Program

TIMBER

For use with Windows 95/98/ME/NT 4.0/2000

Stress Analysis
Section Optimization
Parts List

User Manual

Version: August 2001

© ING.-SOFTWARE DLUBAL GMBH
Am Zellweg 2 • 93464 Tiefenbach • Germany

Telephone: +49 - 96 73 – 92 03 23
Telefax: +49 - 96 73 - 17 70
eMail: info@dlubal.com
Internet: http://www.dlubal.com

1. Introduction

1.1 About TIMBER for Windows

The TIMBER for Windows user manual begins with some general background knowledge and hints for use. It has always been possible to use the DLUBAL STEEL module for stress analysis in timber construction, but now, through adaptation of stresses and the integration of a suitable material library, TIMBER for Windows allows the engineer to carry out exact stress analyses of timber sections according to DIN 1052, including buckling analyses by means of the ω procedure.

As with all additional modules, TIMBER is fully integrated into RSTAB 5 for Windows. Results from TIMBER designs can be integrated in the RSTAB printout report with the result that the entire analysis is presented in one concise, attractive report. Optimized sections can be easily transferred to RSTAB for use in additional calculations. The user also has full use of all profiles in the RSTAB sections library, which can be expanded and also contains an interface to the DLUBAL SHAPE and DICKQ (German only) modules.

TIMBER for Windows is also programmed with 32-bit technology, which among other things, results in quick computing time.

Finally, the [F1] key can be used to obtain direct online assistance while using the TIMBER modules.

We hope RSTAB and TIMBER will contribute to the success of your work and company.

1.2 The TIMBER Team

The following people contributed to the development of TIMBER for Windows:

- **Program Coordinators:**
  Dipl.-Ing. Georg Dlubal
  Dipl.-Ing. (FH) Walter Rustler
  Dipl.-Ing. Pavel Bartoš
- **Programmers:**
  Roman Horenovský
  Dr.-Ing. Jaroslav Lain
  Mgr. Jaroslav Krul
- **Program Testing:**
  Dipl.-Ing. (FH) Walter Rustler
  Dipl.-Ing. (FH) Matthias Entenmann
- **User Manual and Help System:**
  Dipl.-Ing. (FH) Matthias Entenmann
- **English Translation:**
  Jana Rustler

2. TIMBER Installation

2.1 System Requirements

The following minimum system requirements are recommended to successfully run TIMBER:

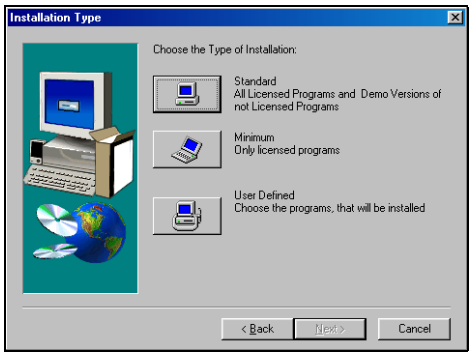
- Operating System Windows 95 / 98 / NT 4.0 / 2000
- 200 MHz Processor
- 32 MB Memory
- CD-ROM- and 3.5"-disk drive for the Installation
- 2 GB total hard disk capacity with about 15 MB reserved for installation
- Graphic card with 4 MB and a resolution of 1024 x 768 pixels

With the exception of the operating system, no product recommendations are made.

TIMBER and RSTAB basically run on all systems that fulfill the system requirements. Your computer does not need to have “Intel Inside”, and it is also unnecessary to have an expensive 3D graphic card. Because TIMBER and RSTAB are generally used for extensive calculations, the phrase “more is better” holds true.

2.2 Installation Process

As a licensed TIMBER user, choose [Standard] at the point during the installation process to install all admissible programs – including TIMBER. At the same time, all other available additional modules are installed as limited demo versions.



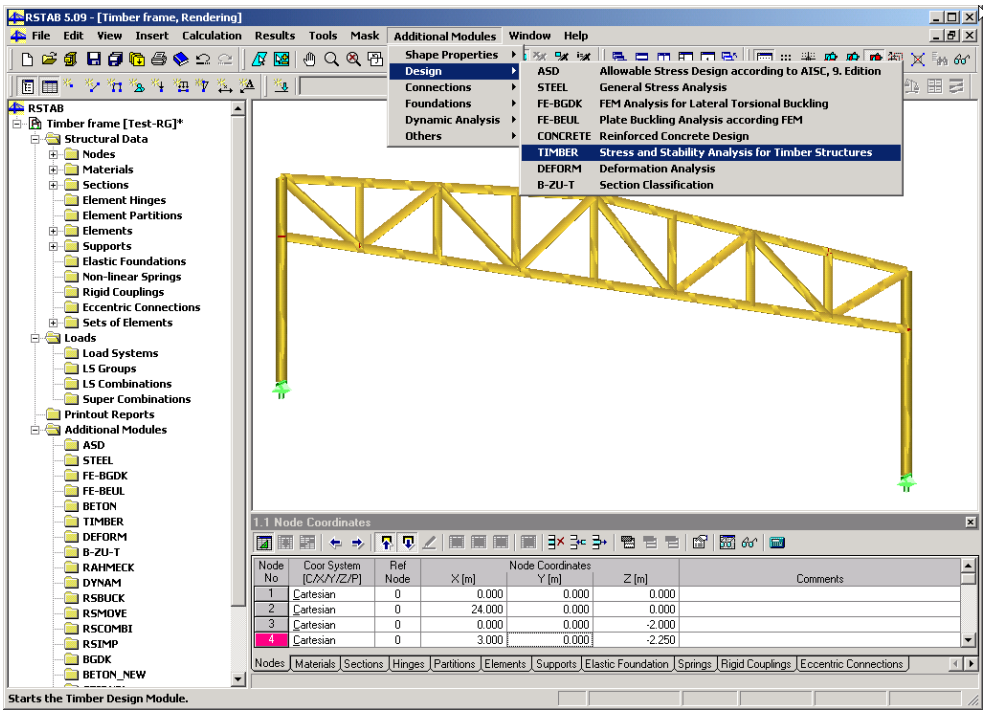
Installation options

If you lack free hard disk capacity, or it needs to be saved, choose [Minimum]. Only licensed full versions will then be installed. The rest of the installation process, including the authorization process, is the same as for RSTAB, explained in detail in the RSTAB manual.

