To open the library, click the buttons shown on the left or use the function key [F7].

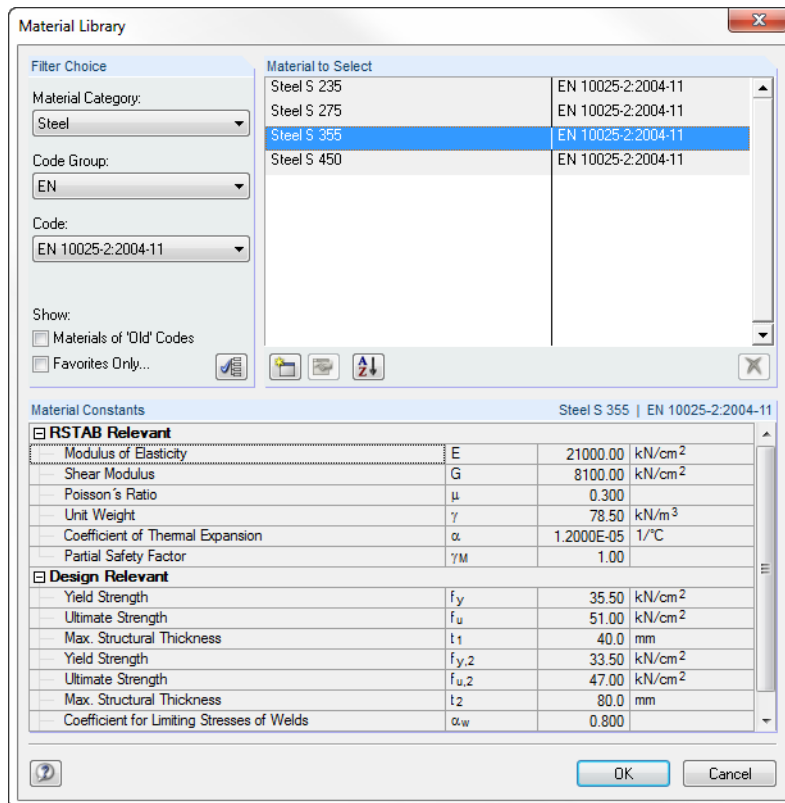
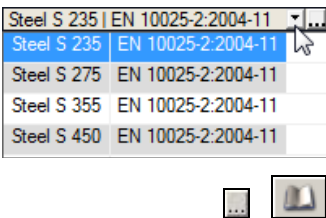


Figure 2.5: Dialog box *Material Library*

OK

In the *Filter Choice* section, *Steel* is preset as material category. You can select a material from the list *Material to Select* on the right and check the corresponding parameters in the lower part of the dialog box. Click [OK] or use the [↵] button to import the selected material to table 1.2 of the add-on module.

Chapter 5.2 of the RSTAB manual describes in detail how materials can be filtered, added or rearranged.

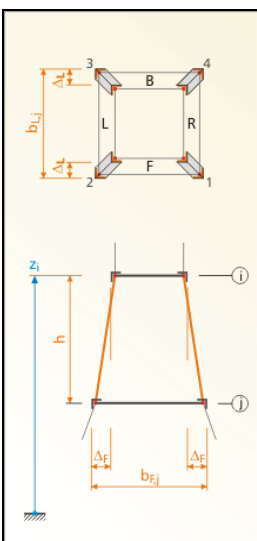
2.3 Tower Segments

The legs of lattice towers may be arranged differently along the tower height. Often, they are not parallel in the bottom area, and the tower is tapered towards the top. In table 1.3 *Tower Segments*, you have the possibility to define the construction's geometry by means of tower segments.

Segment No.	A		B		C		D		E		F	
	Level Top zi [m]	Height h [m]	Bottom Width [m]		Width Change [mm/m]							
			bFj	bLj	ΔF	ΔL						
1	20.000	5.000	2.250	2.250	50.00	50.00						
2	15.000	5.000	2.500	2.500	50.00	50.00						
3	10.000	5.000	2.750	2.750	50.00	50.00						
4	5.000	5.000	3.000	3.000	50.00	50.00						
5												
6												

Leg No.	A		B		C		D	
	Level Top zi [m]	Height h [m]	Cross-Section		Comment			
1	10.000	5.000	1 - L 100x10					
2								
3								
4								

Figure 2.6: Table 1.3 Tower Segments



Total Dimensions

In the upper part of the table, you define the *Overall height H*. In addition, you have to define the starting width of the first tower segment b_F . For towers with a square cross-section the input field b_L is deactivated.

Slopes via

You have two selection options. You can specify the respective starting width of a tower segment. Then, the add-on module will determine the inclination by means of segment height and difference of widths. But it is also possible to specify the slope by a width change set in [mm/m]. In this case, the widths of tower segments will be determined automatically.

On the right, a scheme is displayed for better understanding, showing the variables used in the input tables.

Tower Segments

In the central table, you define the tower segments by their height h , the width b or the width change. Depending on the previous setting for the slope definition, the program enables the relevant part in the table.

Tower Segment - Cross-sections

You have to assign a cross-section to each portion of the tower. Select a tower segment in the central table, and then assign a cross-section from column C in the lower table. The assignment is based on the cross-sections defined in table 1.2 *Cross-Sections*. If you want to use different cross-sections for a tower segment, you first have to define the partial heights h in column B of the lower table. Then, you can assign a cross-section as usual. For a large number of subdivisions you can use the button [=] shown on the left to assign identical cross-sections to subsequent table rows.

To visualize structural data already entered, several buttons are available. The following table lists the buttons and describes their functions.








Button	Function
	Switches between the rows of the central table.
	Switches between the central and lower table.
	Displays the scheme or model.
	Displays and hides the structure nodes.
	Displays the wire-frame or the rendered model.
	Model view in direction of axes +X, -Y and +Z
	Displays isometric view.

Table 2.2: Buttons for visualization of structural data

2.4 Vertical Bracings

To stabilize the tower legs, you can define vertical bracings in table 1.4 which is divided into two parts.

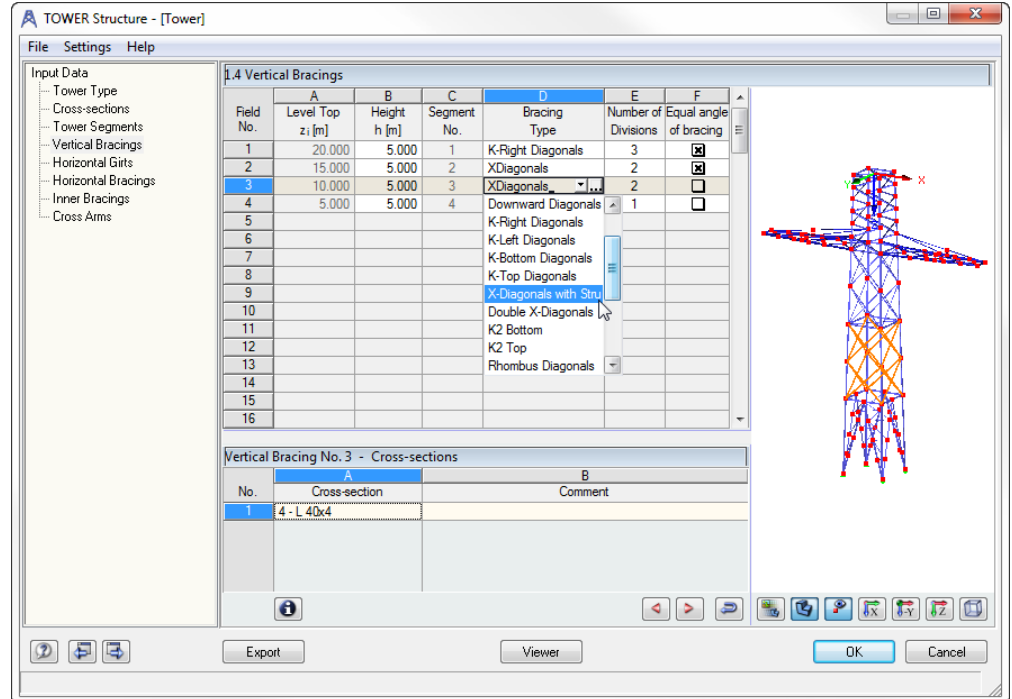


Figure 2.7: Table 1.4 Vertical Bracings

In the upper table, you specify the geometry. The *Height h* is imported from table 1.3 *Tower Segments* as a recommendation. When specifications for the field heights are smaller than for the height of the tower segment, TOWER Structure will automatically create another bracing field.

To facilitate the definition of bracing types, TOWER Structure can access a database where common arrangements of vertical bracings are already stored. To open the database (see Figure 2.8), place the pointer in the relevant row of table column D and click the button [...] or use the function key [F7].



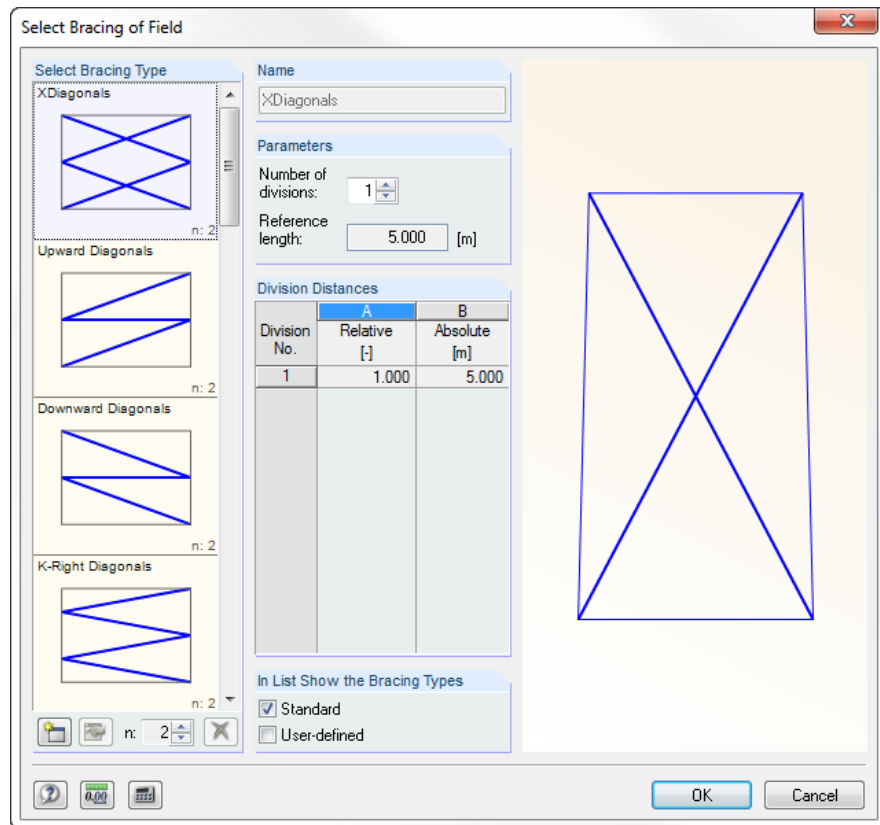


Figure 2.8: Database of bracing geometries for vertical bracings

Parameters and Division Distances

The available data records can be adjusted by means of the parameters *Number of divisions* and *Reference length*. TOWER Structure creates automatically divisions in the table *Division Distances*. Of course, you can change the entries manually. The reference length defined above will be considered and the lengths of the respective divisions will be adjusted.

2.4.1 User-defined Bracing Types

You can extend the database by any user-defined bracing type. With a click on the [New] button shown on the left you open a new window for editing data. To simplify the creation of a user-defined geometry, it is recommended to use a template from the database.

Generating bracing members is based on applying lines referring to reference lines. You define the start and endpoint of a line by means of a relative distance to the start or end of a reference line. All entered data is directly displayed in the graphic to the right. To have a clear overview, it is possible to activate the line numbering and the line orientation by using the two buttons shown on the left.

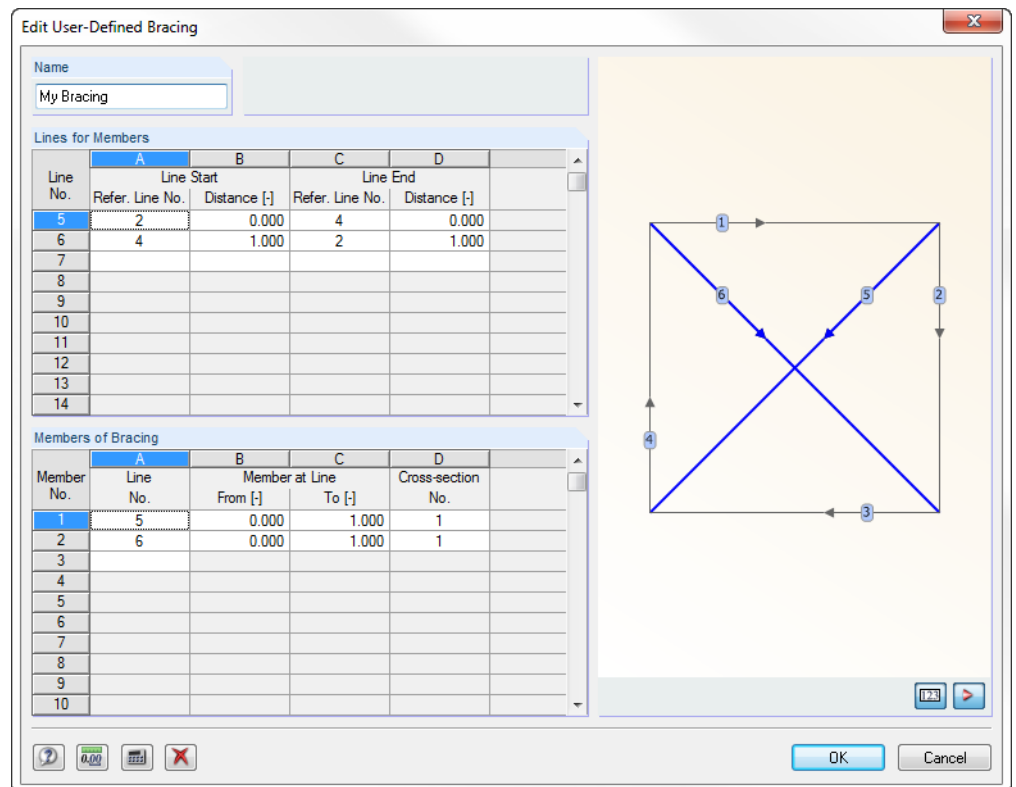
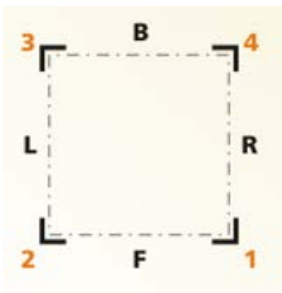
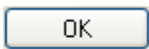


Figure 2.9: Dialog box *Edit User-Defined Bracing*

When you have created a user-defined bracing, you have to enter a name for it. Then, you quit the table with the [OK] button. To close the database, click [OK] again.

Finally, you assign a cross-section to the bracing. Select a bracing field in the upper part of table 1.4. Then, in the lower part, you assign a cross-section.

When the option *Identical opposite sides* was activated in table 1.1, you define the front and back side in table 1.4 *Vertical Bracings - Faces F, B* and the left and right side in table 1.5 *Vertical Bracings - L, R*. When the third option *Different sides* was set, the vertical bracing is defined for sides F, L, B and R in the tables 1.5 to 1.7.



2.5 Horizontal Girts

In table 1.8, you can define horizontal girts at transitions where tower segments are merging.

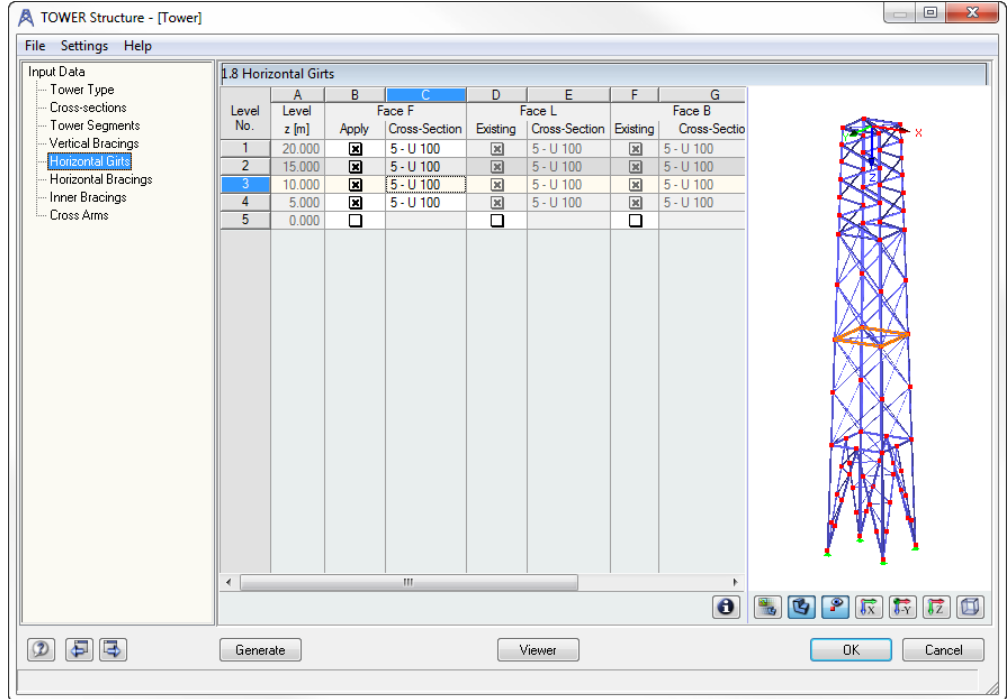


Figure 2.10: Table 1.8 Horizontal Girts

When the tower's side walls are of the same length, table columns D to I are inactive. The definitions from columns B and C are taken over automatically. When you have selected non-identical side lengths in table 1.1 (see chapter 2.1, page 8), you can define horizontal girts for the respective side walls.

2.6 Horizontal Bracing

You can define horizontal bracings of different layouts in the plane of the horizontal girts to insert additional stiffenings.

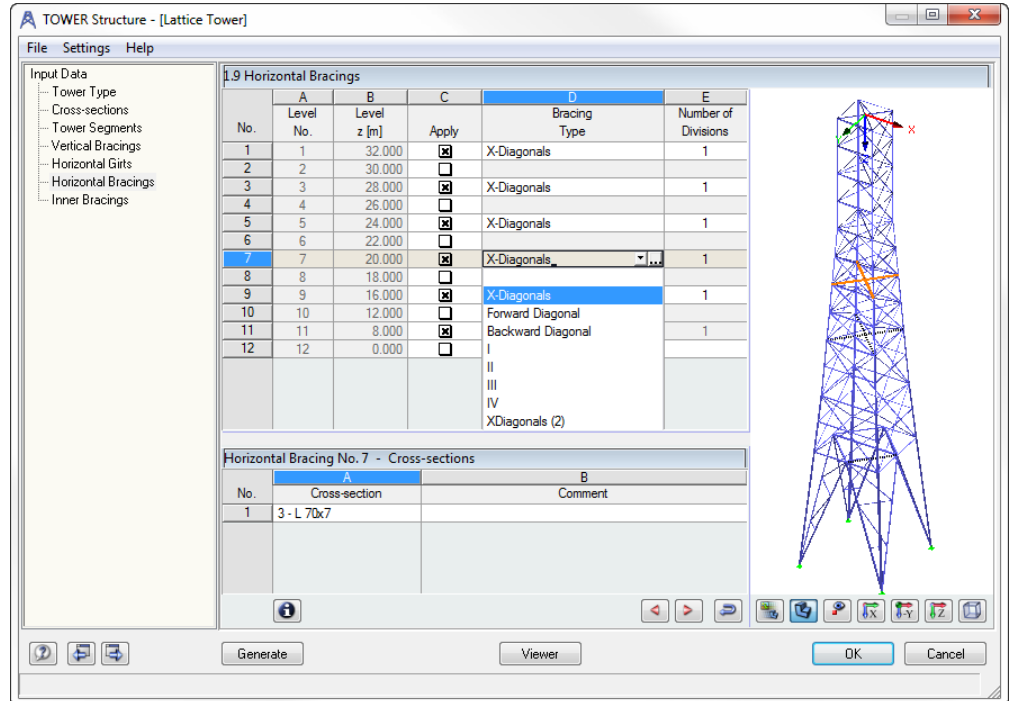


Figure 2.11: Table 1.9 Horizontal Bracings

Again, the TOWER module offers you a database with predefined arrangements of members used for bracings. However, if you want to apply a user-defined horizontal bracing, you can create it in the same way as described in chapter 2.4.1 on page 17.

2.7 Inner Bracings

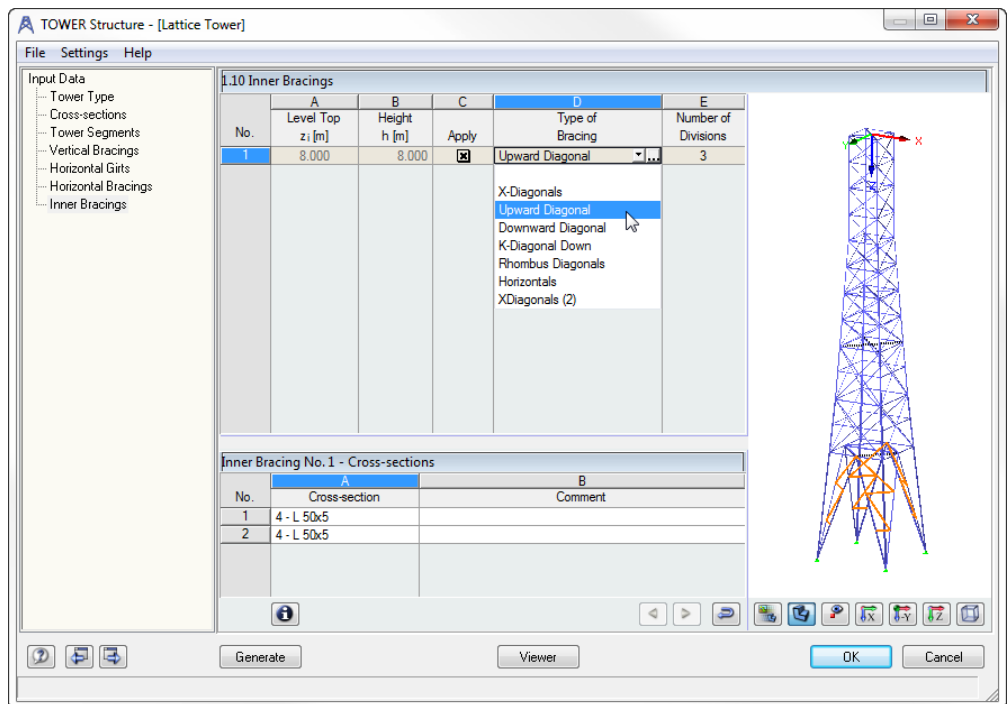
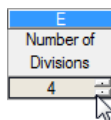


Figure 2.12: Table 1.10 Inner Bracings

If bracing members of side walls are very slender, it may be necessary to reduce the buckling length by structural measures. TOWER Structure provides settings for inner bracings. You define them separately for each tower segment. In this way, it is possible to adjust the stiffenings individually to the given static requirements.