3. Working with TIMBER

3.1 Starting TIMBER

The TIMBER module can be started from either the *Additional Modules* → *Design* → *TIMBER* menu or through the tree item [Additional Modules] in the *Position Navigator* or *Project Navigator*.



Starting TIMBER with the Additional Modules Menu or the Navigator

Before opening TIMBER, the position to be designed must first be opened in RSTAB.

TIMBER offers the possibility to analyse RSTAB sections in individual *TIMBER Cases.* This means, that the user can combine, in individual dimensioning cases, any element, load system, LS group and LS combination in the form of a single stress analysis with respect to one or more permissible stress permutations. Alternative stress checks regarding other material grades could be carried out for example. Consequently, the most economical design can be determined very quickly via the parts list and the resulting prices. A typical application of differing dimensioning cases is exemplified by the different limiting boundary conditions of the load cases H and HZ according to DIN 1052.

3.2 Masks

The input of *TIMBER Cases,* as well as the output of the results on the screen, is done in masks. After starting TIMBER, the *TIMBER Navigator* appears to the left. It enables viewing and accessing all available input and output masks. At the top, a pull down list box with all available *TIMBER CASES* can be found. Open the list box by clicking on the [Arrow down] key, after which the appropriate *TIMBER CASE* can be selected. Under the title bar the *File, Edit* and *Help* menus are found. These menus are explained in detail in chapter 3.6.

Access to all masks is controlled either by clicking on the respective item in the *TIMBER Navigator* or by flipping through the masks in succession. Use the [F2] and [F3] keys or

the buttons [<<] and [>>] to flip between the masks. Access to the libraries – if relevant - is possible with the [F7] key or by clicking the right mouse button.

OK and Graphic

With [Graphic], the graphic display of the results of the current TIMBER case is activated. More about graphic display and output of the results is found in chapter 4.

OK

Cancel

Help

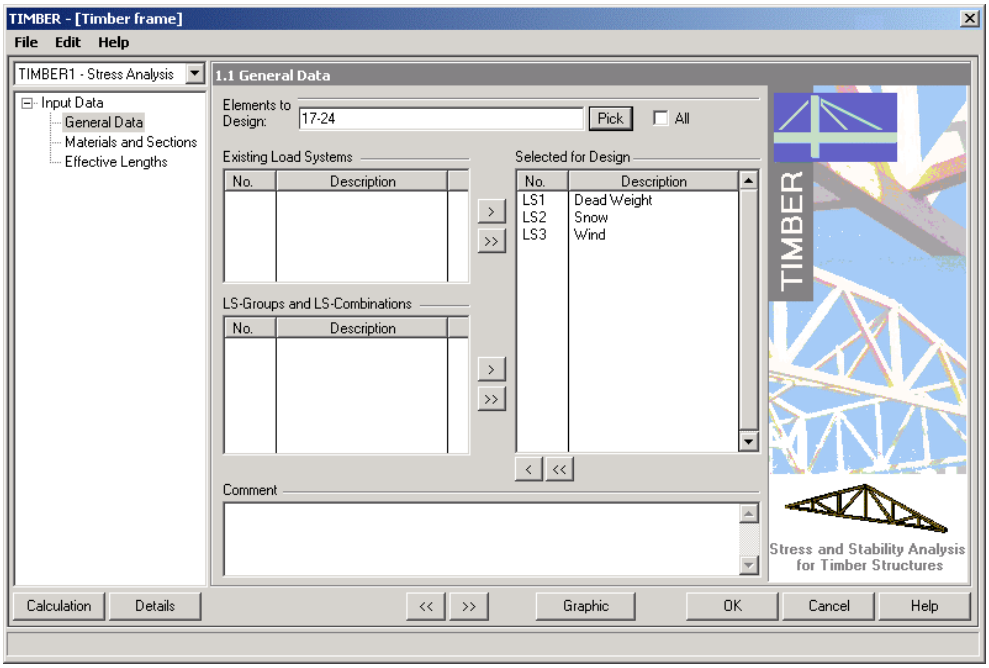
[OK] saves the input and results of TIMBER and closes TIMBER, while [Cancel] closes TIMBER without saving any editing. The button [Help] or the [F1] key starts the online help system.

3.3 Input Masks

All necessary entries for the stress analysis and several parameter adjustments must be made in the input masks.

3.3.1 Mask 1.1 General Data

After starting TIMBER, the window opens with the ***1.1 General Data*** mask.



Mask 1.1 General Data

Pick

The ***Elements to Design*** option is a default setting set on *All*. This can be deactivated by clicking on the check mark in the box to the left of *All*. Then the input line can be edited and the numbers of the elements to design can be entered manually. Use the [Pick] button to make this selection graphically.

> >>

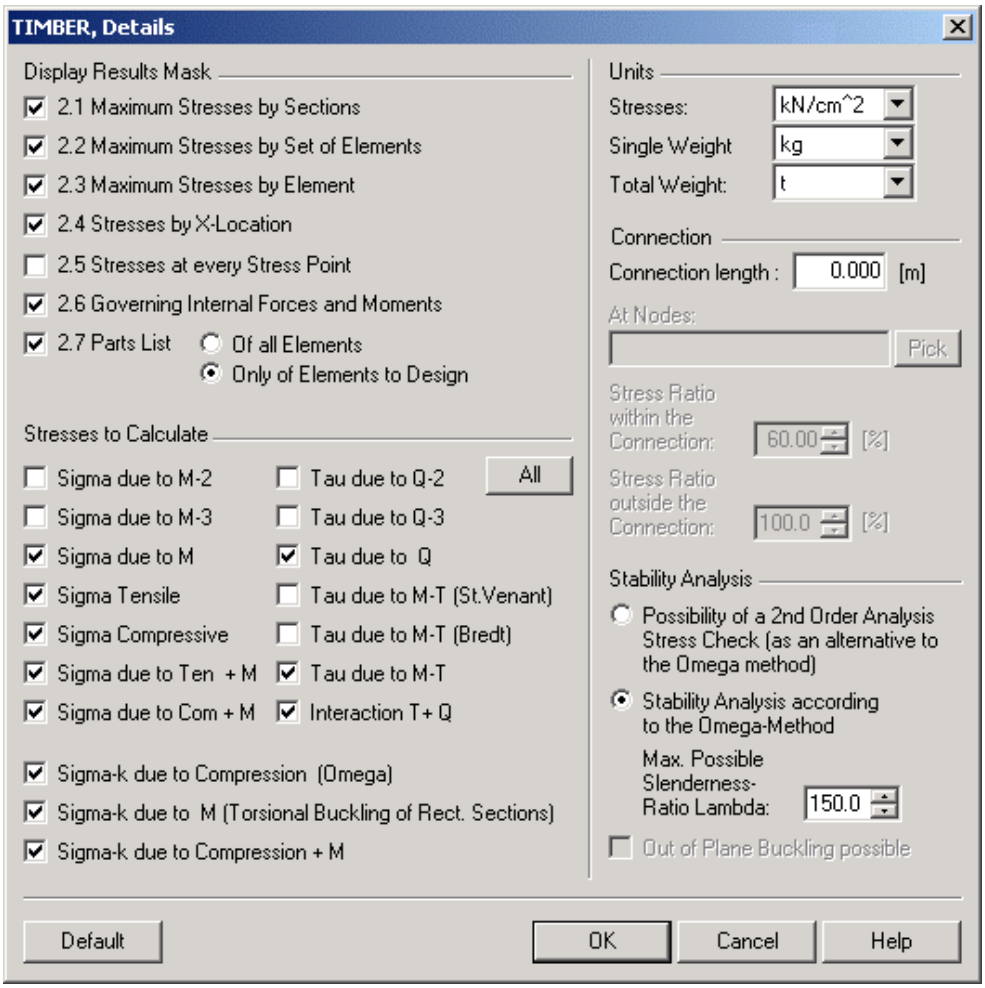
< <<

The ***Existing Load Systems, LS Groups and LS Combinations*** and ***Selected for Design*** lists are located in the middle section. To select one or several load systems, LS groups or LS combinations for the stress analysis, just double click on your choice. It will be moved to the *Selected for Design* list. This can also be done by marking your selected load systems, LS groups and LS combinations and then using the button [>]. If the button [>>] is used, all existing load systems, LS groups or LS combinations are moved to the right. To remove single or all load systems, LS groups or LS combinations, use the buttons [<] or [<<] in the same way.

Calculation

Details

It is also possible to add ***Comments*** to every TIMBER case. Before the [Calculation] is started, check [Details] for the correct settings of several parameters. They are usually set on an appropriate default setting.



TIMBER, Details

The result masks 2.1 to 2.7 can be activated under ***Display Result Masks***. In the *2.6 Governing Internal Forces and Moments* mask you can also select for which stress type the governing internal forces are displayed. Use the [Arrow Down] key to change the selection in the list box. In the *2.7 Parts List* mask, check whether you want the parts list *Of All Elements* or *Only of Elements to Design*.

Under ***Stresses to Calculate,*** you can activate the desired stress type by clicking in the appropriate check boxes. [All] will activate all stress types.

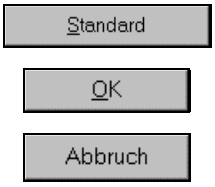
In the ***Units*** area, *Stresses, Single Weight* and *Total Weight* can be set from a list box with the use of the [Arrow Down] button. The unit selections in TIMBER are independent of RSTAB.

Selections under ***Joints*** include *Joint Length, Stress Ratio Inside the Joint* and *Stress Ratio Outside the Joint.*

In addition, set whether the ***Stability Analysis*** is to be performed. When *Optional 2nd Order Analysis Stress Check (as an alternative to the Omega method)* is checked, the TIMBER program will perform no explicit buckling check. However, if one or several load groups have been defined in RSTAB, the user can perform a buckling check by means of a conventional stress analysis using the governing forces and moments resulting from a second order analysis. Similarly, the *Stability Analysis according to the Omega-Method* can be done (see chapter 3.4). When the Omega method is selected, the *1.3 Effective Length* mask will open once the *TIMBER Details* dialog is closed. Mask 1.3 and the *Out of Plane Buckling Possible* option in the *Details* dialog are explained in chapter 3.3.3.