Table 1.10 Inner Bracings is only active if you have selected vertical bracing types (table 1.4 or 1.5 to 1.7) for which an additional stiffening is useful. As the inner bracings are defined identically for all four tower faces, you have to select for all sides a bracing type that allows for an inner bracing when tower sides are not equal in their length.



In the upper part of the table, you activate the available bracings in column C. Then, you assign a bracing type. The number of divisions can be selected in column E where you can enter a figure directly. You can also use the buttons that will be enabled when you click into the table cell.

The database of inner bracings is adjusted to the geometric boundary conditions. Again, it is possible to create user-defined types of bracings. Chapter 2.4 on page 15 describes how to work with the database.

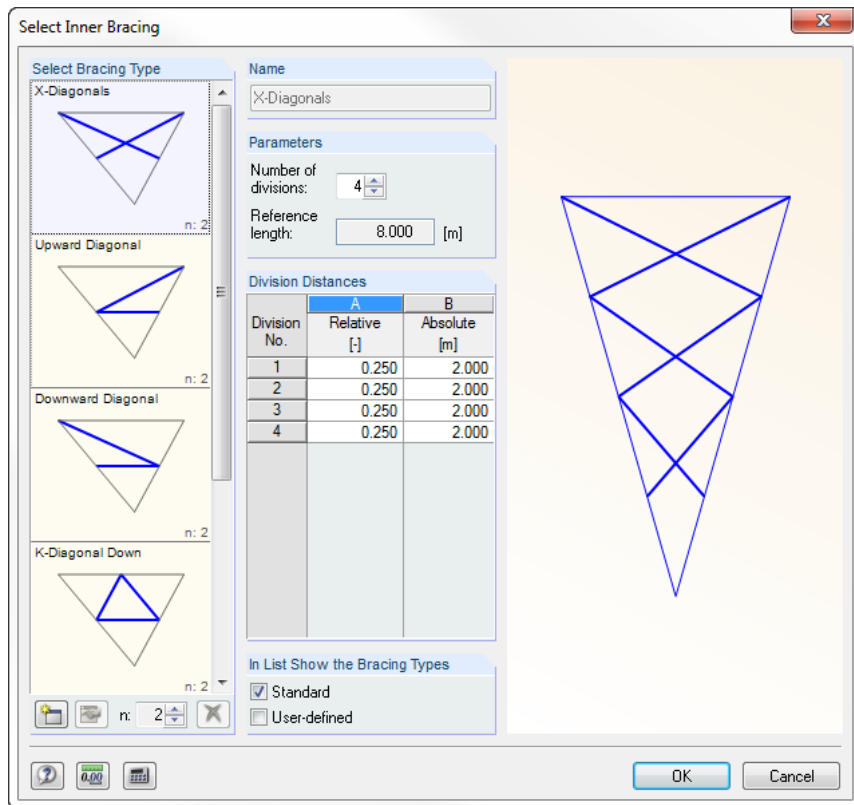


Figure 2.13: Database of bracing geometries for inner bracings

2.8 Cross Arms

In table 1.11 *Cross Arms*, you can generate jibs for supporting power lines. The table is only active if you have ticked the option *Cross arms* in table 1.1.

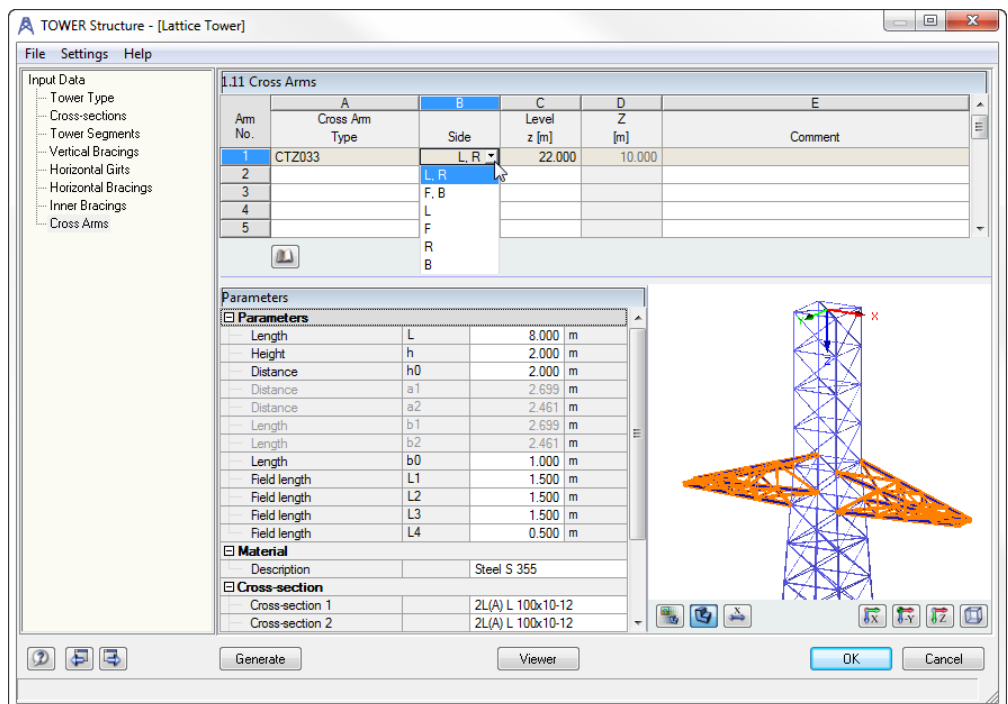


Figure 2.14: Table 1.11 Cross Arms

To facilitate the definition of cross arms, TOWER Structure can access a database where various jib layouts are already stored. To open the database, place the pointer in a relevant row of table column A and click the button [...] or use the function key [F7].

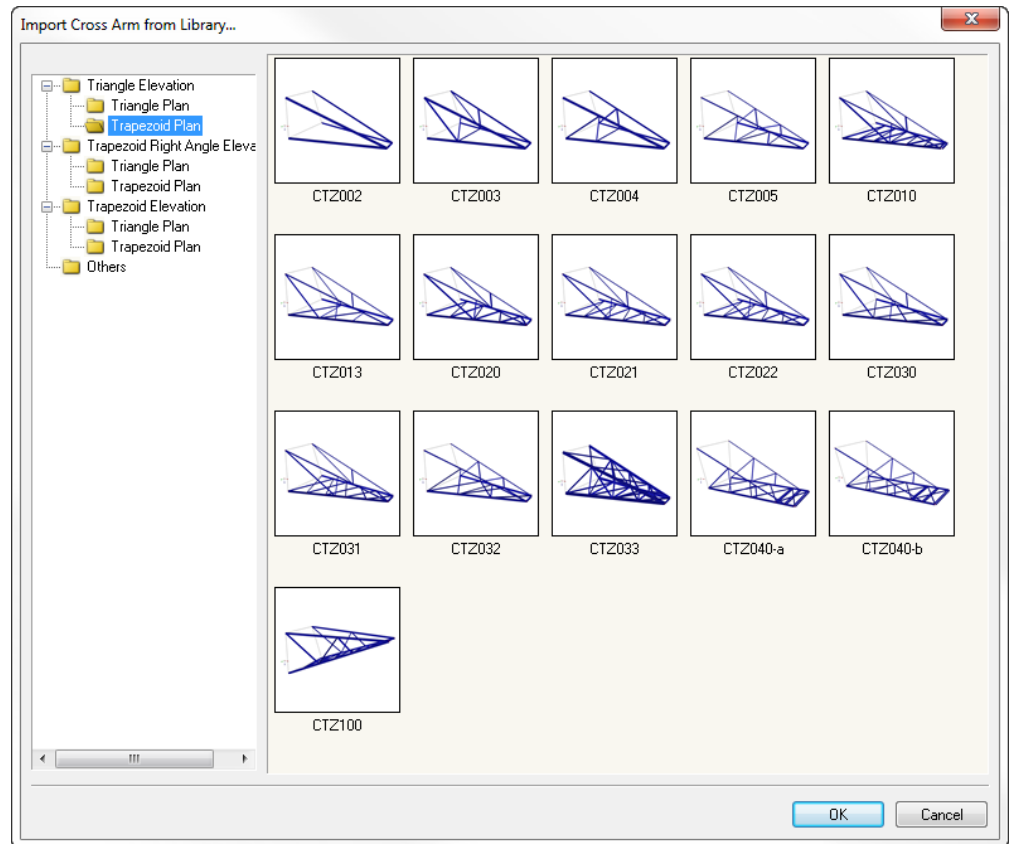


Figure 2.15: Database for Cross Arms

In table column B, you assign the selected arm to the tower sides. Use column C to set the height where the cross arm's position is defined. In the lower part of table 1.11, you can adjust a number of parameters for the jib. In addition to geometric conditions, you can adjust material and cross-sections of individual structural components.

3. Generated Data

Generate

When all data has been completely entered, you start generating the tower structure by clicking the [Generate] button. If the program detects incorrect or missing entries in the input tables, the add-on module displays a corresponding message.

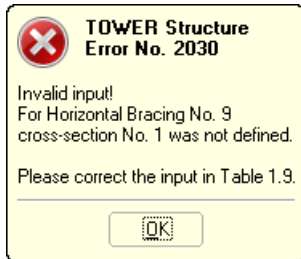


Figure 3.1: Error message

The relevant input table is clearly indicated so that you can correct the entered data quickly.

3.1 Member Releases

When the generation was successful, TOWER Structure displays results table 2.1 showing an overview about member releases.

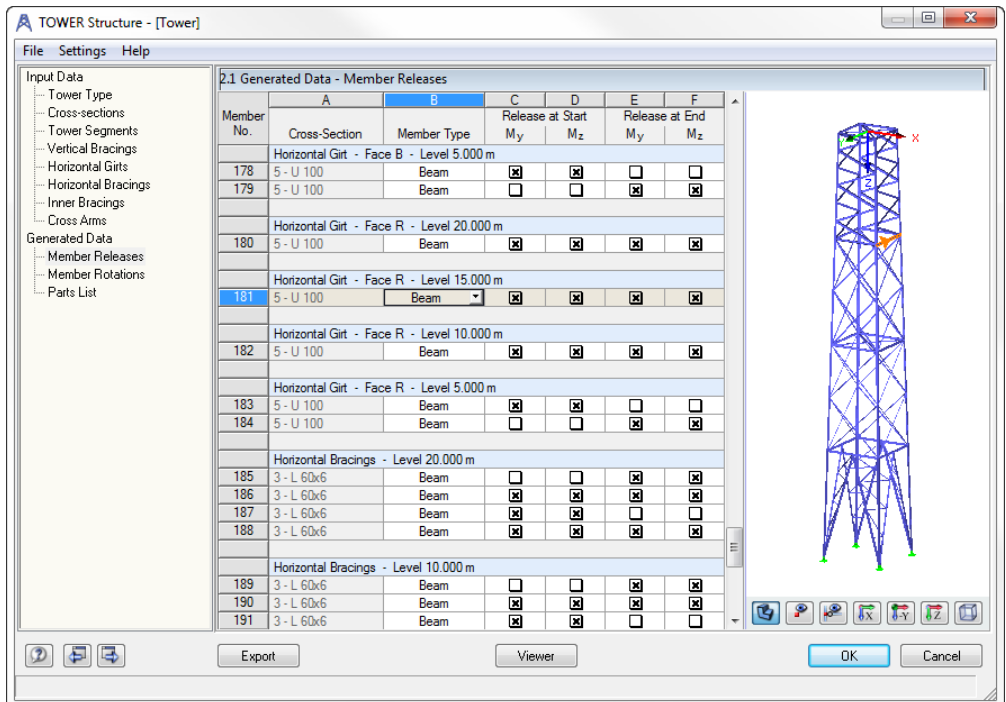


Figure 3.2: Table 2.1 Generated Data - Member Releases

The table is clearly subdivided into individual structural groups. During the generation the add-on module has already assigned member releases to the corresponding structural components. You can adjust them easily according to your needs. You just have to tick or clear the check box of the respective degree of freedom in table columns C to F. It is also possible to modify the member releases after the data export in RSTAB/RFEM.

3.2 Member Rotations

This table shows information about the inclination of the principal axis α and the member rotation angle β .

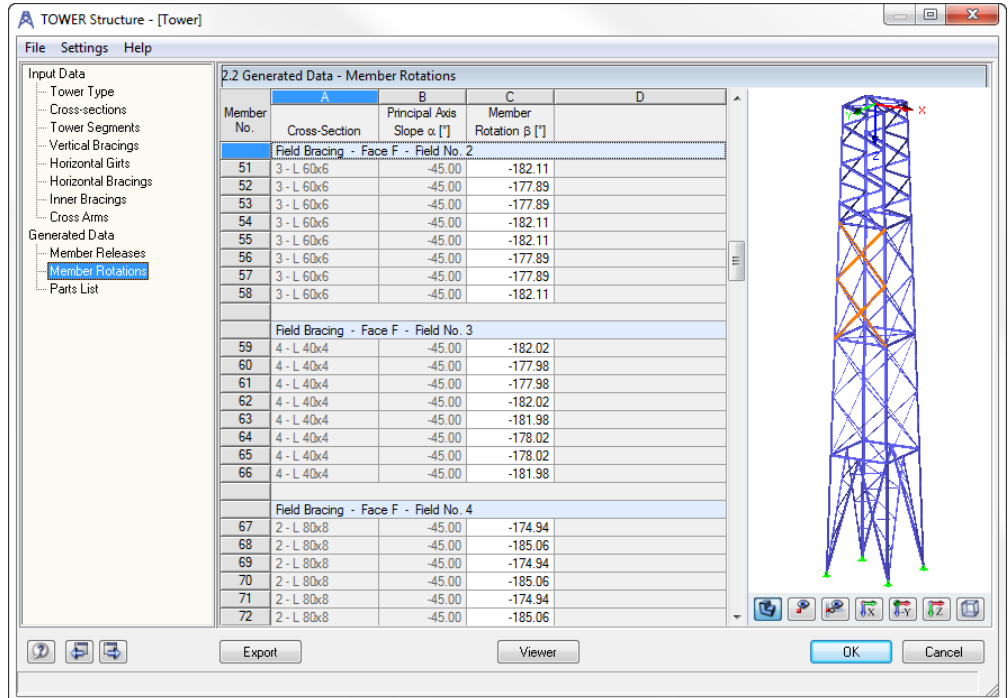


Figure 3.3: Table 2.2 *Generated Data - Member Rotations*

The results are organized in component groups. This subdivision may help you if you want to change the angles of several members of a particular zone. It will be sufficient to enter the numerical value of the first rotation angle that needs to be changed. The subsequent cells can be filled with the same value by means of the [F8] key. The possibility to adjust member rotations manually may be useful when you want to adjust the orientation of angle legs of L-sections in relation to the inclination of the tower sides.

3.3 Parts List

Table 2.3 provides a detailed overview about the cross-sections that are used.

Item No.	A	B	C	D	E	F	G
	Cross-Section	Number of Members	Length [m]	Tot. Length [m]	Surf. Area [m ²]	Volume [m ³]	Unit Weight [kg/m]
9	L 100x10	4	2.157	8.630	3.37	0.02	15.07
10	L 100x10	4	2.120	8.481	3.31	0.02	15.07
11	L 100x10	8	0.868	6.942	2.71	0.01	15.07
12	L 100x10	8	0.833	6.667	2.60	0.01	15.07
13	L 100x10	8	0.800	6.404	2.50	0.01	15.07
14	L 80x8	32	1.303	41.709	12.97	0.05	9.66
15	L 60x6	8	1.798	14.382	3.21	0.01	5.42
16	L 60x6	4	1.728	6.911	1.54	0.00	5.42
17	L 60x6	16	1.703	27.253	6.08	0.02	5.42
18	L 60x6	8	1.614	12.911	2.88	0.01	5.42
19	L 60x6	4	1.374	5.497	1.23	0.00	5.42
20	L 40x4	4	2.102	8.409	1.30	0.00	2.42
21	L 40x4	8	1.858	14.865	2.30	0.00	2.42
22	L 40x4	8	1.814	14.515	2.25	0.00	2.42
23	L 40x4	8	1.772	14.175	2.20	0.00	2.42
24	L 40x4	4	1.749	6.996	1.08	0.00	2.42
25	L 40x4	8	1.726	13.808	2.14	0.00	2.42
26	L 40x4	4	1.467	5.868	0.91	0.00	2.42
27	L 40x4	4	1.428	5.714	0.89	0.00	2.42
28	L 40x4	4	0.952	3.809	0.59	0.00	2.42
29	L 40x4	4	0.476	1.905	0.30	0.00	2.42
30	U 100	4	2.444	9.774	3.64	0.01	10.60
31	U 100	4	2.194	8.774	3.26	0.01	10.60
32	U 100	4	1.944	7.774	2.89	0.01	10.60
33	U 100	8	1.347	10.774	4.01	0.01	10.60
Sum		216		369.351	107.74	0.43	

Figure 3.4: Table 2.3 Generated Data - Parts List

In addition to single lengths and weights, the table shows information about the surface area. The information is helpful for the selection of an anti-corrosion agent.

Export

Export

Finally, the generated data must be exported to RSTAB/RFEM for further processing because it represents the basis for data input in the add-on modules TOWER Equipment and TOWER Loading. To start the export, click the [Export] button.

With the use of the export function, all structural data already existing in RSTAB/RFEM will be overwritten. Therefore, a corresponding warning appears before the export will be carried out.

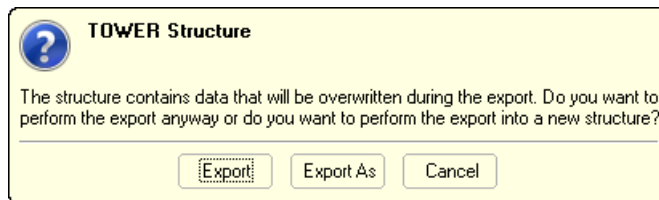


Figure 3.5: Warning before export of data

4. General Functions

You can use the menu bar at the top of the TOWER module window to access various general functions.

4.1 Delete a TOWER Case

To delete a TOWER case,

select **Delete Case** on the **File** menu in the TOWER add-on module.

When you confirm the selection, the program deletes the case and you quit the add-on module. Data that has already been exported is preserved in RSTAB/RFEM.

4.2 Units and Decimal Places

The units and decimal places for RSTAB/RFEM and all add-on modules are managed in one global dialog box. To open the dialog box out of the TOWER module,

select **Units and Decimal Places** on the **Settings** menu.

The following dialog box opens, which you already know from RSTAB/RFEM. The TOWER add-on module is preset.

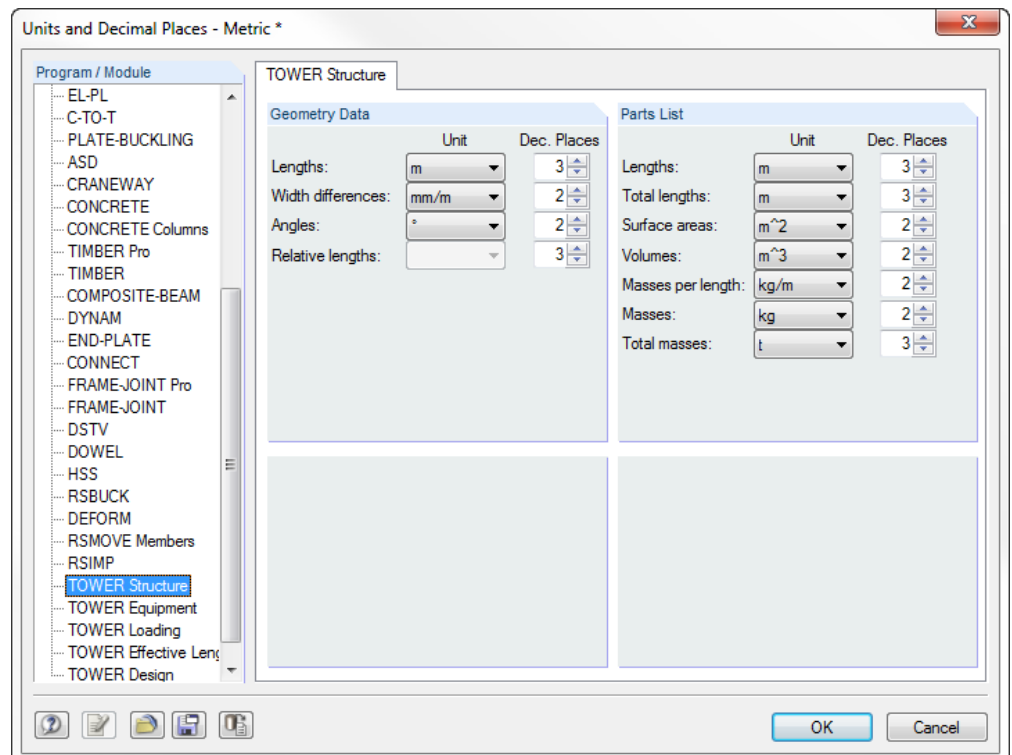


Figure 4.1: Dialog box *Units and Decimal Places*



The settings can be saved as user profile to reuse them in other structures.

5. TOWER Equipment

5.1 Introduction

Tower constructions created in **TOWER Structure** are used as supporting structures for transmitting antennas and further equipment such as platforms, ladders and cable lines. As it is necessary to take into account such structural components, especially antennas, the DLUBAL company has developed the add-on module **TOWER Equipment**. Take advantage of this powerful tool to model in no time a complete transmitter tower including all design-relevant equipment. The simple and clear program structure helps you to enter data. The definition of single components is quite easy due to database-supported input options and informative graphics.