The user can define the *Maximal Permissible Slenderness Ratio Lambda*. The default value of 150 is the maximal allowable value for compression members according to DIN 1052, Part 1. Other than typing in a value, the small arrow buttons can be used to increase or de-

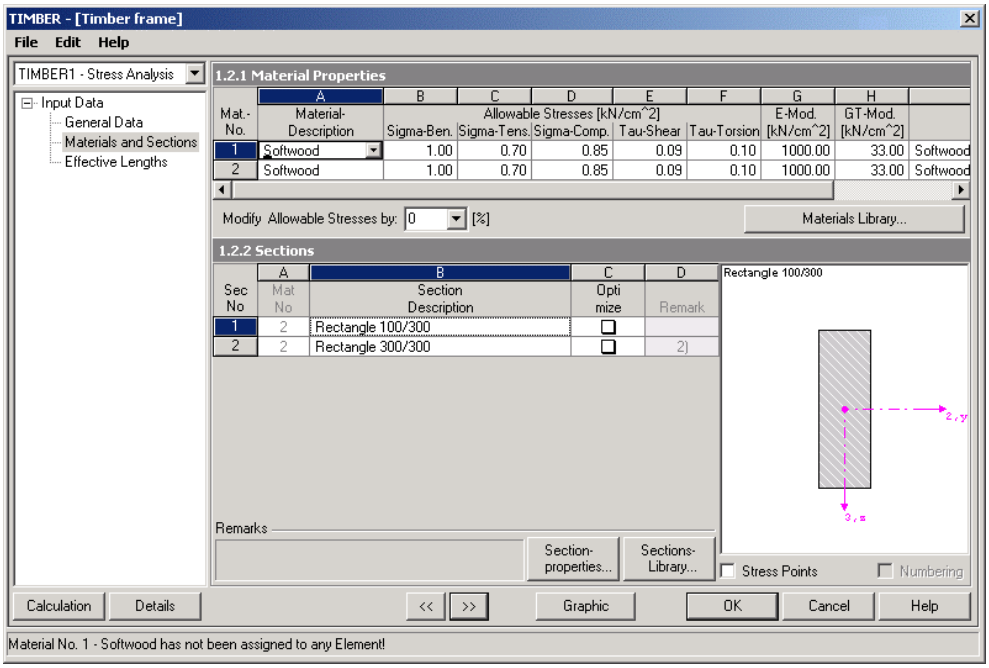
crease this input. If the maximum slenderness ratio of an element has been exceeded, then the analysis requirements according to the Omega method, in contrast to the that of limiting stress σ_K, are not fulfilled.



The [Default] button restores all presettings in this mask. [OK] closes the dialog with editing applied. [Cancel] closes the dialog without applying changes.

3.3.2 Mask 1.2 Materials and Sections

In the upper section of this two-part mask, the allowable stresses to be compared with the calculated stresses are entered. For a vast number of materials, the corresponding values are stored in the [Materials Library] and are automatically applied by the TIMBER program. The values listed in mask ***1.2.1 Material Properties*** can be altered manually or replaced by another set of material data from the materials library at any time.

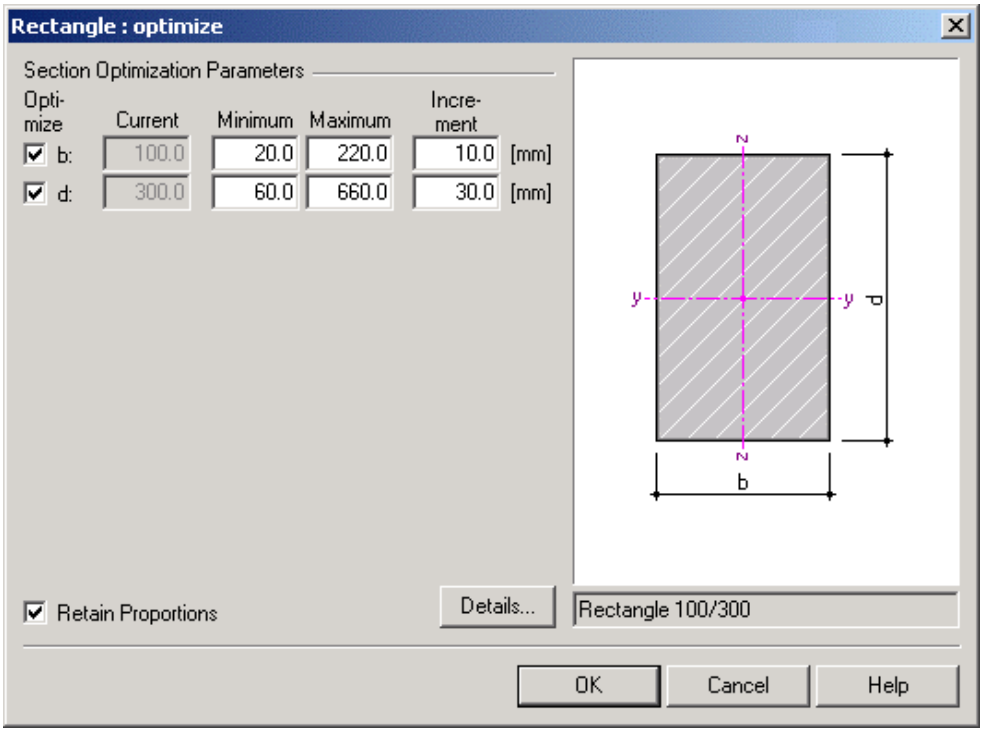


Mask 1.2 Material properties and Sections

The [F7] key or the context menu can also be used to open the materials library. More information about the Materials library can be found in chapter 3.6.2.