5.2 Open the Add-on Module

RSTAB/RFEM provide the following options to start the add-on module **TOWER Equipment**.

Menu

To start the program in the menu bar,

point to **Towers** on the **Add-on Modules** menu, and then select **TOWER Equipment**.

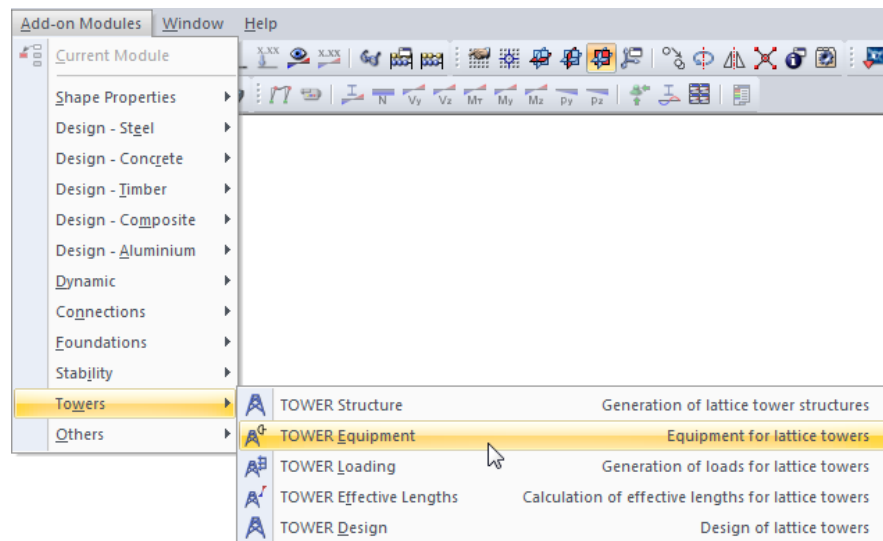


Figure 5.1: Menu: *Add-on Modules* → *Towers* → *TOWER Equipment*

Navigator

To start TOWER Equipment in the *Data* navigator,

select **TOWER Equipment** in the **Additional Modules** folder.

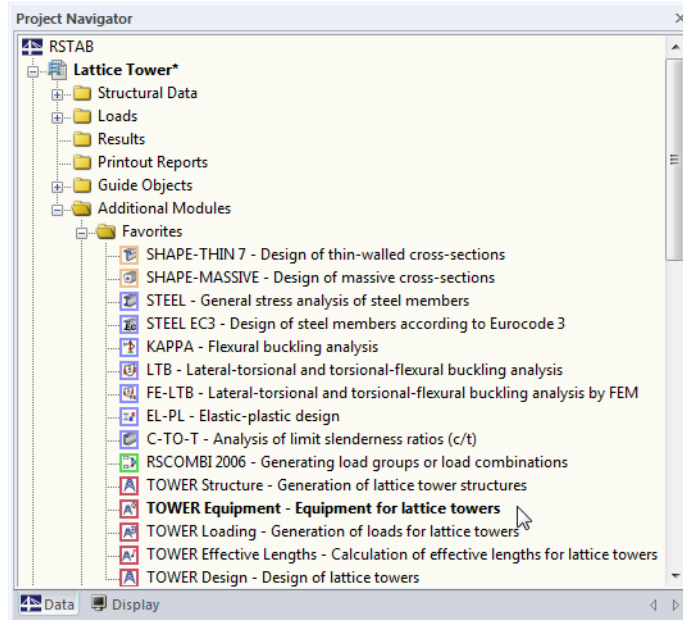


Figure 5.2: Data navigator: *Additional Modules* → *TOWER Equipment*

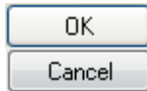
6. Input Data

The input for defining tower equipment is entered in tables.

When you have started the TOWER add-on module, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently.



To select a table, click the corresponding entry in the TOWER navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.



To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the module but without saving the data.

6.1 General Data



Table 1.1 shows you an overview about the structural data imported from RSTAB/RFEM, listed and sorted by structural tower components. When you select a cell in the table *Allocated Members*, you can see the corresponding structural parts highlighted in the model displayed to the right. If the model is not displayed, click the button shown on the left.

The lower part of the table lists the most important geometrical data of the tower construction.

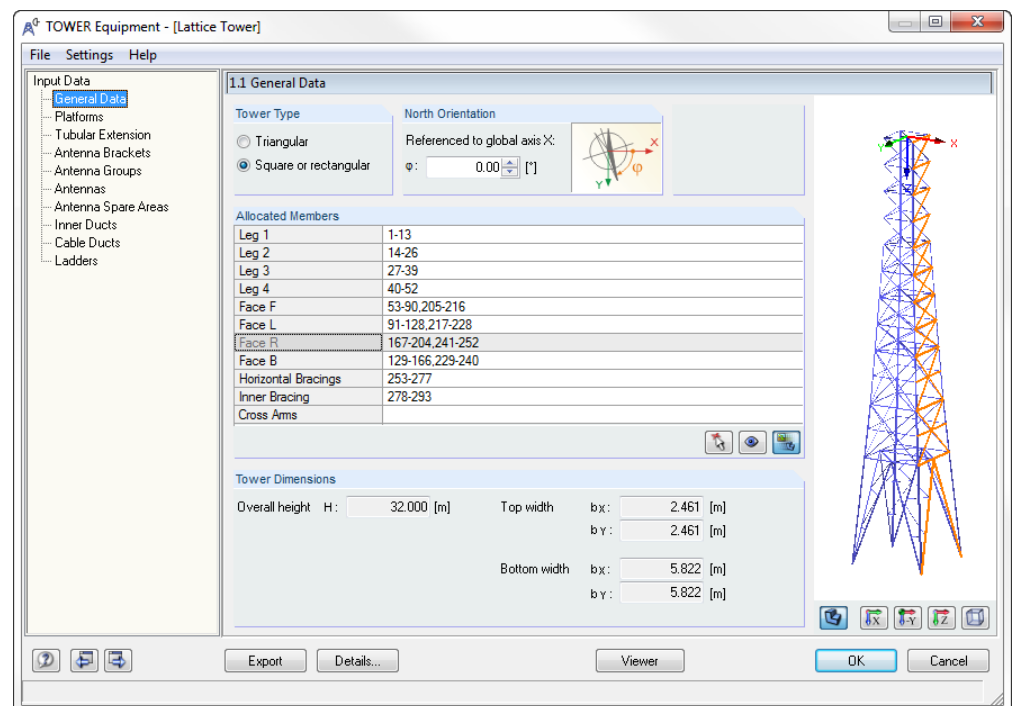
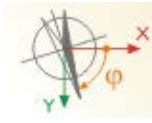


Figure 6.1: Table 1.1 General Data

When the tower structure was created without using the add-on module **TOWER Structure**, or when it was modified after the export from **TOWER Structure**, the table *Allocated Members* is empty. In this case, you can enter the member numbers manually into the table rows. You can also use the [Pick] function to select the relevant structural components in the RSTAB/RFEM structure.

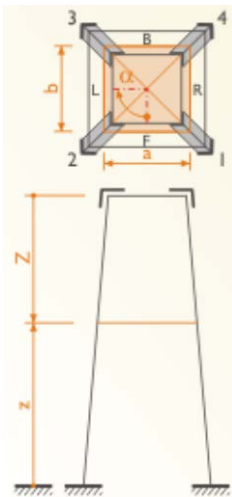
North Orientation

The orientation of the aerials is important for the load definition and the tower design. You define the orientation of the tower in reference to the north by means of the angle ϕ that refers to the global axis X in RSTAB.



6.2 Platforms

Platforms are important components of towers. With table 1.2 the program provides settings to integrate various types of platform constructions into the tower structure.



Platform No.	Platform Type	Rotation	Level z [m]	Coordinate Z [m]	To leg from outside	Comment
1	PRA004	0.00	31.000	1.000	<input checked="" type="checkbox"/>	
2	PRI002a	0.00	29.000	3.000	<input checked="" type="checkbox"/>	
3	PRI003-x	0.00	27.000	5.000	<input checked="" type="checkbox"/>	
4	PRI002a-x	0.00	25.000	7.000	<input checked="" type="checkbox"/>	
5						

Parameters - Platform No. 1	
Length	a 2.440 m
Length	b 2.440 m
Distance	a1 0.500 m
Length	a2 0.650 m
Distance	a3 0.500 m
Distance	b1 0.450 m
Distance	F3 0.500 m
Distance	L1 0.500 m
Distance	L3 0.500 m
Distance	B1 0.500 m
Distance	B3 0.500 m
Distance	R1 0.500 m
Distance	R3 0.500 m
Material	
Description	Steel S 355
Cross-section	
Cross-section 1	UPE 180

Figure 6.2: Table 1.2 Platforms

In the upper part of the table you define the *Platform Type*. You can take advantage of templates that you select from a database. To open the database, place the pointer in a relevant row of table column A and click the button [...] or use the [Library] button. The altitude of the platform is defined by entering values for the height *Level z* or the *Coordinate Z*. The definition of these variables can be checked in the graphical representation of the tower.

As it is not always required to arrange platform members symmetrically, you have the possibility to define a platform *Rotation*, for example to facilitate the straight duct of a cable line. In table column B, you can define an angle of 0°, 90°, 180° and 270°.

The parameters referring to the platform selected in the upper part are displayed in the lower part of the table. The position of the individual platform beams is defined by means of the respective distances. Use the buttons described in Table 6.1 to adjust the display individually for a better overview. If parameters are highlighted in red, it is not possible to integrate the platform into the tower geometry, using the dimensions that you have defined or taken from the database. In this case, point to the parameter description and TOWER Equipment shows you the possible range of values of the parameter.

B1	1.600 m
B3	1.400 m
R1	Distance
R3	Range of values: (0; 1.273) m









Button	Function
	Displays the scheme or model.
	Displays the wire-frame or the rendered model.
	Displays the current object or all objects.
	Displays non-selected objects as transparent.
	Displays dimensions.
	Displays dimensions with symbols.
	Model view in direction of axes +X, -Y and +Z
	Displays isometric view.

Table 6.1: Buttons for visualization of platforms

The platform construction may be made of another steel grade than the basic construction of the tower. Therefore, you can set the material as a parameter in the lower part of table 1.2. The cross-sections of the platform members are already defined for each template of the database. Click the [...] button to select another cross-section size. You can also use the cross-section library to define a new cross-section. The use of the library is described in detail in chapter 2.2 on page 10.

6.3 Creating User-defined Platforms

If the database templates are not adequate for the platform construction that you require, you can create templates yourself. Open an empty RSTAB or RFEM structure file where you create your platform structure. Then, save the platform as a block. As many dimensions of the platform are parameterized, it is recommended to use a standard platform as template for generating the customized component. It is necessary to observe some rules when you create and specify the template so that TOWER Equipment can recognize it correctly.

Scheme for exterior platforms

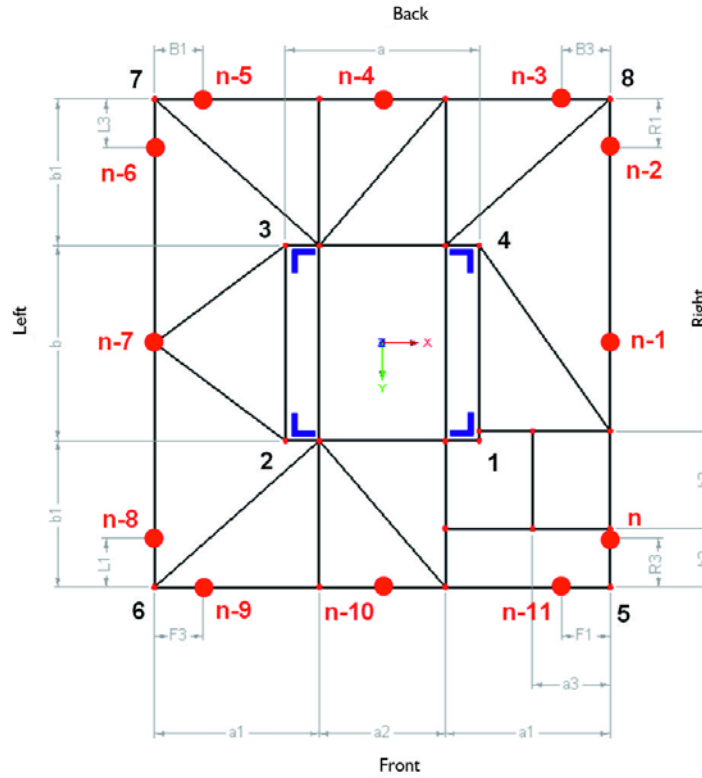


Figure 6.3: Scheme for exterior platforms

Scheme for interior platforms

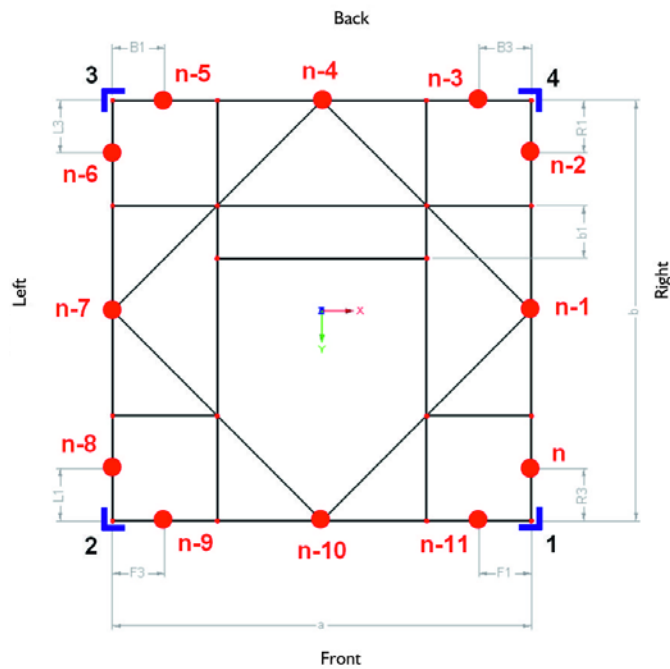


Figure 6.4: Scheme for interior platforms

If you consider the following specifications, you can use all kinds of user-defined platforms in TOWER Equipment.

- Planar platforms are created as 2D structures.
- Platforms with L-sections must be created as 3D structures (XY).
- The nodes of platform corners must be numbered with 1, 2, 3, 4. Node no. 1 lies within the first quadrant (positive X- and Y-coordinates), all other corner nodes are orientated clockwise.
- Parameter "a" refers to the distance between the nodes 1 and 2 (3 and 4). Parameter "b" refers to the distance between nodes 1 and 4 (2 and 3).
- The twelve nodes of the highest numbering (in case of three nodes for each platform corner) are available for attaching antenna brackets.
- Nodes in the outside corners of exterior platforms must be numbered with 5, 6, 7, 8. Node no. 5 lies within the first quadrant (positive X- and Y-coordinates), all other corner nodes are orientated clockwise.
- Cross-section no. 1 is always assigned to exterior members. If you want another member of the platform to have the same cross-section, you have to define a new cross-section.
- The new defined platforms must be saved as blocks and labeled according to Table 6.2.

Block Description	Tower Construction	Platform	Platform Shape
PRR	quadrilateral	exterior	rectangular
PRO			octagonal
PRT			triangular
PRC			circular
PRI		interior	interior
PRA			interior with support for tubular extension
PTR	triangular	exterior	rectangular
PTP			polygonal
PTT			triangular
PTC			circular
PTI		interior	interior

Table 6.2: Descriptions for user-defined platforms

6.4 Tubular Extension

Tubular extensions represent devices supporting antennas and lightning rods. In the upper part of table 1.3, you can determine geometry and position of the extension pipe. The tower picture displayed to the right indicates the different parameters.

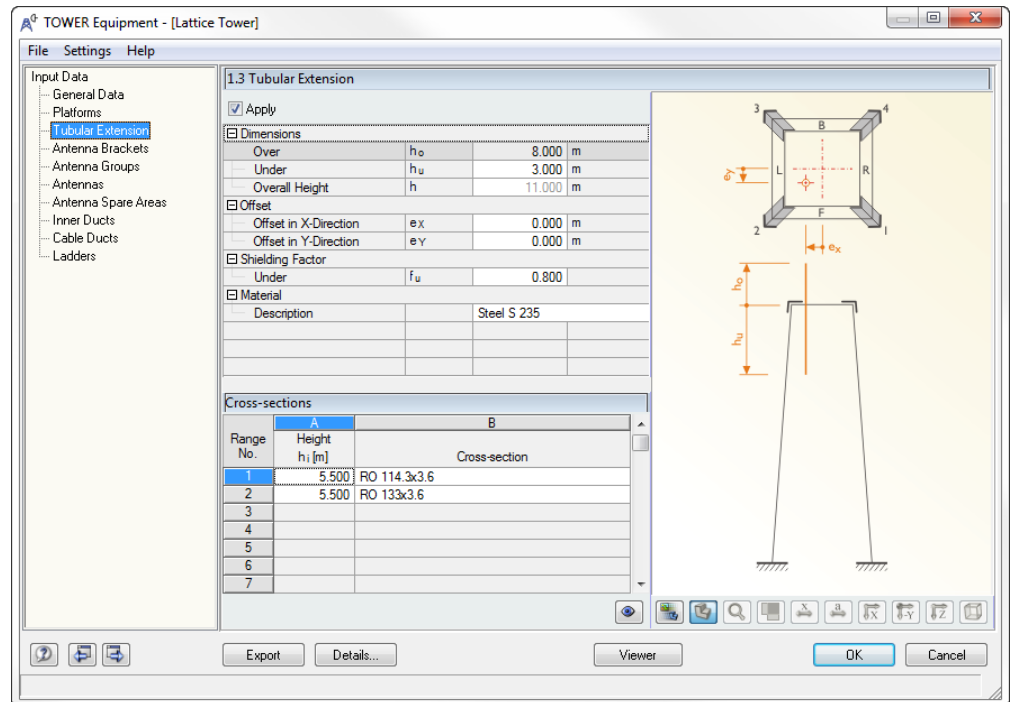


Figure 6.5: Table 1.3 Tubular Extension

The *Shielding Factor* is set to 1.0 by default. According to DIN 4131, annex A, chapter A1.3.2.2, it is possible to reduce the wind load by up to 20% for design elements included in the tower's outline area.

In the bottom part of the table, you define the *Cross-sections* of the tubular extension. Moreover, you can subdivide the structural component into several ranges and thus different cross-sections by defining partial heights h_i .

6.5 Antenna Brackets

The antenna brackets are defined in table 1.4. As these structural components may have different shapes depending on the antenna that you want to attach, **TOWER Equipment** provides an extensive database of templates.

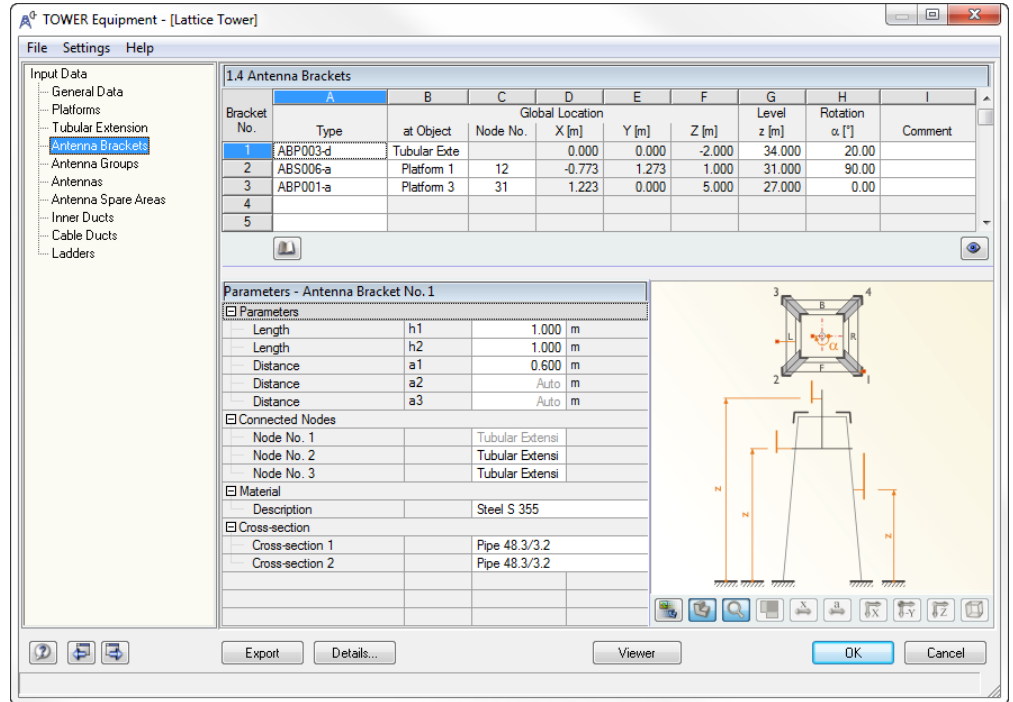


Figure 6.6: Table 1.4 Antenna Brackets

No.	Location	Order
22	Face F	2
23	Face F	3
24	Face L	1
25	Face L	2
26	Face L	3
27	Face B	1
28	Face B	2
29	Face B	3
30	Face R	1
31	Face R	2

In table column B you define the structural tower component to which you want to attach the antenna bracket. You can choose between tubular extensions, legs or platforms to take over the role of a bearing support structure. When you select a platform to be the supporting structure, you have to specify the corresponding node in table column C. Use the pull-down menu to select the nodes for assignment to the relevant tower sides.

6.6 Antenna Groups

A grouping of antenna assignments is useful when you want to evaluate the design loading with regard to particular mobile network operators. In table 1.5, you define the corresponding antenna groups.