In the lower mask *1.2.2 Sections* is a list of all sections to be designed with the *Mat. No.* input in column A. *Optimize* each respective section by clicking on its box in column C. When checked, the *Optimization Parameters* dialog opens. In addition, checking a box in column C then offers a small [Downward Arrow] button which, when used, offers *Yes, No* and *Advanced* functions. If ,during calculation, it turns out that a smaller section from the list is also sufficient, this will be employed during the activated optimization. In this case, TIMBER calculates the required moment of inertia with internal forces from RSTAB and and uses the section which comes closest to a 100% utilization of the section's capacity. When this happens, the *Remarks* in column D will show the footnote *3).* At the bottom of mask 1.2.2, the *Remarks* are explained and here you see *3) The Section will be optimized, i.e. the Section with the most optimal stress ratio will be sought out !*

With certain sections, the option *Details* becomes additionally available in the context menu of the Section. Selecting this option opens the *Optimization Parameters* dialog.



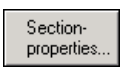
Optimization Parameters

Determine the ***Section Optimization Parameters***, by initially selecting the fundament parameters with the mouse under Optimization Parameters. Underneath you can see the current dimensions of the respective available parameter. As soon as you have chosen one or several parameters for optimizing, you can stipulate in the *Maximum/Minimum* fields the dimension's upper and lower limits as well as the *increment*. In addition, you can *Retain Proportions*. If this position is marked, the section will retain its initial proportions as the section is enlarged during the optimization process.

During the selection of the optimization parameters, it must be noted that these must be chosen with care in order to attain sensible results. It makes no sense, for example, to activate only the height d of a rectangular section for optimization, if the members to be calculated are subjected predominantly to buckling in the direction of the weaker axis.

If the foot note 2) appears next to a Section in mask 1.2.2, then this means that:*2)This section will not be designed as no element of this section has been selected in 1.1 !*

Footnote 1): means *1) This Section will not be designed as its section properties have not been defined!* In this case you must switch back to RSTAB and enter the required input data. The forces and moments are, by the way, not automatically recalculated automatically with the optimized sections. It is left to the user, which sections to incorporate into RSTAB for a new calculation run and when this is to be done. On account of the altered Stiffnesses, the forces and moments which were determined with the optimized section can sometimes differ considerably. In this case it's recommended, after the initial optimization, to recalculate the forces and to then optimize the sections once again. The incorporation of the sections in RSTAB can be done by means of the pull-down-menus Edit→*Export Optimized Sections to RSTAB*.



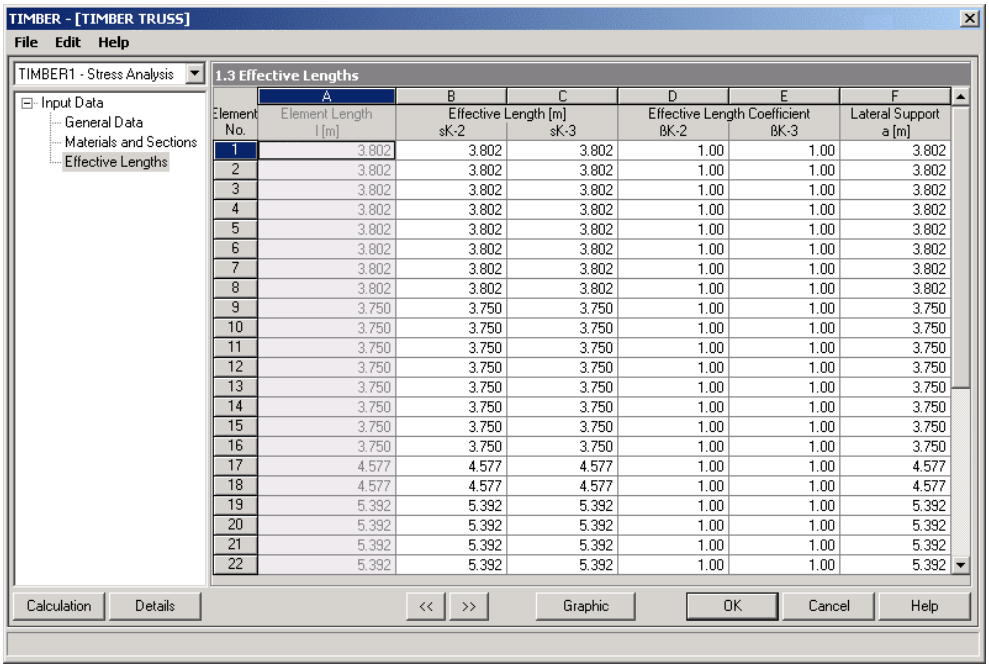
[Section-Properties] displays, as in RSTAB, the relevant statical properties of the section , while [Sections-Library] opens the RSTAB Sections' library with all its familiar selection and editing functions.

In the right-hand part of 1.2.2, the section is displayed graphically, where it can be additionally displayed inclusive of its ***Stress Points*** and corresponding ***Numbering***.

3.3.3 Mask 1.3 Effective Lengths

This mask is only available when the *Stability Analysis According to Omega Method* is checked in the *TIMBER, Details* dialog. If the *Out of Plane Buckling Possible* option is not checked in the *Details* dialog, columns B and D can be edited and buckling in both directions is possible. This is relevant if you use a 3D created framework in RSTAB for the timber design. With plane frames, the *Effective Lengths sK-3* and the *Effective Lengths Coefficient* ßK-3 cannot be edited. However, for the consideration of the governing weak axis on rafters, return to the *TIMBER, Details* dialog and check the *Out of Plane Buckling Possible* option. Then columns C and E in mask 1.3 can then be edited for plane frames.

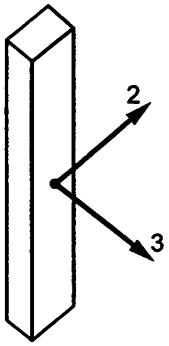
By default, the effective lengths are set in this mask to the total length of the member. Since the effective length in the timber construction can often be chosen more favorably, these values should be modified appropriately.