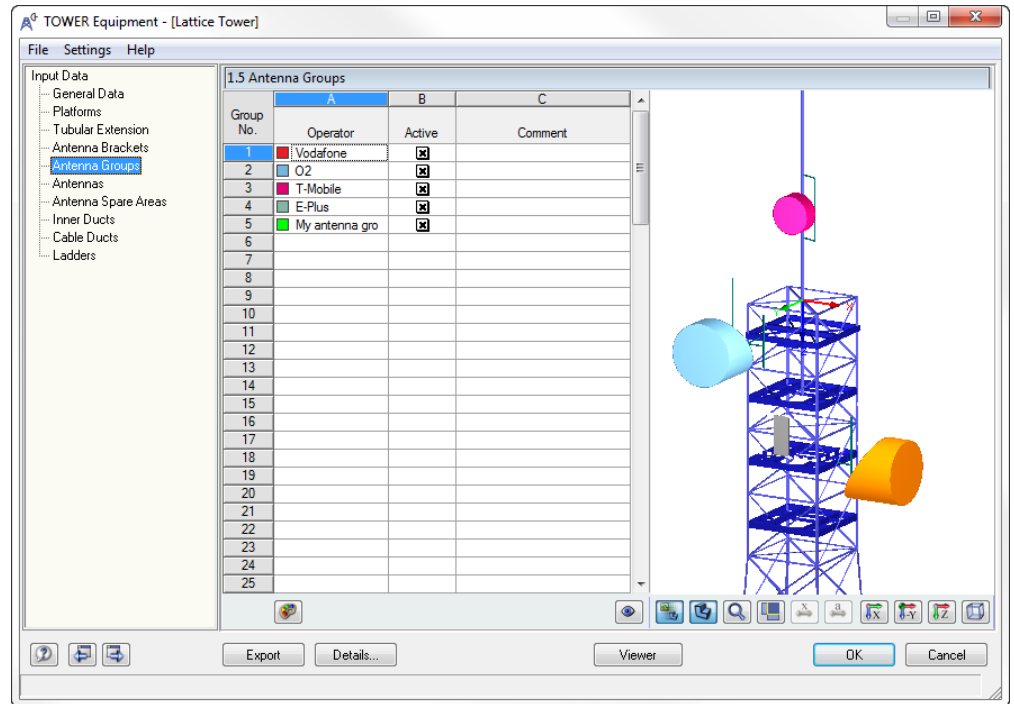


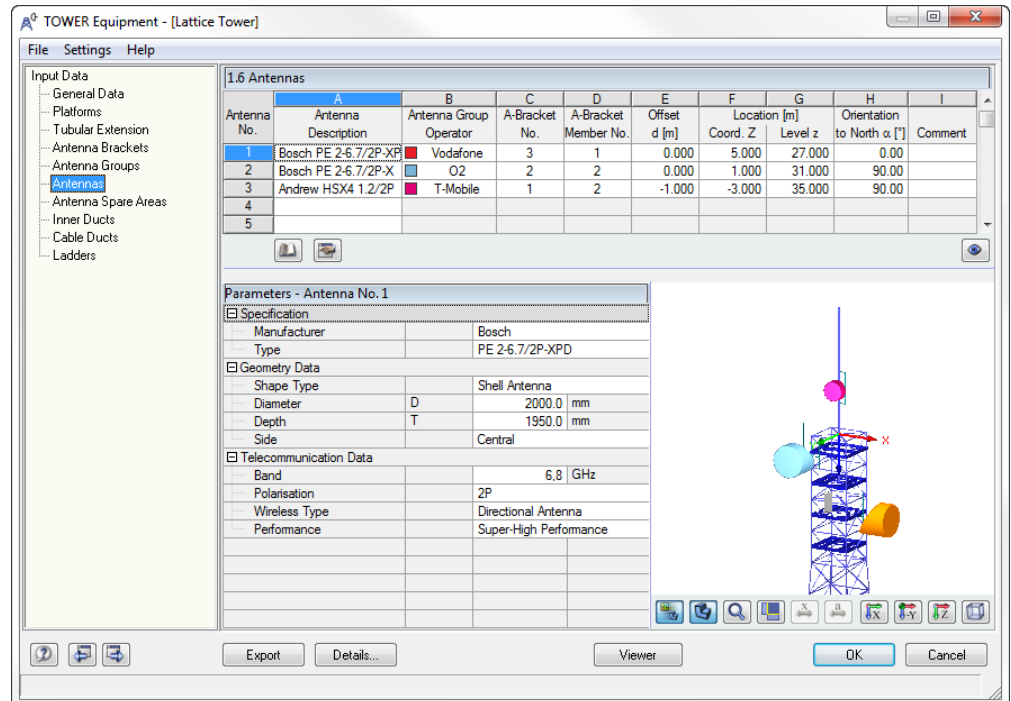
Figure 6.7: Table 1.5 Antenna Groups

Germany's major mobile network operators are already specified in the figure and highlighted with colors.

You can also define your own operators. The colors can be set individually by using the color table.

6.7 Antennas

Because of their shape and size antennas may represent big contact surfaces for occurrent wind loads. Therefore, the definition of their position and orientation is significant for the design.



The screenshot shows the '1.6 Antennas' table with the following data:

Antenna No.	Antenna Description	Antenna Group Operator	A-Bracket No.	A-Bracket Member No.	Offset d [m]	Location [m] Coord. Z	Level z	Orientation to North α [°]	Comment
1	Bosch PE 2-6.7/2P-XP	Vodafone	3	1	0.000	5.000	27.000	0.00	
2	Bosch PE 2-6.7/2P-X	O2	2	2	0.000	1.000	31.000	90.00	
3	Andrew HSX4 1.2/2P	T-Mobile	1	2	-1.000	-3.000	35.000	90.00	
4									
5									

The 'Parameters - Antenna No. 1' dialog box shows the following details:

- Specification:** Manufacturer: Bosch, Type: PE 2-6.7/2P-XPD
- Geometry Data:** Shape Type: Shell Antenna, Diameter: D 2000.0 mm, Depth: T 1950.0 mm, Side: Central
- Telecommunication Data:** Band: 6.8 GHz, Polarisation: 2P, Wireless Type: Directional Antenna, Performance: Super-High Performance

Figure 6.8: Table 1.6 Antennas

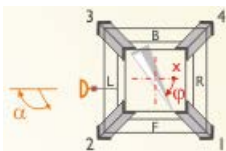
Column A of table 1.6 defines the antenna type. You can take advantage of templates that you select from a comprehensive database. To open the database, place the pointer in a relevant row of table column A and click the button [...] or use the [Library] button.

When you have defined the *Antenna Group* in column B, you specify the exact antenna position on the tower. It can be useful to maximize the antenna display by zooming into the table's graphic window.

The *Location* of the antenna is defined by means of the antenna brackets defined in table 1.4. Depending on the bracket type, a different number of members, to which you can attach the antenna, is available for selection in table column D. To modify the position of the receiver unit within the member, use the *Offset* value.

Another important definition is the specification of the antenna orientation which is significant for the loading to be defined later. The *Orientation* of the antenna is defined by the angle α in relation to the northern direction.

The bottom part of table 1.6 shows detailed information about parameters of the antenna and its geometrical data.



6.8 Antenna Spare Areas

In case the type of antenna construction is not yet clearly defined when the design is performed, or if it cannot be selected in the database, you can define an equivalent surface in table 1.7 to consider this design element. The surface will be used for the generation of the real wind or ice loading determined in the add-on module **TOWER Loading**.

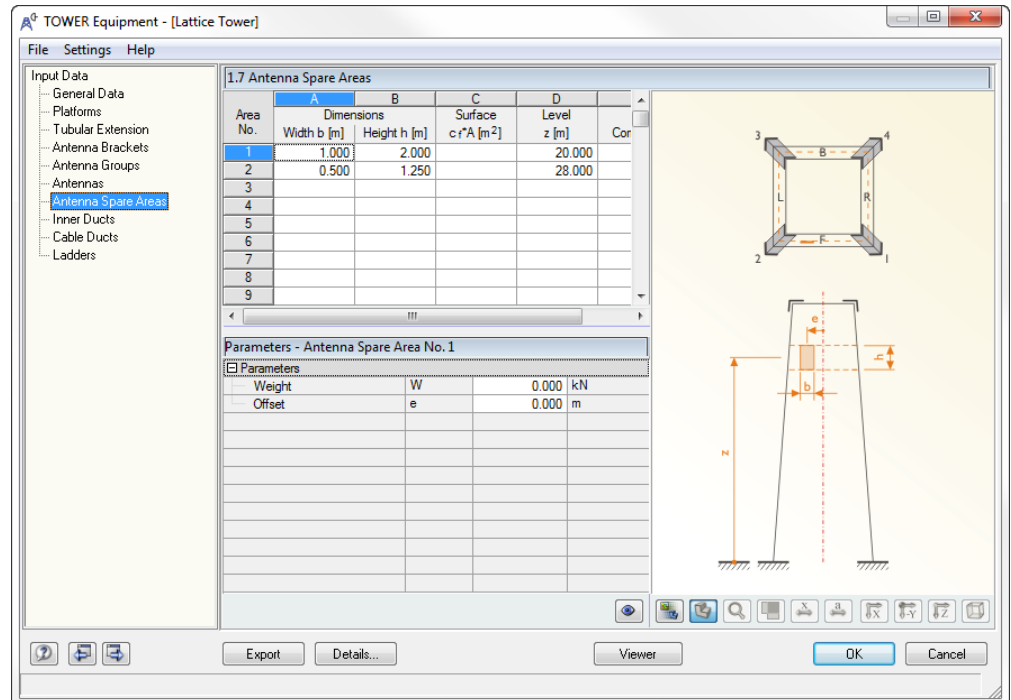


Figure 6.9: Table 1.7 *Antenna Spare Areas*

The table defines dimensions and level of the equivalent area. It is not necessary to determine a particular tower side. The decisive tower face will be determined in the add-on module **TOWER Loading** when the loading is generated.

To take into account the antenna's self-weight, it is possible to define the *Weight* W as self-weight force among the parameters. Furthermore, it is possible to specify an *Offset* e . The value will be considered for the wind load generation in the add-on module **TOWER Loading**.

6.9 Inner Ducts

Inner ducts are required to insert ladders and cable ducts. In table 1.8 you can define the size, position and number of inner ducts.

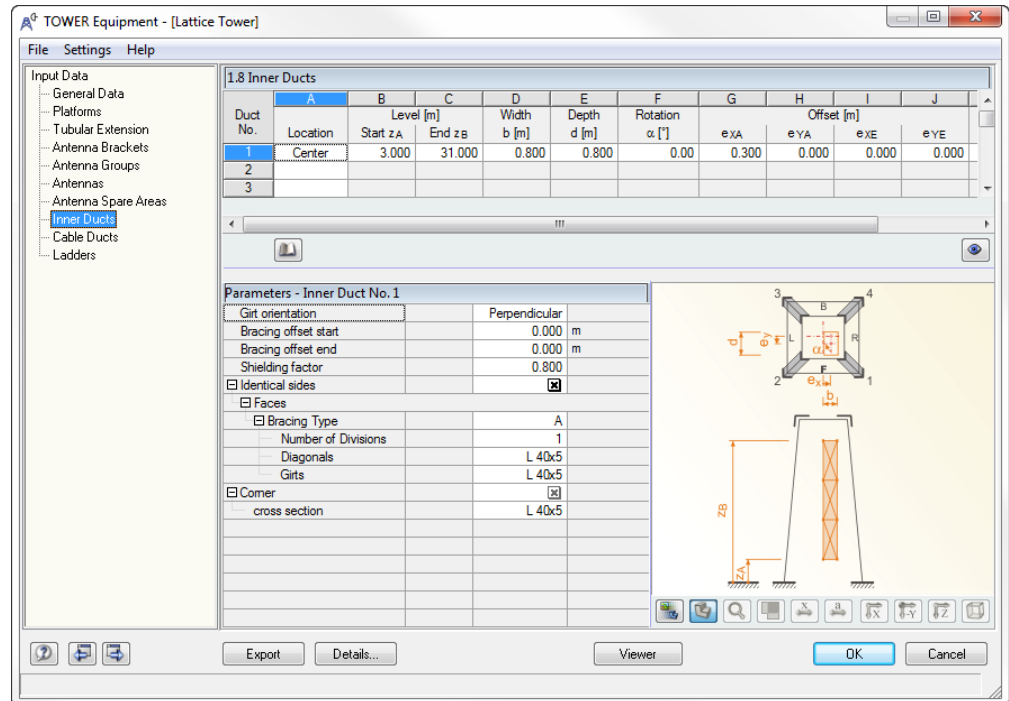


Figure 6.10: Table 1.8 Inner Ducts

The inner ducts can be applied inside the tower structure or to its sides.

With the height $Level\ z_A$ and z_B you determine the start and end of the respective duct. The $Rotation\ \alpha$ in table column F and the $Shielding\ factor\ K_{sh}$ in the dialog section *Parameters* are settings required for the subsequent determination of the appropriate wind load.

With the help of values entered in table columns G to J you can define an offset for the inner duct's start and end position. Furthermore, you can create a height offset of the inner duct's bracing by means of the settings for the *Bracing offset* in the *Parameters* dialog section.

The sides of the inner ducts are designed with identical layouts by default. If an individual design is needed for each side, clear the check box for *Identical sides* in the *Parameters* dialog section.

Again, the already familiar library is available for the cross-section selection. To select a different cross-section, place the pointer into the cell of the cross-section that you want to change, and then click the [...] button.

6.10 Cable Ducts

Similar to the input of inner ducts you enter the specifications for cable ducts which must be considered for the wind load of self-weight and ice weight.

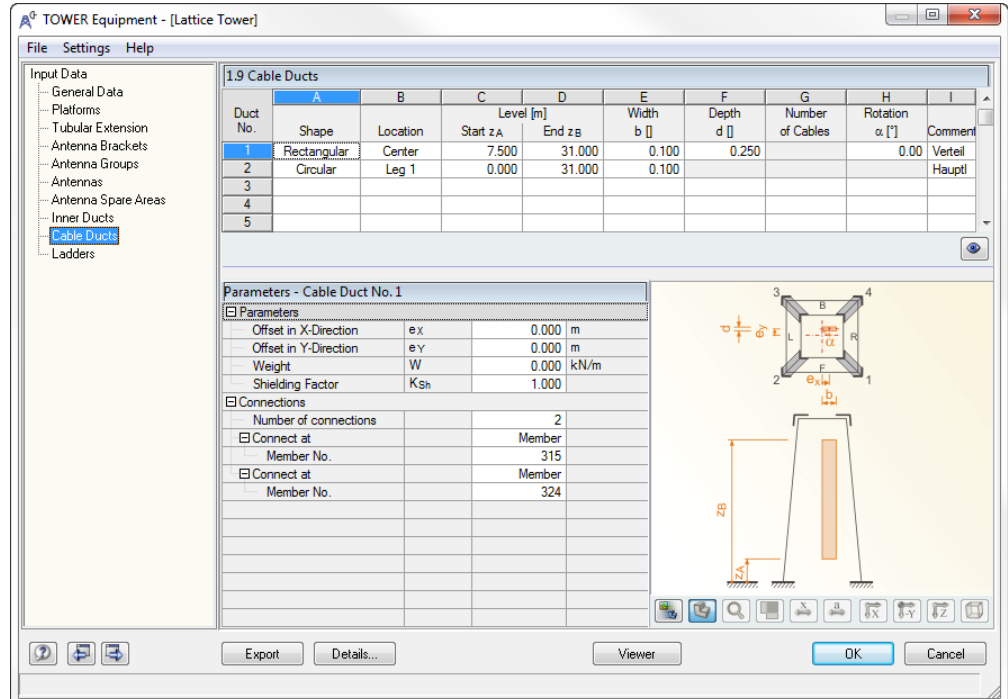


Figure 6.11: Table 1.9 Cable Ducts

You can define rectangular and circular cable ducts. Data is entered in the same way like data for inner ducts. To take into account the leeward zone appropriately when generating wind loads, it is possible to place several cables in a row. If the *Row* shape is selected in table column A, column G will be enabled and you can specify the number of cables.

The start and end of the respective cable duct are defined by means of the height *Level* z_A and z_B . The *Rotation* α , the *Shielding Factor* K_{Sh} and the *Offset* are defined by the parameters of each single cable duct. The self-weight is indicated as linear load in [kN/m].

With *Connections* you specify supports for the cable ducts on the tower construction. The setting is significant for applying loads from self-weight, wind and ice to the load-bearing structure.

6.11 Ladders

Table 1.10 contains the ladders required to climb the tower.

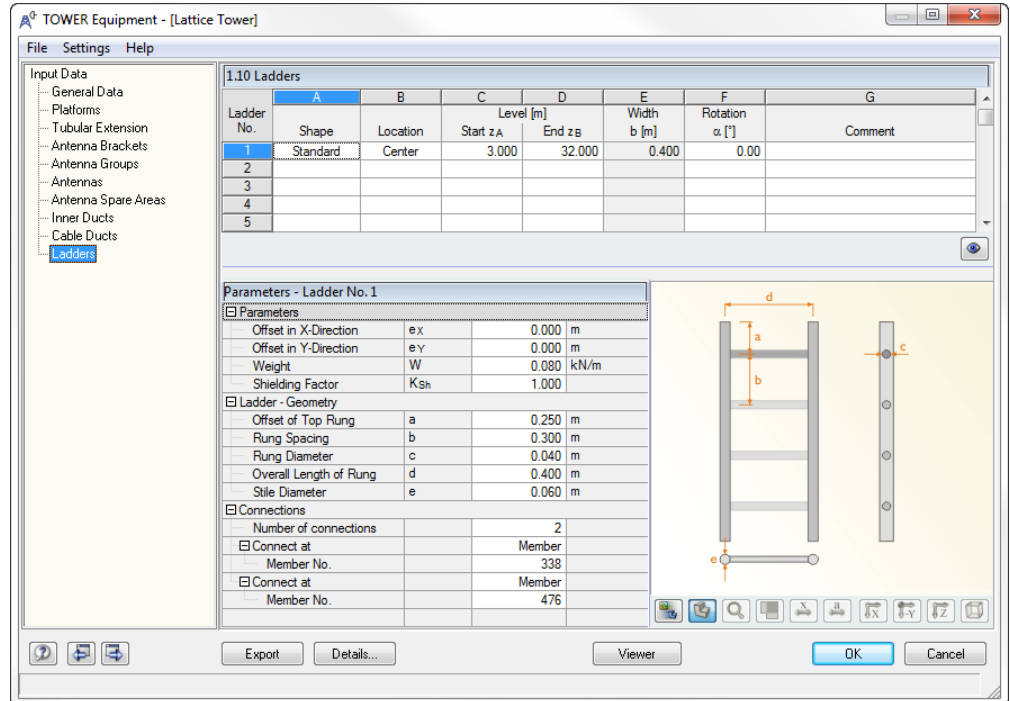


Figure 6.12: Table 1.10 Ladders

Five ladder types are available for selection. You see the standard ladder in Figure 6.12. More types are shown in the following.

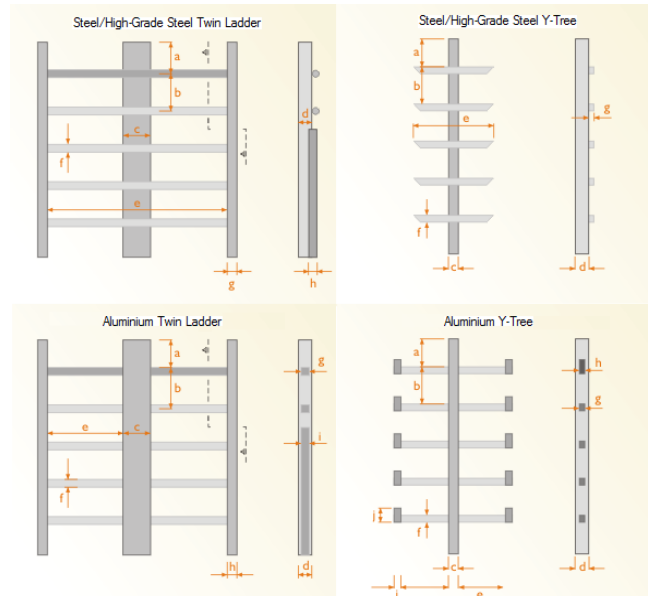


Figure 6.13: Ladder types

The start and end of the respective ladder are determined by means of the height $Level z_A$ and z_B . The $Rotation \alpha$, the $Shielding Factor K_{Sh}$ and the $Offset$ are defined by the parameters of each single ladder. The self-weight is already specified for the standard parameters. If you change the ladder geometry, you have to change the weight accordingly.

Finally, you have to decide how often and to which locations of the tower construction you want to attach the equipment. The ladder definition is important for the right determination of the tower loading.

6.12 Export

When you have defined all equipment, you can export the equipment data to RSTAB/RFEM.

Statically effective components like antenna brackets, tubular extensions and inner ducts will be created as members during the export. To visualize other equipment, a TOWER Equipment case is generated in the main program. The case data can be displayed like a normal load case.

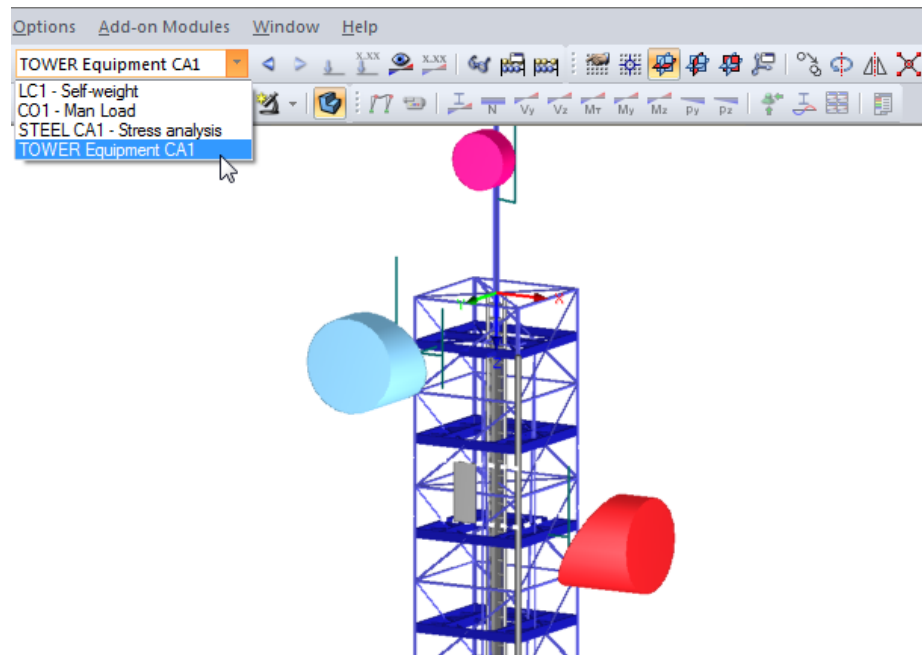


Figure 6.14: TOWER Equipment case in RSTAB/RFEM

7. General Functions

You can use the menu bar at the top of the TOWER module window to access various general functions.

7.1 Delete a TOWER Case

To delete a TOWER case,

select **Delete Case** on the **File** menu in the TOWER add-on module.

When you confirm the selection, the program deletes the case and you quit the add-on module. Data that has already been exported is preserved in RSTAB/RFEM.

7.2 Units and Decimal Places

The units and decimal places for RSTAB/RFEM and all add-on modules are managed in one global dialog box. To open the dialog box out of the TOWER module,

select **Units and Decimal Places** on the **Settings** menu.

The following dialog box opens, which you already know from RSTAB/RFEM. The TOWER add-on module is preset.

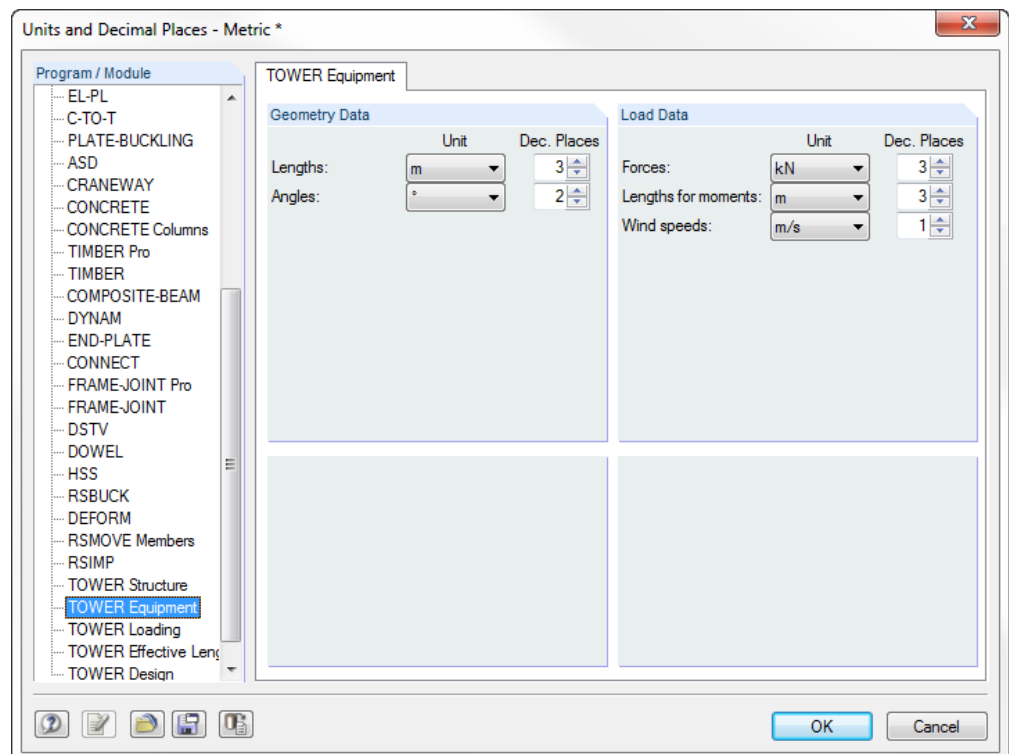


Figure 7.1: Dialog box *Units and Decimal Places*



The settings can be saved as user profile to reuse them in other structures.

8. TOWER Effective Lengths

8.1 Introduction

This add-on module creates buckling lengths for the individual members. The program is able to generate effective lengths for lattice towers with square, rectangular or triangular floor plans. Furthermore, it is possible to consider tower data from **TOWER Structure** and **TOWER Equipment** for the input. However, the add-on module **TOWER Effective Lengths** works independently which means that you can use any truss tower from RSTAB/RFEM for the determination of the effective lengths. After the length generation you can export the generated buckling lengths to take them into account for the design in the add-on module **TOWER Design**.

The generation of effective lengths is based only on the tower geometry. Thus, it is independent of the loading. You can influence the calculation by using the options in the *Details* dialog box. The buckling lengths can be edited manually after the calculation. TOWER Effective Lengths represents an efficient and powerful tool for the determination of buckling lengths and is described in the following.

8.2 Start the Add-on Module

RSTAB and RFEM provide the following options to start the add-on module **TOWER Effective Lengths**.

Menu

To start the program in the menu bar,

point to **Towers** on the **Add-on Modules** menu, and then select **TOWER Effective Lengths**.

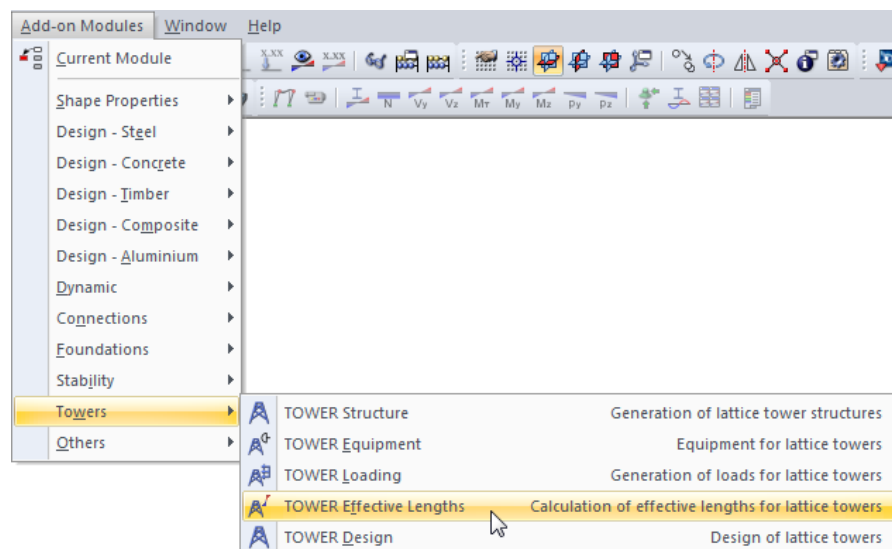


Figure 8.1: Main menu: *Add-on Modules* → *Towers* → *TOWER Effective Lengths*

Navigator

To start TOWER Effective Lengths in the *Data* navigator,
select **TOWER Effective Lengths** in the **Additional Modules** folder.

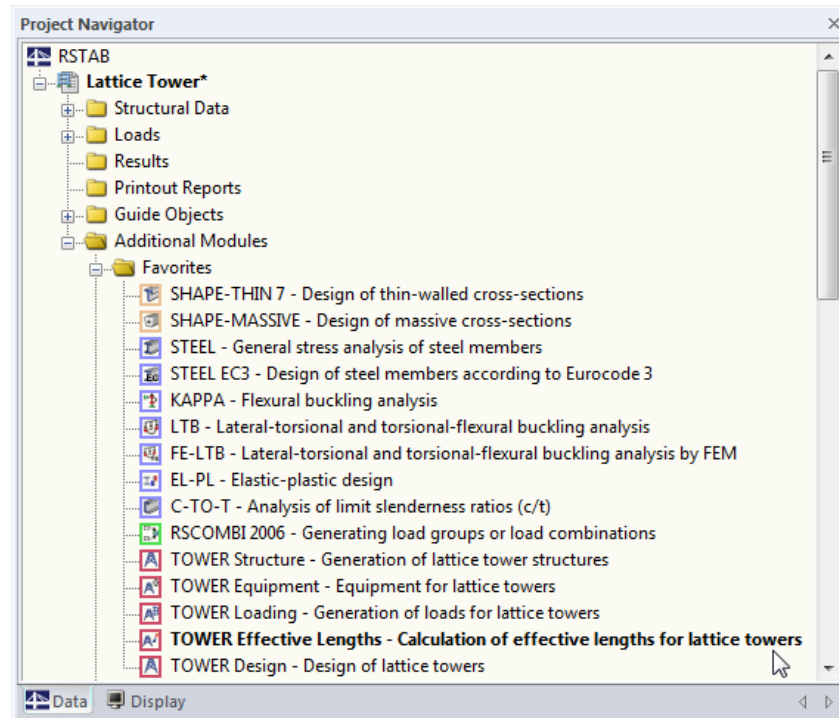
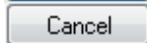
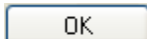


Figure 8.2: Data navigator: *Additional Modules* → *TOWER Effective Lengths*

9. Input Data

The data required for the determination of effective lengths is entered in table 1.1 *General Data*.

When you have started **TOWER Effective Lengths**, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently.



To select a table, click the corresponding entry in the TOWER navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.

To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the module but without saving the data.

9.1 General Data

Table 1.1 shows you a general overview about the structure. In the dialog sections *Tower Type* and *Number of*, you define the basic parameters of the tower. Depending on your specifications, you see table rows appearing in the dialog section *Allocated Members*.



Use the [Pick] button to select members graphically from RSTAB/RFEM for the analysis. If a tower was created with the help of the add-on modules TOWER Structure and/or TOWER Equipment, the program assigns the members automatically. When you click into a row of the table *Allocated Members*, you can see all corresponding members highlighted in the structure graphic to right. To activate the graphic, click the button [Show Figure or Rendering].

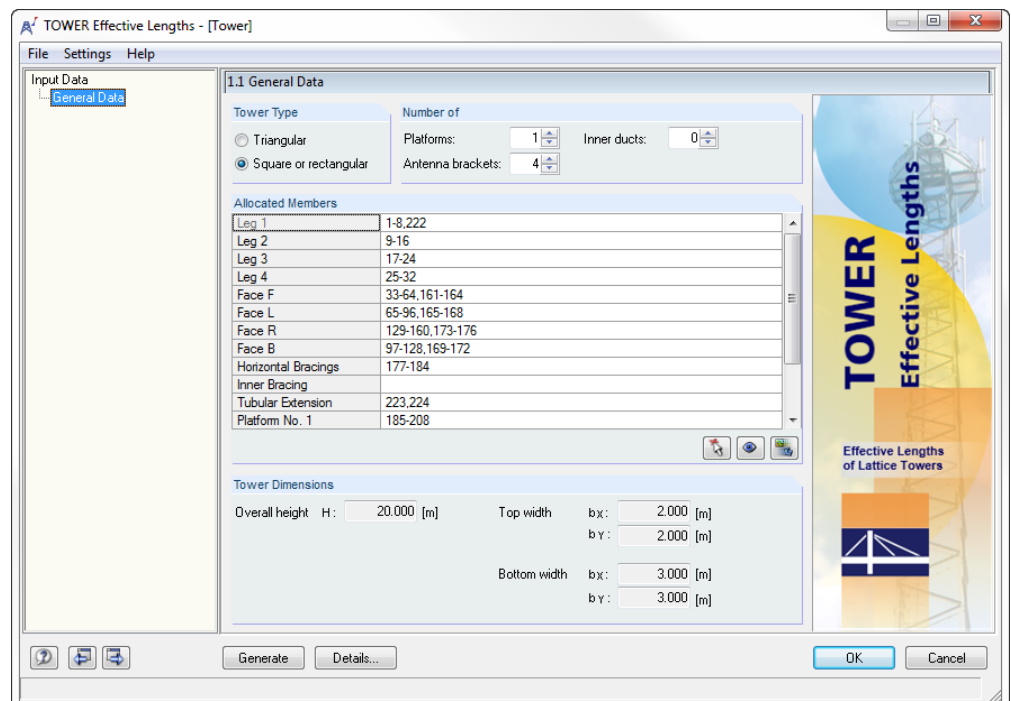


Figure 9.1: Table 1.1 *General Data*

9.2 Details

Details...

Click the [Details] button to open a dialog box with numerous settings which important for the determination of effective lengths.

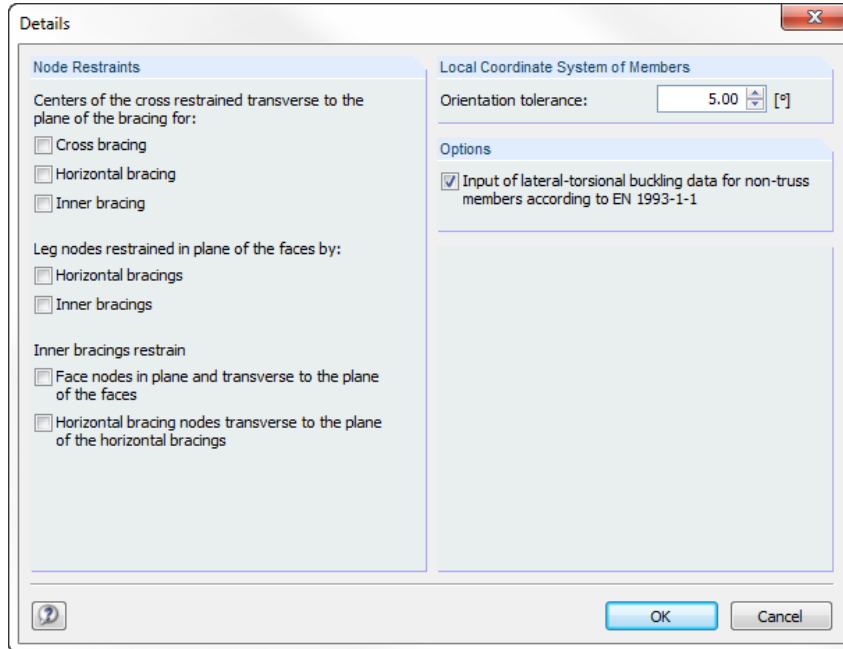


Figure 9.2: Dialog box *Details*

Node Restraints

For the generation of effective lengths it is necessary to define the nodes which you want to treat as restrained by other members and the direction for which they are fixed, and which nodes are not considered to be restrained. The dialog section *Node Restraints* provides the corresponding options.

Figure 9.3 shows the options used to define restraints perpendicularly to the bracing plane of the node where two bracing members are intersecting. When a check box is ticked, the program considers the intersection node of the respective bracing as restrained and generates the effective length L_1 perpendicular to the bracing plane (see Figure 9.4). If the check box is not ticked, the respective node is not treated as restrained. In this case, the effective length L_2 will be generated.

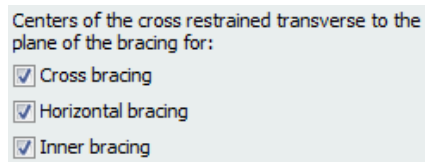


Figure 9.3: Check boxes for nodal restraint perpendicular to bracing plane

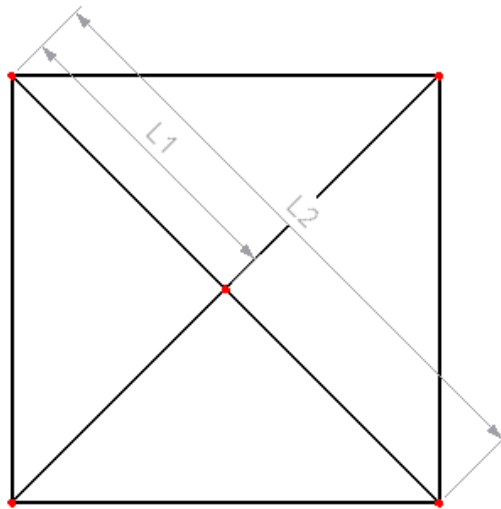


Figure 9.4: Generated effective lengths depending on node restraint

The options shown in Figure 9.5 are used to define the way how tower legs of horizontal bracings are restrained. If the check box *Horizontal bracings* is inactive, the program restrains the legs only in diagonal directions due to the bracings (see Figure 9.6a). When the check box is ticked, the tower legs will be fixed both diagonally and in directions of the tower sides (see Figure 9.6b). In the same way, it is possible to fix legs by *Inner bracings*.

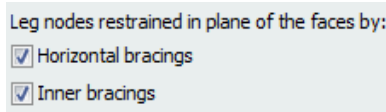


Figure 9.5: Restraint of tower leg nodes

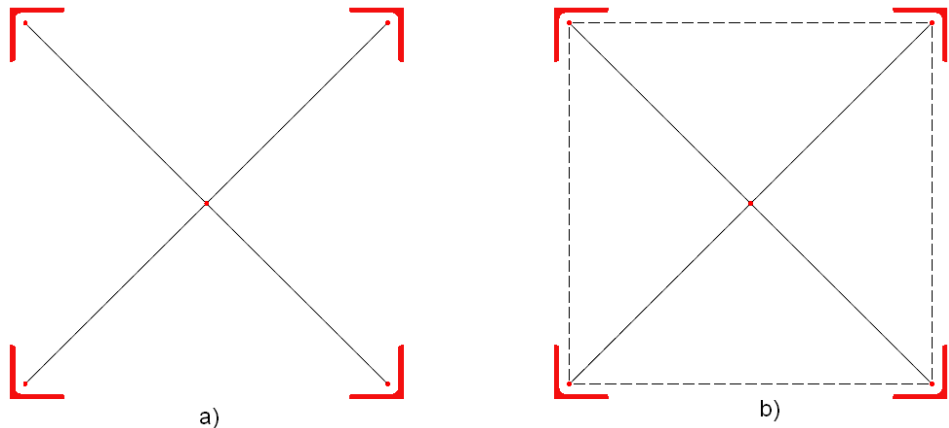


Figure 9.6: Restraint of leg nodes by horizontal bracing

In case inner bracings exist in the tower, they can be taken into account for the determination of buckling lengths for the horizontal bracings as well as the side members. The setting options are shown in Figure 9.7.

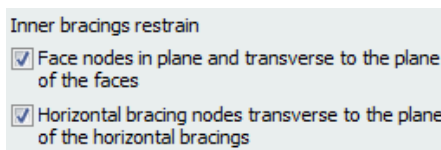


Figure 9.7: Inner bracing restrain

Local Coordinate System of Members

It is important for the generation of buckling lengths that the cross-sections of tower elements are aligned correctly. The following rules are valid for the orientation of the member axes:

- **Tower legs:**
The local axes y and z must be parallel to the sides of the tower (see Figure 9.8a). The alignment of tower legs is irrelevant for triangular towers because their effective lengths will be determined by a simplified method.
- **Side members:**
The local axis y or z must be parallel to the plane of the side (see Figure 9.8b).
- **Horizontal bracings:**
The local axis y or z must be perpendicular to the bracing plane.

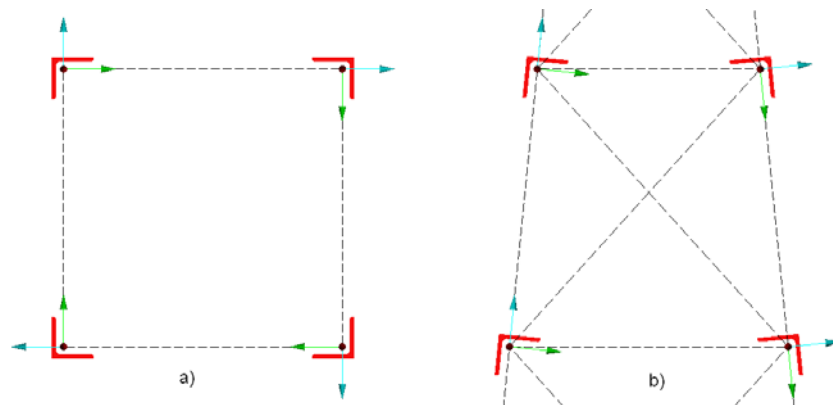


Figure 9.8: Orientation of leg and side members

When the tower was created with **TOWER Structure**, the members are aligned according to the rules. If the tower has been modeled manually in RSTAB/RFEM, you have to check the alignment of the individual members to ensure a correct generation of buckling lengths. If necessary, the orientation of single members must be adjusted in RSTAB/RFEM.

However, the accurate determination of member orientations can be troublesome, especially for inclined members. Therefore, the input field *Orientation tolerance* offers you the possibility to determine the member orientation approximately. You can define the tolerance from 0° to 44.99° . When the orientation is not in line with the rules above including tolerance, the effective length cannot be determined automatically.

Options

The effective lengths of non-truss members can be defined manually in table 2.2. Later, when you want to design such members according to EN 1993-1-1, you can enter the data for lateral-torsional buckling. To enable the corresponding input fields in table 2.2, tick the check box.

10. Generated Data

Generate

Use the [Generate] button to start the determination of the effective lengths. The program also checks the entered data. As members can be assigned only once in table 1.1 *Allocated Members* (see Figure 9.1), an error message appears if a member is assigned several times.

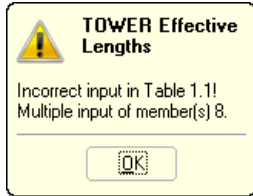


Figure 10.1: Warning in case of multiple assignment of members

The calculation cannot be performed as long as input data of table 1.1 *Allocated Members* is not correct.

When the buckling lengths have been generated successfully, the module's results tables are displayed. They are described in detail in the following chapters.

10.1 Effective Lengths - Truss Members

Table 2.1 *Effective Lengths - Truss Members* displays the generated buckling lengths. It shows only truss members, that means members forming the main framework structure of the tower (tower legs, sides, horizontal and inner bracings). For each member the table lists the *Effective Length Factors* k_x , k_y , k_z and the *Effective Lengths* $L_{cr,x}$, $L_{cr,y}$, $L_{cr,z}$. The values refer to the local coordinate system of the members.

Member No.	Length L [m]	Effective Length Factors and Effective Lengths					
		$k_{cr,x}$	$L_{cr,x}$ [m]	$k_{cr,y}$	$L_{cr,y}$ [m]	$k_{cr,z}$	$L_{cr,z}$ [m]
Leg No. 1 - Segment No. 1							
222	1.501	1.667	2.502	1.667	2.502	1.667	2.502
1	1.001	2.500	2.502	2.500	2.502	2.500	2.502
Leg No. 1 - Segment No. 2							
2	2.502	1.000	2.502	1.000	2.502	1.000	2.502
Leg No. 1 - Segment No. 3							
3	2.502	1.000	2.502	1.000	2.502	1.000	2.502
Leg No. 1 - Segment No. 4							
4	2.502	1.000	2.502	1.000	2.502	1.000	2.502
Leg No. 1 - Segment No. 5							
5	2.502	1.000	2.502	1.000	2.502	1.000	2.502
Leg No. 1 - Segment No. 6							
6	2.502	1.000	2.502	1.000	2.502	1.000	2.502
Leg No. 1 - Segment No. 7							
7	2.502	1.000	2.502	1.000	2.502	1.000	2.502
Leg No. 1 - Segment No. 8							
8	2.502	1.000	2.502	1.000	2.502	1.000	2.502

Figure 10.2: Table 2.1 *Effective Lengths - Truss Members*

The values k_y , k_z and $L_{cr,y}$, $L_{cr,z}$ can be represented in different colors:

- If the values are *black*, the program determined them correctly.
- If the values are *red*, they could not be determined automatically. In this case, the length factors are 1.0. Thus, the effective length corresponds to the member length.
- If a cell is blocked (*gray*), no value is existing. This case typically occurs for the values k_y and $L_{cr,y}$ of members with double symmetrical cross-sections.