Mask 1.3 Effective Lengths

The *Effective Length l* is shown in column A as a control for the user. Columns B and C are for *Effective Lengths sK-2* and *sK-3* input. Columns D and E are for *Effective Length Coeffecient βK-2* and *βK-3* input. The distance between *adjacent lateral supports a* is to be entered in column F.

This lateral support is internally programmed to be oriented along the axis-2 which, with respect to a non rotated section is always taken to be in the direction of the global Y-axis. From this point of view it's important that the sections are defined such that the section is basically narrower than it is depth, and that the width b is at most equal to the depth h.



The effective length factor βK-2 and effective length sk-2 are to be applied when considering bucking about the axis 2 (buckling in the direction 3)

3.4 Calculation

Calculation

When the [Calculation] is started, TIMBER searches for the results used to analyze the load systems, LS groups and LS combinations of the positions to design. If none are found, an RSTAB calculation automatically begins and determines the necessary internal forces for the design. During the automatic computation of the internal forces in TIMBER, the programm resorts to the calculation parameter settings of RSTAB. Provided that it has been activated, the optimization of the section takes place first, before the stresses are computed for the optimized sections. Only the beginning and end positions are optimized for tapered elements. Following this, the moments of inertia are linearly interpolated at the intermediate locations. Since, however, these are expressed to the 4th power, the stress analysis becomes inaccurate with large differences between the depth at the beginning and end of the tapered element. It is then better to divide up the tapered element into individual elements whose respective beginning and end sections display minor differences in their depths. TIMBER computes the following stresses with the following specified formulae and variables.

Normal stress σ (positive = tension, negative = compression) due to axial force N and bending moments M_y and M_z:

$$\sigma_i = \frac{N}{A} + \frac{M_y}{W_{y,i}} - \frac{M_z}{W_{z,i}}$$

i = refers to the section stress points
A = Cross-sectional Area
$W_{y,i}$ $W_{z,i}$ = Elastic Section Moduli based on the following :

$$W_{y,i} = \frac{I_y}{e_{z,i}}$$

$$W_{z,i} = \frac{I_z}{e_{y,i}}$$

I_y, I_z = Moments of Inertia about the y and z axes
$e_{y,i}$, $e_{z,i}$ = Distance [eccentricity) of the i-th stress point from the center of area S in the y and z directions, respectively

Shear stress τ due to shear forces Q_y and Q_z and the torsion T:

τ_Q Shear Force component, where the following is valid for analyses using the statical moment:

$$\tau_{Q,i} = \frac{Q_y \cdot S_{z,i}}{I_z \cdot t_i} + \frac{Q_z \cdot S_{y,i}}{I_y \cdot t_i}$$

t_i governing section thickness at the i th Stress point
$S_{y,i}$ $S_{z,i}$ statical Moment about the y und z axes at the i th point

τ_T Torsional component with

$$\tau_{T,i} = \frac{T}{I_T} \cdot t_i$$ for thin walled open sections (St. Venant) and

$$\tau_{T,i} = \frac{T}{2 \cdot A_{m,i} \cdot t_i}$$ for closed tubular sections (Bredt).

I_T Torsional Constant,
$A_{m,i}$ Mean enclosed Area, to which the i th Stress point belongs

For the determination of shear stresses, it is worthy to note that the equations above can be carried out only when the sections are homogeneous. In timber construction practice we come across, however, almost only built-up sections (with the exception of rectangular sections), whose jointing results in a certain bit of give.

The following equation applies to the interactive combination of Torsion and Shear force in a single case:

$$\frac{\text{exist. } \tau_T}{\text{allow. } \tau_T} + \left(\frac{\text{exist. } \tau_Q}{\text{allow. } \tau_Q} \right)^m \leq 1$$

τ_T shear stress due to torsion
τ_Q shear stress due to shear force
m=1 for hardood (non-coniferous)
m=2 for softwood (coniferous)

The TIMBER module calculates softwood with m=2. .

Stability Analysis according to the Omega procedure:

$$\text{exist. } \lambda \leq \text{allow. } \lambda$$

The slenderness ratio λ is calculated by:

$$\text{exist. } \lambda = \frac{s_K}{i}$$

s_K effective length of the element
i radius of gyration

Allowable λ is preset in the TIMBER program under *Details* in the *1.1 General Data* mask.

The standard values are:

allow. λ = 150 for individual compression members
allow. λ = 175 for built-up and battened compression members
allow. λ = 200 for bracing and tension members, subjected to compression that occurs only due to an additional variable load (DIN 1052 - Zusatzlast)

For built-up compression members, the slenderness ratio λ of the formula above must be replaced by eff λ, which is determined in accordance with the type of construction and jointing employed. The program TIMBER can't determine the slenderness ratio eff λ for battened struts. In the program the general formula for the slenderness ratio is always employed. For this reason, the buckling check for the element type Twin beam (defined in RSTAB) is not executed correctly. In this case, it is recommended to define explicitly in RSTAB parallel beams connected by coupling battens.

$$\frac{\frac{N}{A}}{\text{allow. } \sigma_K} \leq 1$$

The allowable buckling stress is calculated as follows:

$$\text{allow. } \sigma_K = \frac{\text{allow. } \sigma_D}{\omega}$$

allow. σ_D allowable compressive stress (from the material library).