Horizontal Bracing - Bracing No. 135					
277	3.192	1.000	3.192	1.000	3.192

Face B - Bracing No. 116					
154	2.884	1.000	2.884	1.000	2.884
149	2.530	1.000	2.530	1.000	2.530



The values for k_y , k_z and $L_{cr,y}$, $L_{cr,z}$ can be modified manually in table 2.1 as long as they are not grayed out which means locked.

To display the members' local coordinate systems in the graphic, use the button [Display LCS] shown on the left.

10.2 Effective Lengths - Non-Truss Members

Table 2.2 *Effective Lengths - Non-Truss Members* informs you about the buckling lengths of tower equipment parts such as platforms, inner ducts, ladders etc. The values in this table were not determined by the add-on module **TOWER Effective Lengths** because the program calculates only the buckling lengths of trusses. The buckling length coefficients of the non-truss members are set to 1.0. Thus, the effective lengths are equal to the member lengths.

The upper part of the table lists the effective length coefficients and the members' buckling lengths about their principal axes. The data for lateral-torsional buckling can be defined in table columns F to H, provided that the corresponding option was activated in the *Details* dialog box (see Figure 9.2). The table below shows *Details* of the member selected above.

Member No.	Length L	Buckling About y/u $k_{cr,y/u}$	Buckling About z/v $k_{cr,z/v}$	Lateral-Torsional Buckling $k_{z/v}$	k_w	L_w	L_T		
294	0.500	1.000	0.500	1.000	0.500	1.0	1.0	0.500	0.500
295	0.650	1.000	0.650	1.000	0.650	1.0	1.0	0.650	0.650
296	0.070	1.000	0.070	1.000	0.070	1.0	1.0	0.070	0.070
297	0.720	1.000	0.720	1.000	0.720	1.0	1.0	0.720	0.720
298	0.500	1.000	0.500	1.000	0.500	1.0	1.0	0.500	0.500
299	0.500	1.000	0.500	1.000	0.500	1.0	1.0	0.500	0.500
300	0.720	1.000	0.720	1.000	0.720	1.0	1.0	0.720	0.720
301	0.720	1.000	0.720	1.000	0.720	1.0	1.0	0.720	0.720

Cross-Section	UPE 180	
Length	L	0.650 m
<input type="checkbox"/> Buckling About Axis y		
Effective Length Coefficient	$k_{cr,y}$	1.000
Effective Length	$L_{cr,y}$	0.650 m
<input type="checkbox"/> Buckling About Axis z		
Effective Length Coefficient	$k_{cr,z}$	1.000
Effective Length	$L_{cr,z}$	0.650 m
<input type="checkbox"/> Lateral-Torsional Buckling		
Effective Length Coefficient	$k_{z/v}$	1.0
Effective Length Coefficient	k_w	1.0
LTB Length	L_w	0.650 m
Torsional Length	L_T	0.650 m
Comment		

Figure 10.3: Table 2.2 *Effective Lengths - Non-Truss Members*

10.3 Export of Results



The data from table 2.1 and 2.2 can be exported by clicking the [Export] button to use it later in the add-on module **TOWER Design** for further analysis.

11. General Functions

You can use the menu bar at the top of the TOWER module window to access various general functions.

11.1 Delete a TOWER Case

To delete a TOWER case,

select **Delete Case** on the **File** menu in the TOWER add-on module.

When you confirm the selection, the program deletes the case and you quit the add-on module. Data that has already been exported is preserved in RSTAB/RFEM.

11.2 Units and Decimal Places

The units and decimal places for RSTAB/RFEM and all add-on modules are managed in one global dialog box. To open the dialog box out of the TOWER module,

select **Units and Decimal Places** on the **Settings** menu.

The following dialog box opens, which you already know from RSTAB/RFEM. The TOWER add-on module is preset.

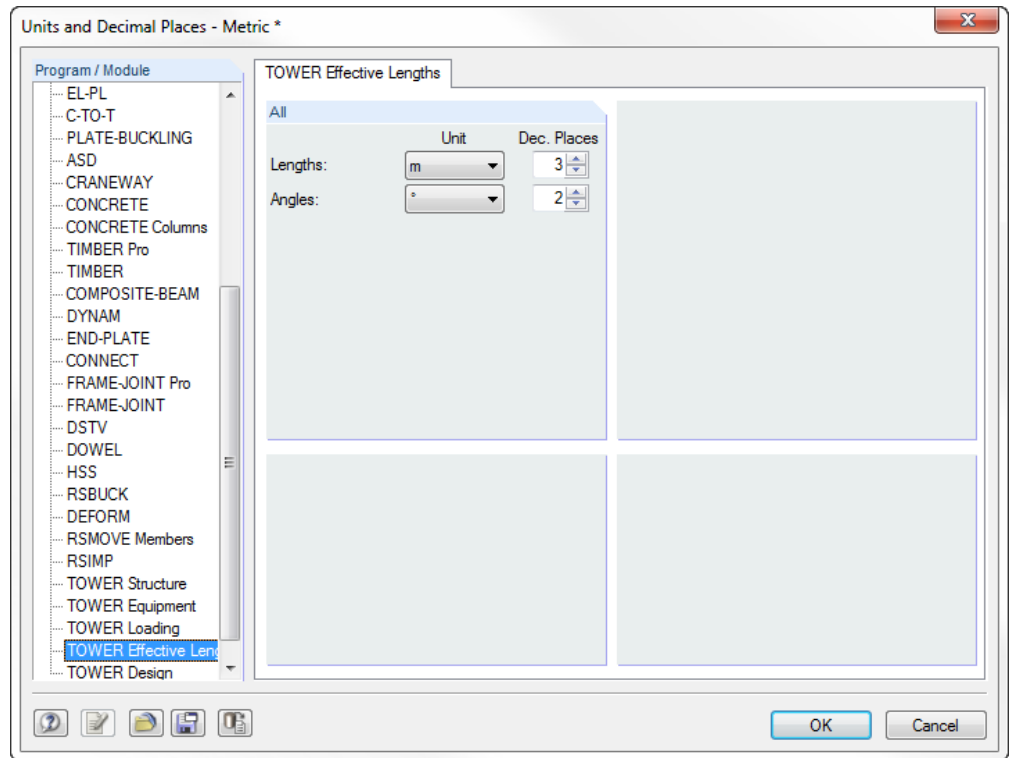


Figure 11.1: Dialog box *Units and Decimal Places*



The settings can be saved as user profile to reuse them in other structures.

12. TOWER Loading

12.1 Introduction

With the add-on module **TOWER Loading** you can generate easily actions that are required for the design. The program considers requirements according to DIN 1055 for self-weight, wind loads and ice loads as well as traffic loads according to DIN V 4131.

It is also possible to generate individual load situations. The following pages describe in detail how you can create a complex load definition with the add-on module **TOWER Loading** in almost no time.

12.2 Open the Add-on Module

RSTAB and RFEM provide the following options to start the add-on module **TOWER Loading**.

Menu

To start the program in the menu bar,

point to **Towers** on the **Add-on Modules** menu, and then select **TOWER Loading**.

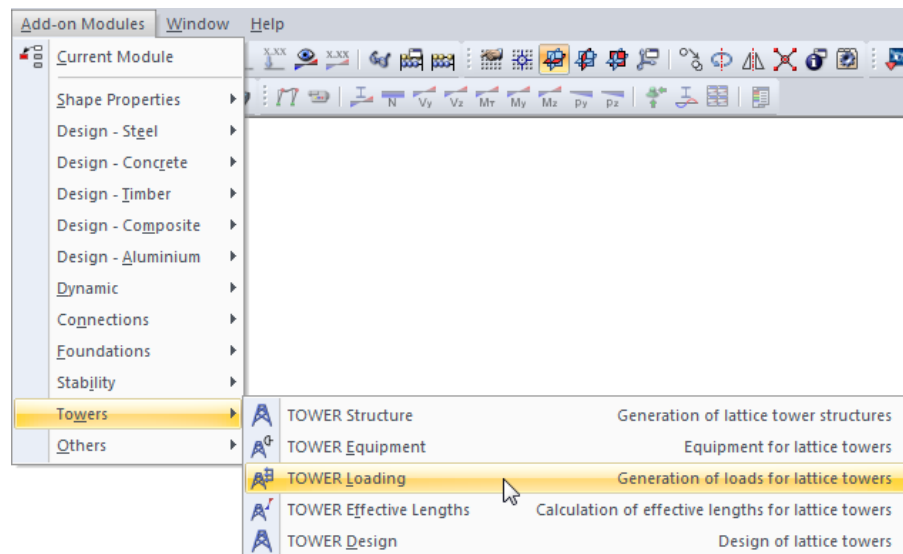


Figure 12.1: Menu: *Add-on Modules* → *Towers* → *TOWER Loading*

Navigator

To start TOWER Loading in the *Data* navigator,

select **TOWER Loading** in the **Additional Modules** folder.

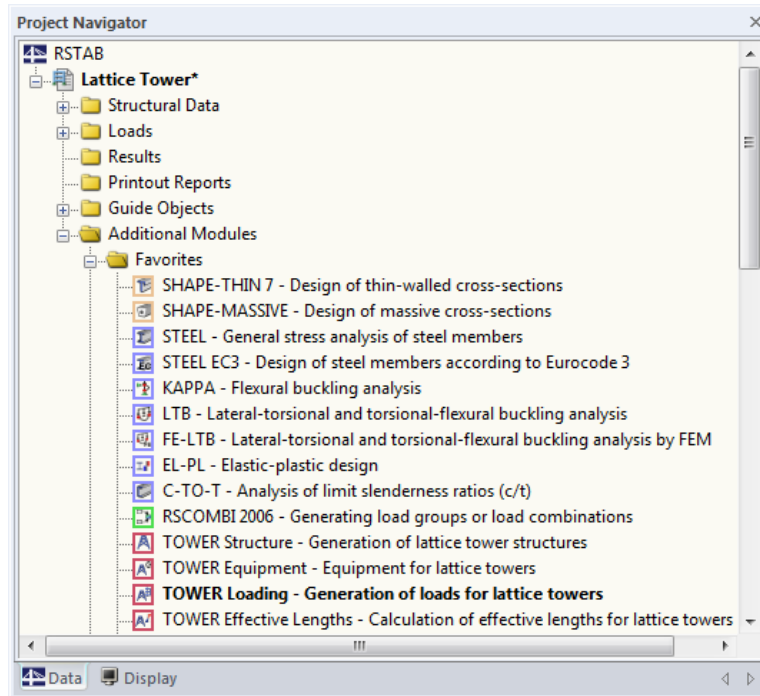


Figure 12.2: Data navigator: *Additional Modules* → *TOWER Loading*

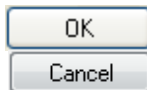
13. Input Data

The input for defining the tower loading is entered in tables.

When you have started the TOWER add-on module, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently.



To select a table, click the corresponding entry in the TOWER navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.



To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the module but without saving the data.

13.1 General Data

In table 1.1, you define essential information about the tower structure and available equipment.

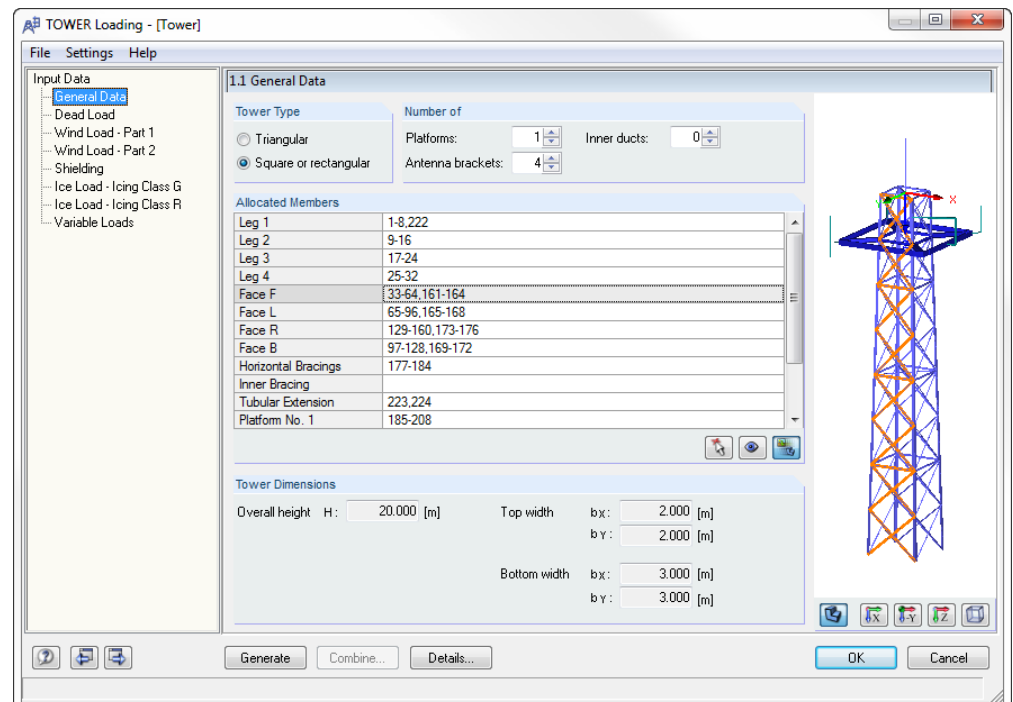


Figure 13.1: Table 1.1 *General Data*

If TOWER Loading recognizes a tower construction exported from the TOWER Structure add-on module, the table *Allocated Members* already contains all recognized tower members. Platforms, antenna brackets and inner ducts defined in TOWER Equipment are displayed together with their number in the upper part of the table.

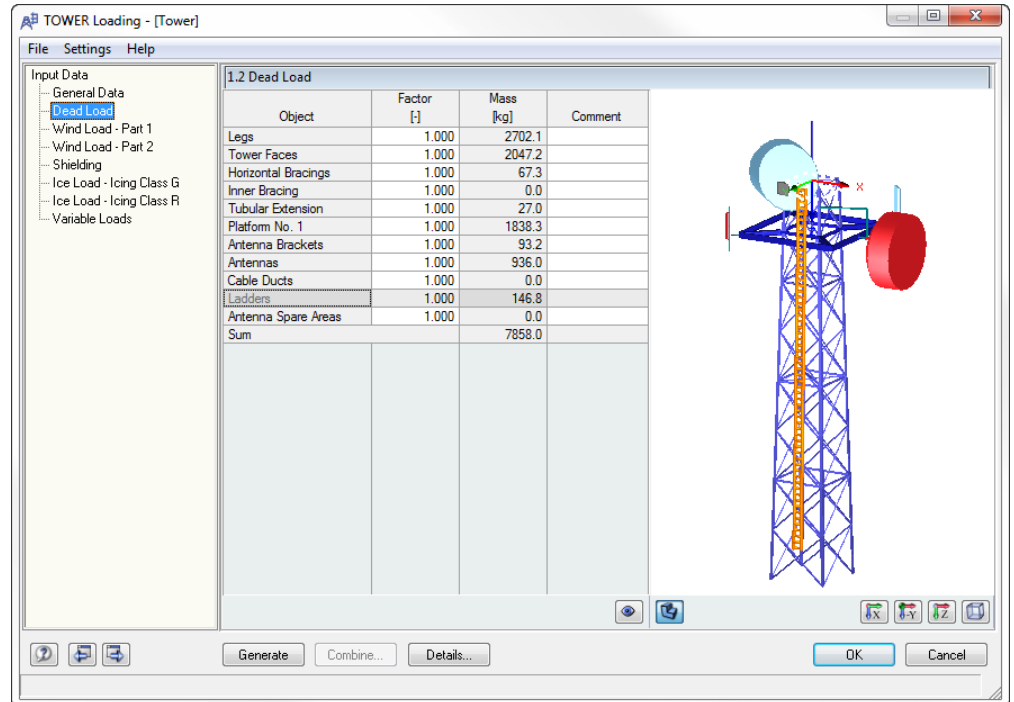


When no data has been exported previously from the add-on modules TOWER Structure or TOWER Equipment and imported to RSTAB/RFEM, you can enter data also manually in table 1.1. Click the button shown on the left to activate the rendering so that you can check and, if necessary, adjust the assignment of members to the given structural tower components displayed in the graphic by selecting the single table rows.

The lower part of the table shows information about the tower's geometrical data.

13.2 Dead Load

In table 1.2 you see the self-weight of the individual structural component groups. The determination is performed automatically, based on the individual cross-section areas and the assigned material.



The screenshot shows the 'TOWER Loading - [Tower]' window. On the left is a tree view under 'Input Data' with 'Dead Load' selected. The main area displays a table titled '1.2 Dead Load' with columns for Object, Factor, Mass, and Comment. To the right is a 3D model of a tower structure with various components highlighted in different colors.

Object	Factor [-]	Mass [kg]	Comment
Legs	1.000	2702.1	
Tower Faces	1.000	2047.2	
Horizontal Bracings	1.000	67.3	
Inner Bracing	1.000	0.0	
Tubular Extension	1.000	27.0	
Platform No. 1	1.000	1838.3	
Antenna Brackets	1.000	93.2	
Antennas	1.000	936.0	
Cable Ducts	1.000	0.0	
Ladders	1.000	146.8	
Antenna Spare Areas	1.000	0.0	
Sum		7858.0	

Figure 13.2: Table 1.2 Dead Load

To take into account galvanization of cross-sections or maybe additional weights from coupling elements, it is possible to define an increase *Factor* for the individual component groups.

13.3 Wind Load - Part 1

Wind is a design-relevant loading for tower structures because of their specific type of construction and building height. In table 1.3, you define the basic data required to determine the wind load considering site and geometry of the tower.

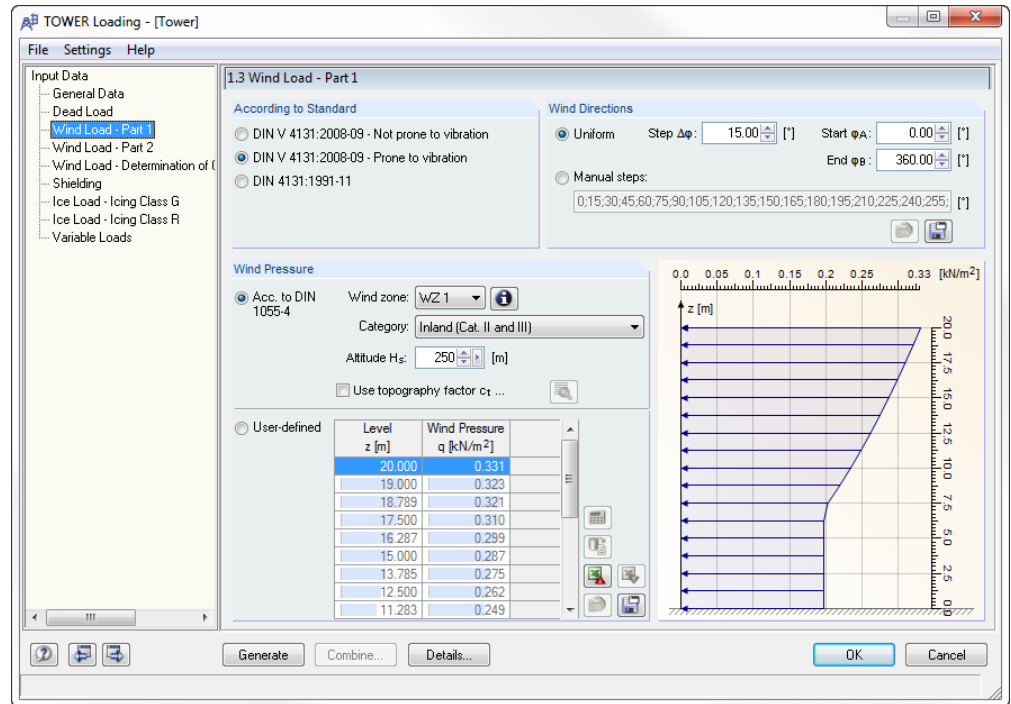


Figure 13.3: Table 1.3 *Wind Load - Part 1*

Structural systems of towers may vibrate due to the wind's gustiness. Dynamic calculations are necessary to determine such vibrations. DIN 1055-4, annex C, offers the possibility to reduce this dynamic calculation to a structural analysis by introducing a gust reaction factor. When you select the standard *DIN V 4131:2008-09 - Prone to vibration*, TOWER Loading will determine the total wind load by taking into account the gust reaction factor.

To determine the gust response factor, the calculation of the first eigenfrequency is required. The program determines the lowest natural frequency in its background. More options to determine the gust reaction factor are described in chapter 13.5 on page 60.

Wind Directions

According to DIN 4131, annex A 1.6, the wind direction must be varying around the tower with a step size of 15°. TOWER Loading presets the specifications of the standard by default. In table 1.3 you are able to modify the steps as well as the start and end angle or to define your own steps manually.

The single steps created by the add-on module for the wind direction can be stored as template. When the manual definition of steps is activated, you can import saved templates by means of the button shown on the left.



Wind pressure according to DIN 1055-4

TOWER Loading determines the wind load distribution along the structure height as a function of *Wind zone*, *terrain Category*, *Altitude* and, if applicable, *topography factor* c_t . Definitions of wind zones and terrain categories are specified in DIN 1055-4, annex A and B.



With the topography factor c_t you can influence the mean wind velocity by taking into account the site of the tower construction with regard to wind-significant terrain like hills or discontinuities in the landscape. When the option for c_t is activated, use the [Details] button to open the detail settings for this factor.