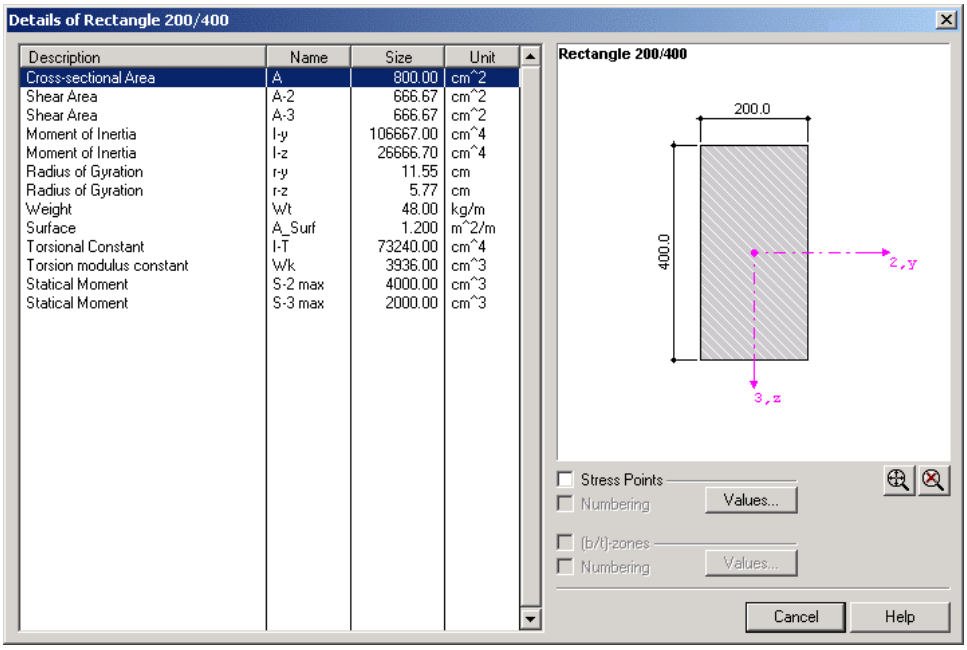
ω Buckling coefficient dependent on λ accord. to DIN 1052 part 1 Tab. 10
λ is taken to be the larger of λ_y und λ_z

3.5 Result Masks

All results are displayed in result masks. In masks 2.1 to 2.5, the *Section* and *Stresses* with the accompanying *Numbering* and *Values* can be viewed graphically.

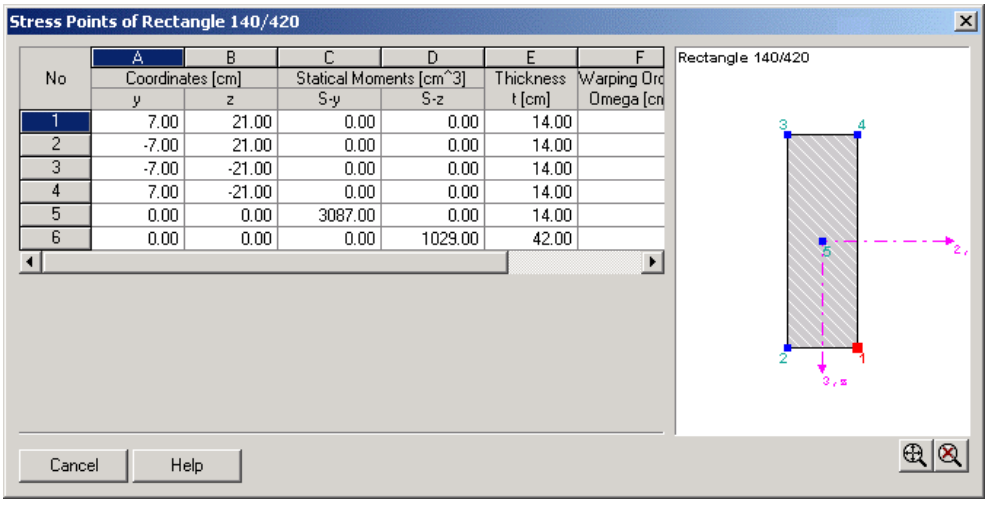
The magnifying glass buttons seen left function as in RSTAB. They can be used to zoom in, move and return to the entire view.

The [Details] button contains the *Section Properties* in detail.



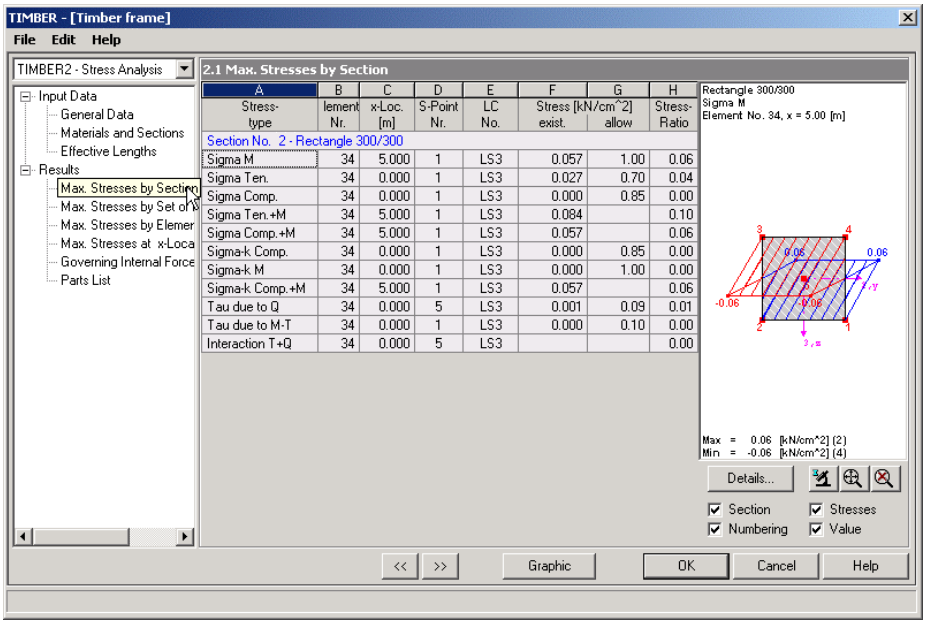
Sections Details

The [Values] button opens a mask containing the ***Statical Moments***.