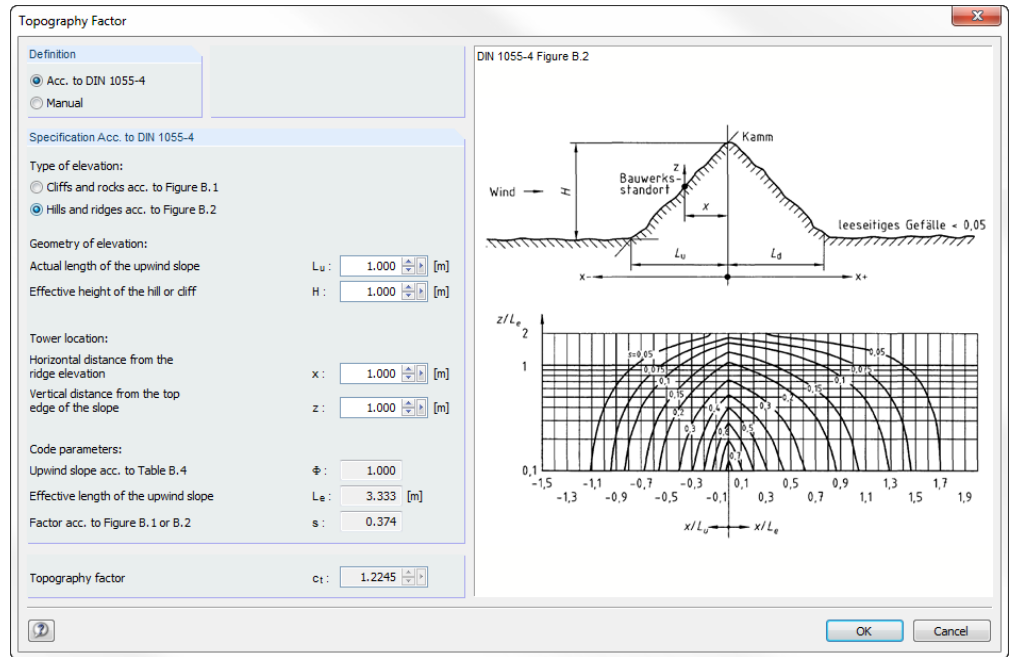


Figure 13.4: Dialog box *Topography Factor*

Based on the boundary conditions given in the dialog box, the program determines the topography factor c_t automatically. It is also possible to specify the factor manually by means of the selection available in the detailed settings.

Wind pressure according to DIN 4131

When you determine the wind load according to DIN 4131:1991-1, TOWER Loading considers the four wind zones and the altitude H_s (if WZ1 is set). The terrain topography can be taken into account by activating the increase for wind pressure Δq .

User-defined wind pressure

If wind pressure conditions are given which cannot be categorized according to the available standards, you can define the wind pressure curve also manually.



To perform a quick definition of wind pressure values with small increments, you can use the import/export function to MS Excel.

The following Table 13.1 describes the buttons required for the manual definition of wind pressure curves.

Button	Function
	Opens Windows calculator and imports calculated value.
	Imports parameters from the standard currently set.
	Exports table values to MS Excel.
	Imports table values from opened MS Excel file.
	Imports table values saved as template.
	Saves current table values as template.

Table 13.1: Buttons for manual definition of wind pressure curve

13.4 Wind Load - Part 2

TOWER Loading determines the aerodynamic force factors automatically by default.

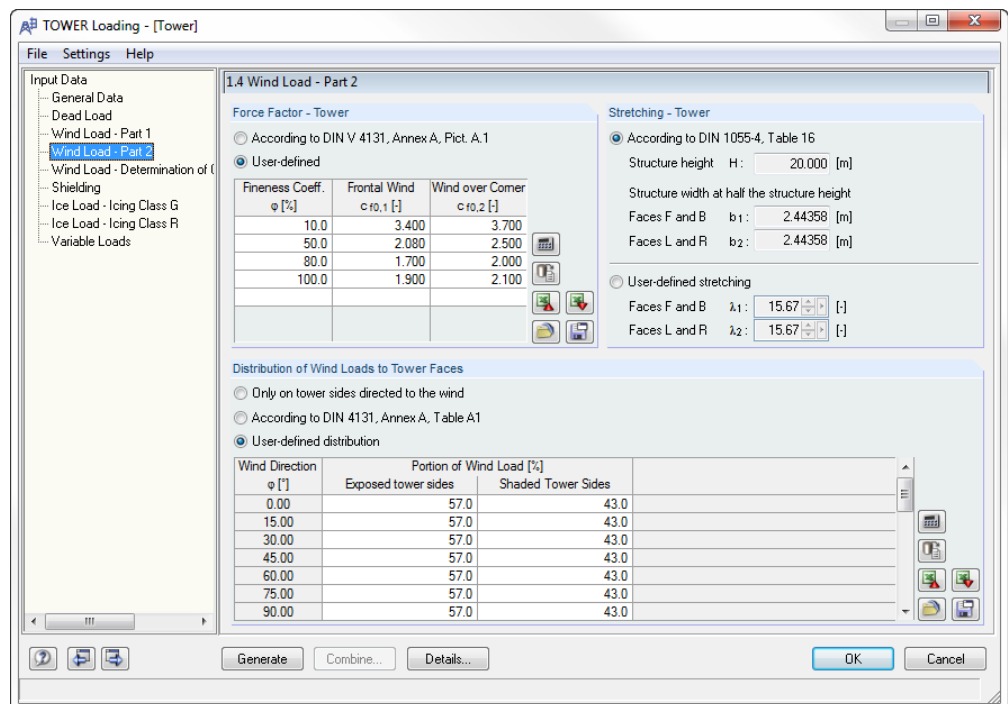


Figure 13.5: Table 1.4 Wind Load - Part 2

Force Factor

The force factor $c_{f,0}$ for spatial frameworks is determined with the fineness coefficient and the wind direction. Again, you have the possibility to specify user-defined input manually by using the buttons described in Table 13.1.

Stretching

Here, standards are using different descriptions. According to DIN 4131, strain corresponds to stretching λ . DIN 1055-4 informs us about effective slenderness. By default, the strain is determined according to DIN 1055-4, table 16.

If you want to take user-defined values as a basis for calculations, you can enter strains separately for the parallel tower sides F / B and L / R.

Distribution of Wind Loads to Tower Faces

With the default setting for wind loads, the upwind tower surfaces (windward) are stressed by 100 %, and the downwind areas (leeward) remain unloaded. A proportional loading of surfaces according to DIN 4131, annex a, table A1, and a user-defined distribution are possible. You can use the common buttons (see Table 13.1 on page 59) for the manual definition.

13.5 Determination of Gust Response Factor

TOWER Loading determines the gust response factor automatically. Based on the smallest eigenmode of the tower structure, the program determines the factor for structural systems that are prone to vibration according to formulas of DIN 1055-4, taking into account the specified calculation parameters.

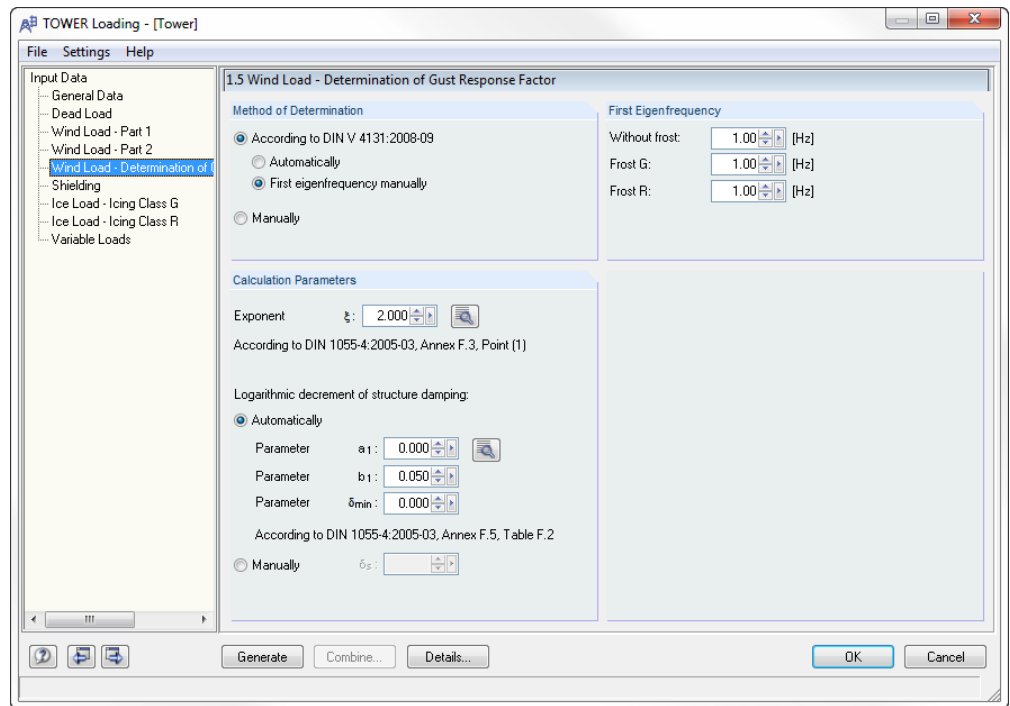


Figure 13.6: Table 1.5 Wind Load - Determination of Gust Response Factor

Gust Response Factor	
Without frost:	1.00 [-]
Frost G:	1.00 [-]
Frost R:	1.00 [-]

When the structure's first eigenfrequency is known, you can enter it into the input fields. Thus, the gust reaction factor will be determined on the basis of the manually defined value and the calculation parameters. Defining the gust response factor manually may be useful for comparative calculations to ensure the same initial situation. When you activate the option *Manually*, you can specify the gust response factor for the load cases self-weight, self-weight with ice and icing class G, or self-weight with icing class R.

13.6 Shielding

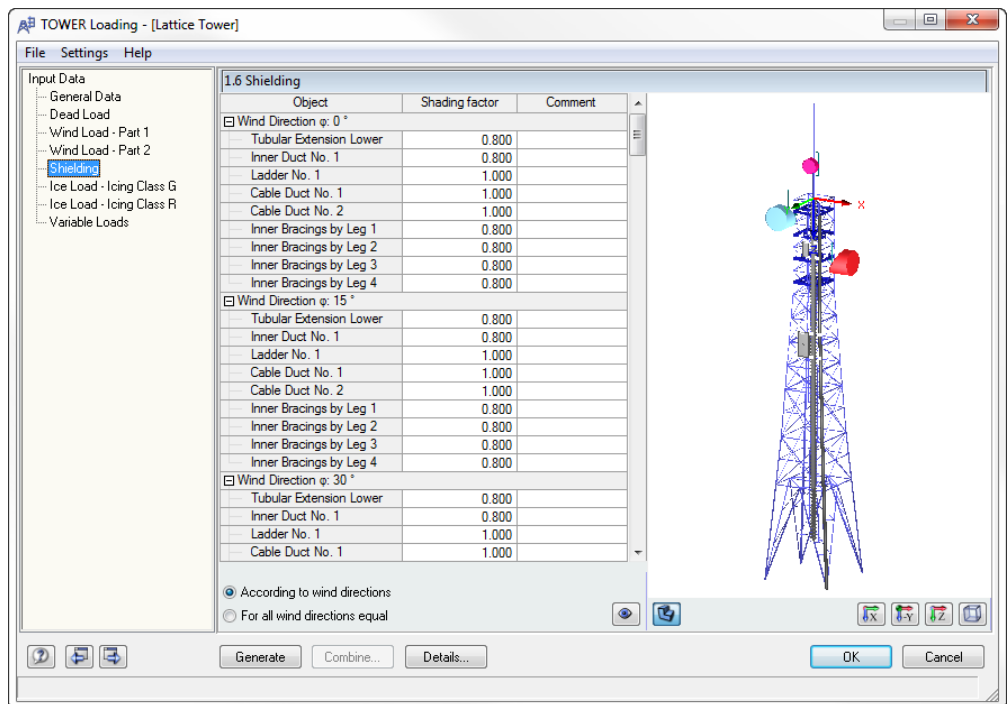


Figure 13.7: Table 1.6 Shielding

According to DIN V 4131, you can calculate with a reduced wind load when taking into account wind shadowing effects for installations and exterior equipment. In table 1.6 you have the possibility to adjust the *Shading factor*.

13.7 Ice Load - Icing Class G

Table 1.7 defines ice loads for a uniform all-over ice coating of structural components.

The icing classes and the specific weight for ice are preset according to DIN 1055-5, annex A.

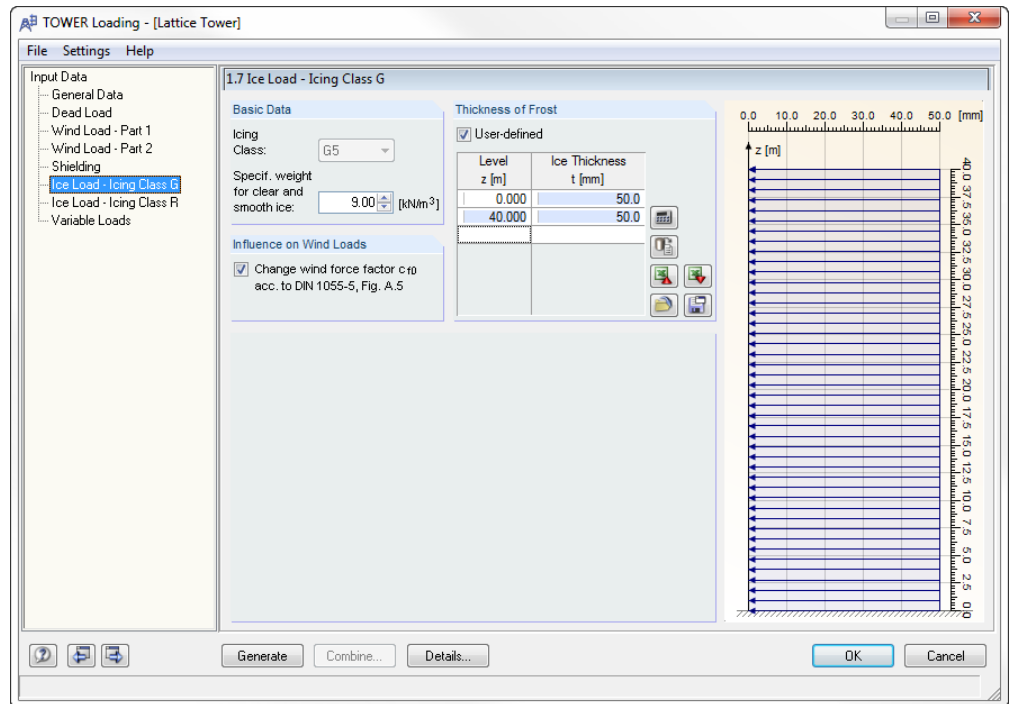


Figure 13.8: Table 1.7 Ice Load - Icing Class G

The thickness of frost is predefined depending on the icing class. Moreover, it distributed uniformly along the tower height. You can enter user-defined frost thicknesses when the check box *User-defined* is ticked.

In table 1.7, it is also possible to activate the option for considering the increasing of cross-sections when wind loads are determined according to DIN 1055-5, annex A, chapter A.4.

13.8 Ice Load - Icing Class R

In case the tower structure is iced over, the prevailing wind direction may lead to the creation of a one-side ice flag increasing against the wind direction.

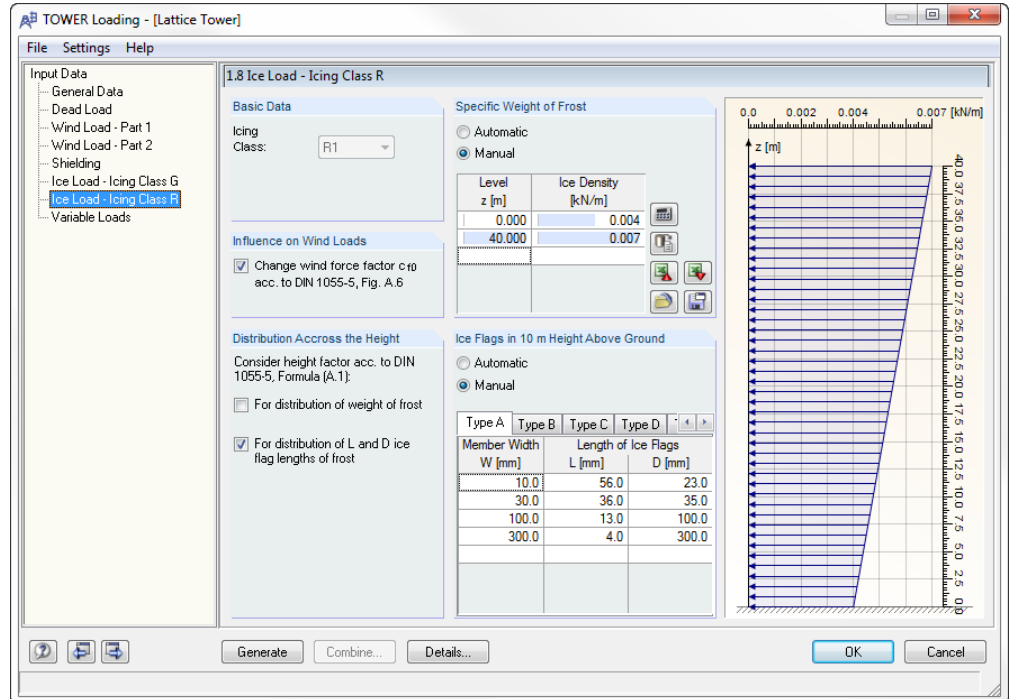
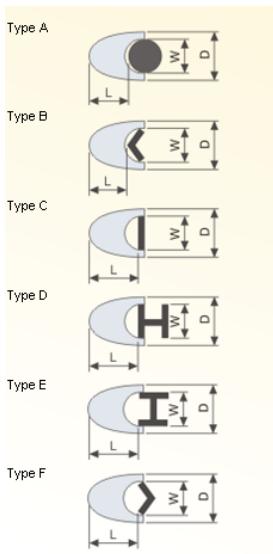


Figure 13.9: Table 1.8 Ice Load - Icing Class R



In table 1.8 you are able to consider this type of frost for the load generation by defining appropriate boundary conditions according to DIN 1055-5. The different ice flags are recognized automatically by TOWER Loading in reference to the cross-section type.

It is also possible to define the shape of the ice flag manually by specifying the length L and the width D . The entered data can influence the wind contact surface arising from the frost.

13.9 Details

Details ...

Use the button shown on the left to open the detail settings affecting the wind load generation. Furthermore, you can reduce the total wind load and consider ice and human loads.

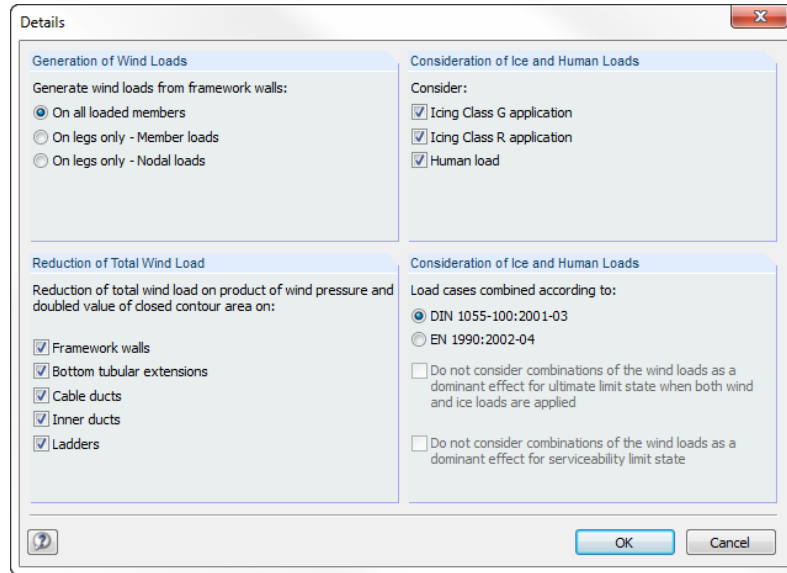


Figure 13.10: Dialog box *Details*

13.10 Variable Loads

Traffic loads to be applied to tower buildings are specified in DIN V 4131 chapter 6.6. Table 1.9 presets the corresponding load values accordingly.

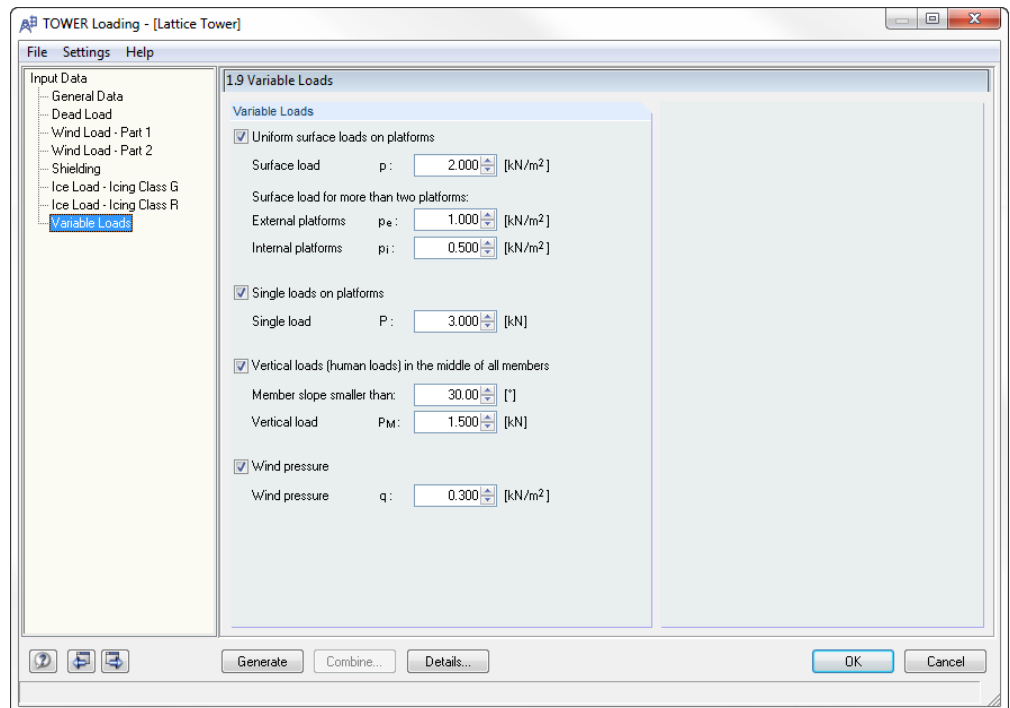


Figure 13.11: Tower 1.9 *Variable Loads*

You can enter individual load values for variable loads when requirements are differing from the standard.

14. Results

Generate

When all data has been completely entered, you start generating the loading by clicking the [Generate] button.

When the load generation was successful, you see an overview of the results tables available in the module navigator of TOWER Loading.

14.1 Load Cases

Table 2.1 shows an overview about all created load cases. Each load case has been given a clear and representative name by the program.

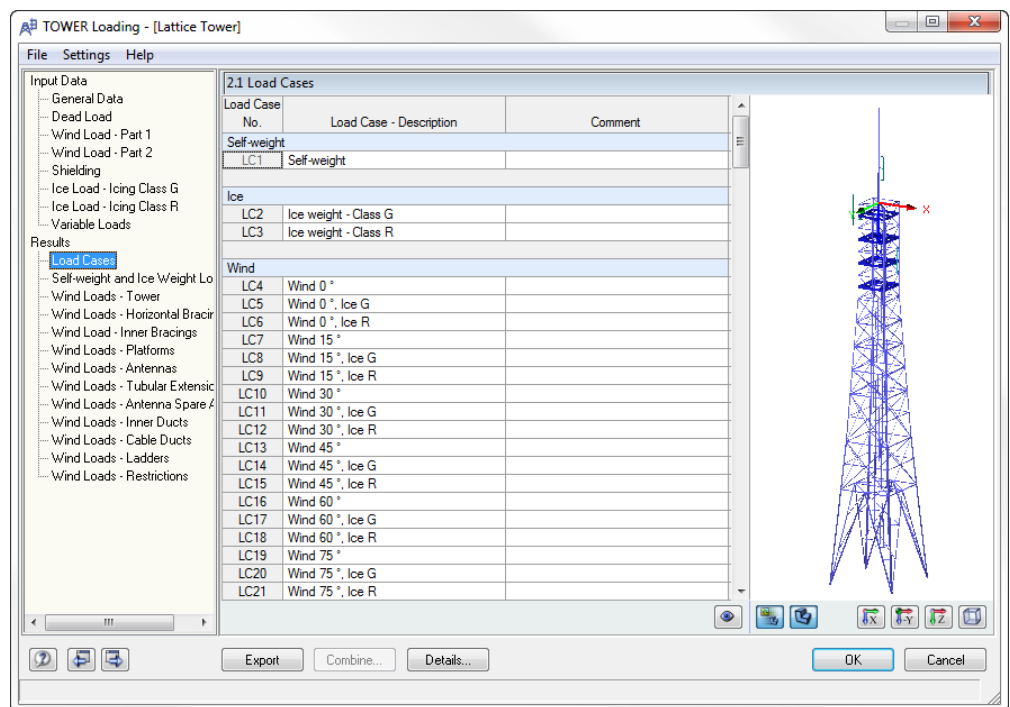


Figure 14.1: Table 2.1 Load Cases

14.2 Self-weight and Ice Weight Loads

Self-weight and ice weight for icing classes G and R are displayed in table 2.2, sorted by tower structure components and equipment.

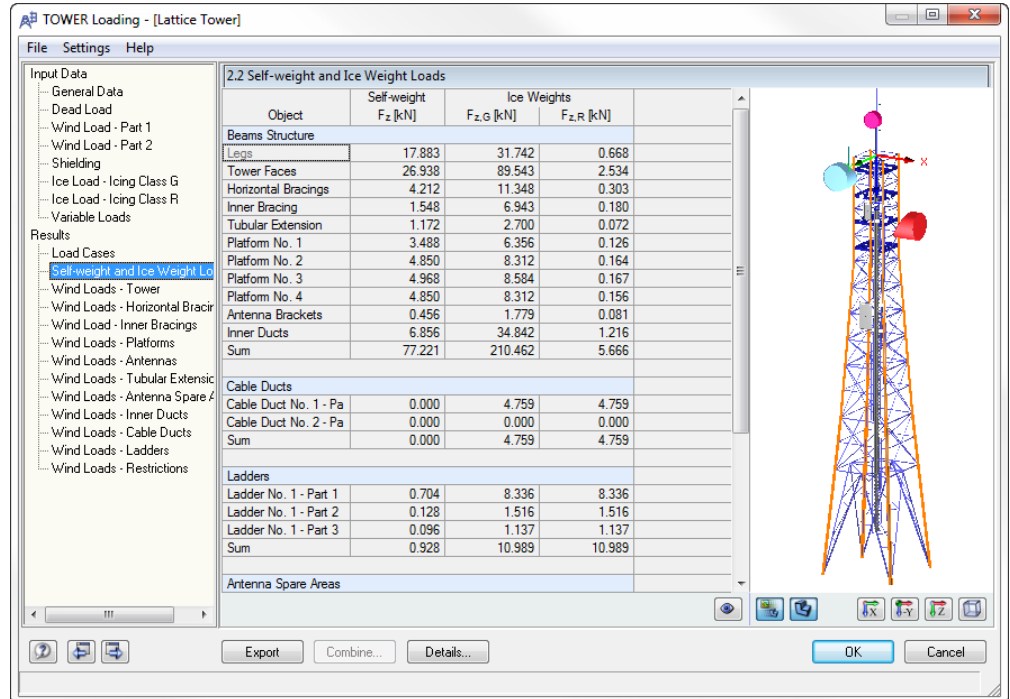


Figure 14.2: Table 2.2 Self-weight and Ice Weight Loads

The output includes a summarizing total sum for all groups of structural components.

14.3 Wind Loads - Gust Reaction Factor

When you have set the wind load determination according to *DIN V 4131:2008-09 - Prone to vibration* or *DIN 4131:1991* in table 1.3, TOWER Loading calculates the gust reaction factor additionally. The results are shown in table 2.3 (see Figure 14.3 on the following page).

A resolver for eigenvalues integrated in the program determines the first eigenfrequency that is required for the calculation. It is also displayed in the results table. See chapter 13.5 on page 60 to learn how to determine the gust response factor with a given first eigenfrequency or how to specify it manually.

The program determines the gust response factor G also for tower structures charged by ice loading. *Details* of the calculation are displayed in the lower part of the table. To switch between the results, click into the respective table row in the table part above.

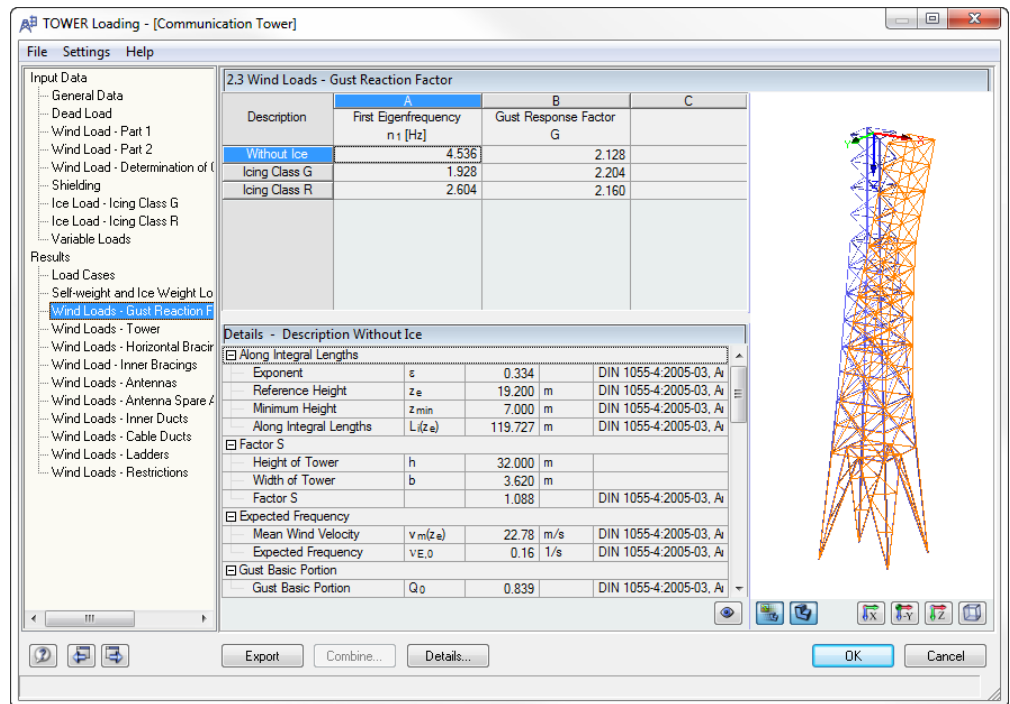


Figure 14.3: Table 2.3 Wind Loads - Gust Reaction Factor

14.4 Wind Loads - Tower

Table 2.4 lists all determined wind loads acting on the tower structure.

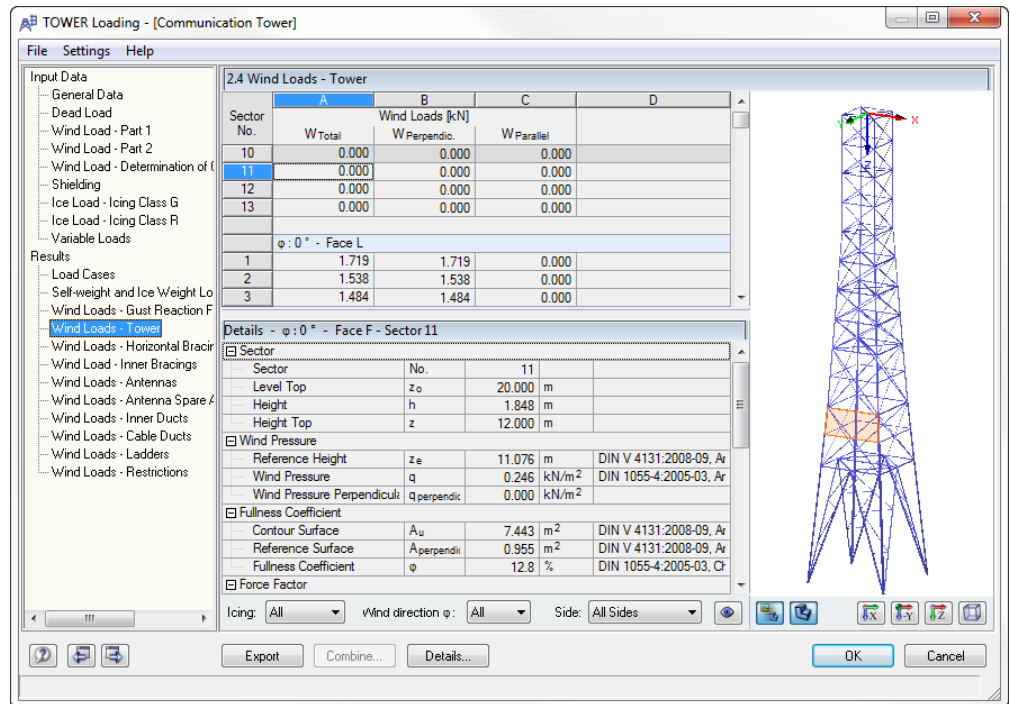
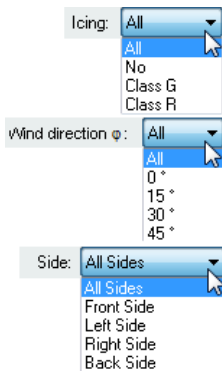


Figure 14.4: Table 2.4 Wind Loads - Tower

Due to the standard's requirement to apply wind loading in sub-steps to the tower structure, the results table may display a large amount of data, depending on the step specifica-



tion set in table 1.3. The upper table shows the results sorted by wind direction and tower face. The bottom part of the table lists the corresponding result details.

To provide you with comfortable functions for the results evaluation, three picklists are available at the bottom edge of the results table. Use the lists to select the contents of the output table as follows:

The *Icing* option allows you to display the wind loading with the respective icing class or without frost. With the *Wind direction* list you can jump to the respective sub-step. Set a tower *Side* to select the results sorted by the four tower faces,

14.5 Wind Loads - Horizontal Bracings

Table 2.5 presents the wind loads determined for horizontal bracings.

The screenshot shows the '2.5 Wind Loads - Horizontal Bracings' table with the following data:

Member No.	W _φ	W _x	W _y
254	0.075	0.075	0.000
255	0.075	0.075	0.000
256	0.075	0.075	0.000
φ: 0° - Horiz. bracing 2 - z: 28.000 m			
257	0.070	0.070	0.000
258	0.070	0.070	0.000
259	0.070	0.070	0.000
260	0.070	0.070	0.000

The details for Member 258 are as follows:

Property	Value	Standard
Bracing No.	2	
Level z	28.000 m	
Member No.	258	
Reference Height z _a	28.000 m	DIN 1055-4:2005-03, A
Wind Pressure q	0.392 kN/m ²	DIN 1055-4:2005-03, A
Force Factor c _{f,0}	2.000	DIN 1055-4:2005-03, C1
Length of all Members in Or	3.480 m	
Height of Member h	0.037 m	
Slenderness λ	70.000	DIN 1055-4:2005-03, C1
Reduction Factor ψ ₂	0.911	DIN 1055-4:2005-03, C1

Figure 14.5: Table 2.5 Wind Loads - Horizontal Bracings

The results are evaluated by members. The results output can be filtered by using the picklists for *Icing* and *Wind direction* already described in chapter 14.4.

The results tables 2.6 to 2.13 for further equipment results can be evaluated in the same way as described for the tables above. Depending on the equipment, results data for wind loads refer to the global axes or are indicated by the direction *Perpendicular* and *Parallel*.

For example, when no cable ducts were defined in the add-on module TOWER Equipment, the add-on module TOWER Loading will show no results table for this equipment. In this case, the table numbering is not continuous.



14.6 Wind Loads - Restrictions

According to DIN V 4131, annex A, section A.2.6.2.2, it is allowed to reduce the total wind load to the value $2.0 \cdot A_c \cdot q$.

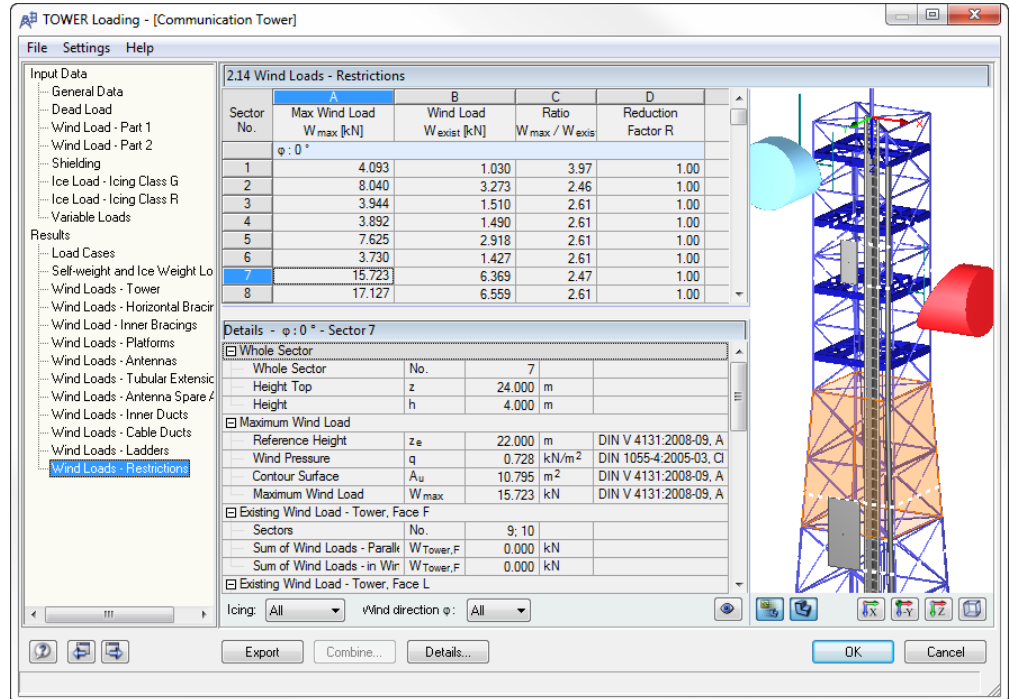


Figure 14.6: Table 2.14 Wind Loads - Restrictions

The program compares the existing total wind loading with the limit wind loading and introduces the reduction factor R when the limit load is exceeded.

14.7 Export of Results

Export

For further data editing and structure design you have to transfer the generated load cases to RSTAB/RFEM. Use the [Export] button to start the transfer of the load cases displayed in table 2.1. In addition, TOWER Loading creates the load combination *CO1 Man Load*. According to DIN V 4131, section 6.6, it is allowed to design the human load as single force in combination with the wind load by using a consistent wind pressure of $q = 0.3 \text{ kN/m}^2$.

Combine...

When a license for the add-on module RSCOMBI/RF-COMBI is available, the button shown on the left is enabled after the successful export of load cases. Use this button to start the COMBI add-on module directly out of TOWER Loading. The COMBI program helps you to generate possible load combinations.

15. General Functions

You can use the menu bar at the top of the TOWER module window to access various general functions.

15.1 Delete a TOWER Case

To delete a TOWER case,

select **Delete Case** on the **File** menu in the TOWER add-on module.

When you confirm the selection, the program deletes the case and you quit the add-on module. Data that has already been exported is preserved in RSTAB/RFEM.

15.2 Units and Decimal Places

The units and decimal places for RSTAB/RFEM and all add-on modules are managed in one global dialog box. To open the dialog box out of the TOWER module,

select **Units and Decimal Places** on the **Settings** menu.

The following dialog box opens, which you already know from RSTAB/RFEM. The TOWER add-on module is preset.

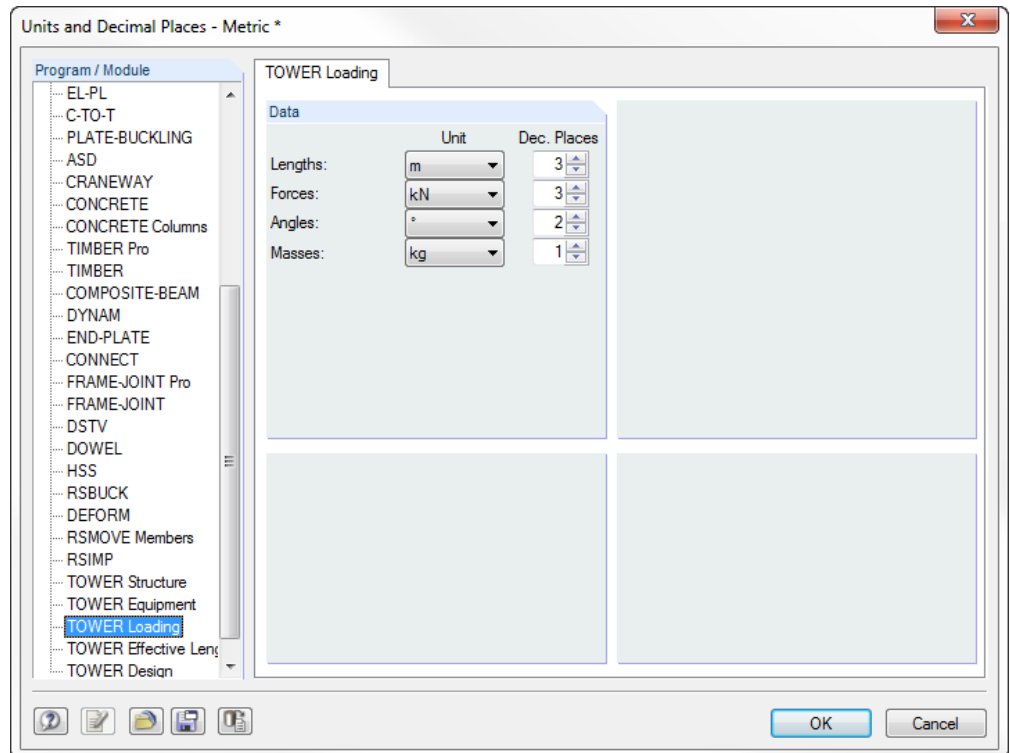


Figure 15.1: Dialog box *Units and Decimal Places*



The settings can be saved as user profile to reuse them in other structures.

A Literature

- [1] DIN V 4131:2008-09 Antenna steel structures
- [2] DIN 1055-4:2005-03 Actions on structural systems, part 4: Wind loads
- [3] DIN 1055-5:2005-07 Actions on structural systems, part 5: Snow and ice loads

B Index

A		Inner ducts	39
Antenna brackets.....	35	Installation	6
Antenna groups.....	36	J	
Antenna spare areas	38	Jib.....	22
Antennas	37	L	
B		Ladders.....	41
Bracing type.....	17	Leg extensions.....	9
Bracings	9, 15, 19, 20, 48, 68	Legs.....	13
Buckling lengths	44, 51	Load cases	65
C		M	
Cable ducts	40	Material library.....	12
Comment.....	9	Member releases	23
Coordinate system.....	49	Member rotations	24
Cross arms	21	N	
Cross-section information.....	11	Navigator	8, 29, 55
Cross-section library.....	10	Node restraints.....	47
Cross-sections	10	Non-truss members.....	51
D		North.....	30
Decimal places	26, 43, 52, 70	O	
Distribution of wind loads	60	Options	9
E		P	
Effective length factor	50	Parts list	25
Effective lengths	47, 50, 51	Plan	8
Eigenfrequency.....	66	Platforms.....	30, 33
Export	25, 42, 51, 69	Prone to vibration	57
F		Q	
Force factor	59	Quit TOWER	8, 29, 55
Frost	62	S	
G		Selecting tables	8, 29, 55
General data	46, 55	Self-weight.....	56, 66
Girts.....	18	Shading factor.....	61
Gust reaction factor.....	66	Shielding	61
Gust response factor.....	60	Slopes.....	13
H		Start program	7, 27, 44, 53
Human loads	64	Start TOWER.....	7, 27, 44, 53
I		Stretching.....	59
Ice flag.....	63	T	
Ice loads	62, 63	Tables.....	8, 55
Ice weight.....	66	TOWER Effective Lengths	44

TOWER Equipment 27
Tower legs 48
TOWER Loading 53
Tower segments 13
TOWER Structure 7
Tower support 9
Tower type 8
Truss members 50
Tubular extension 34

U

Units 26, 43, 52, 70
User profile 26, 43, 70

V

Variable loads 64

W

Width change 13
Wind direction 57, 59
Wind loads 57, 59, 66, 67
Wind pressure 58