Stress Points and Statical Moments

3.5.1 Mask 2.1 Max. Stresses by Section

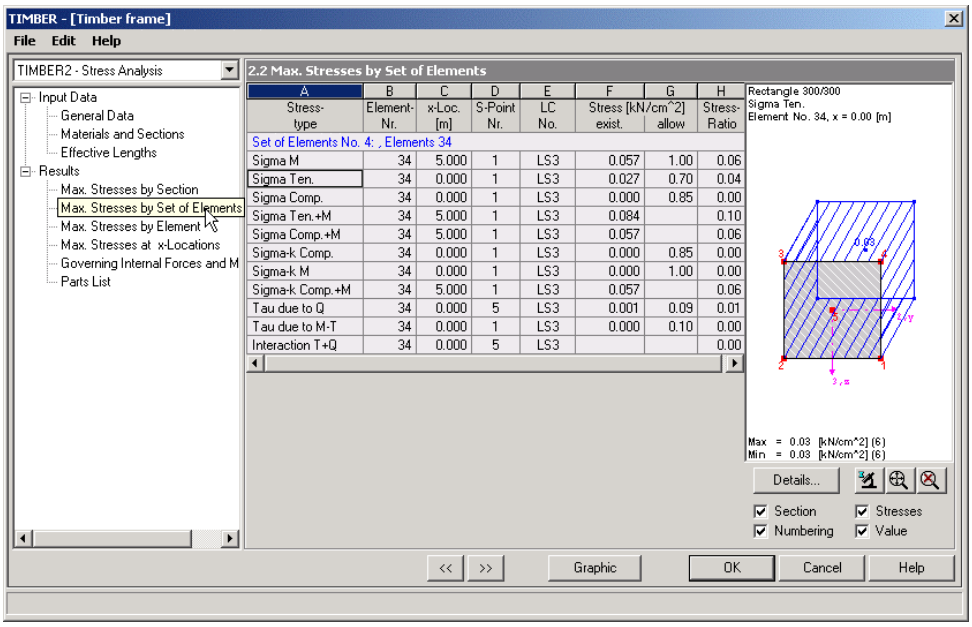


Mask 2.1 Max. Stresses by Section

The *2.1 Max. Stresses by Section* mask contains the results of the maximum stresses for all elements, LS, LG, LC selected in mask 1.1 under [Details]. The results are sorted by Section. For each *Stress Type,* the *Element N., X-Position [m], S Point N., LS No, Stress [kN/m²] Exist(ing), Stress [kN/m²] Allow(able)* and *Stress ratio* are given. The stress ratio of the section is computed by means of the preset reference stress so that one can obtain quickly an overview regarding the utilization of the section's capacity.

3.5.2 Mask 2.2 Max. Stresses by Sets of Elements

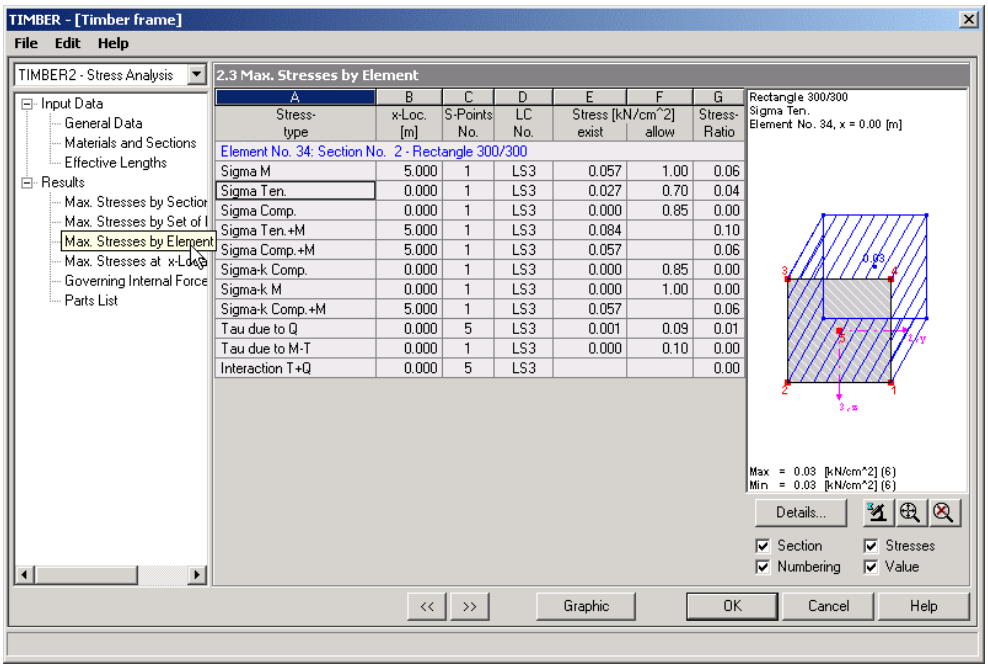
Stresses can be viewed when sorted according to the RSTAB defined sets of elements in mask 2.2. In addition, the *Set of Element No* is displayed with the data listed in mask 2.1.



Mask 2.2 Max. Stresses by Set of Elements

3.5.3 Mask 2.3 Max. Stresses by Element

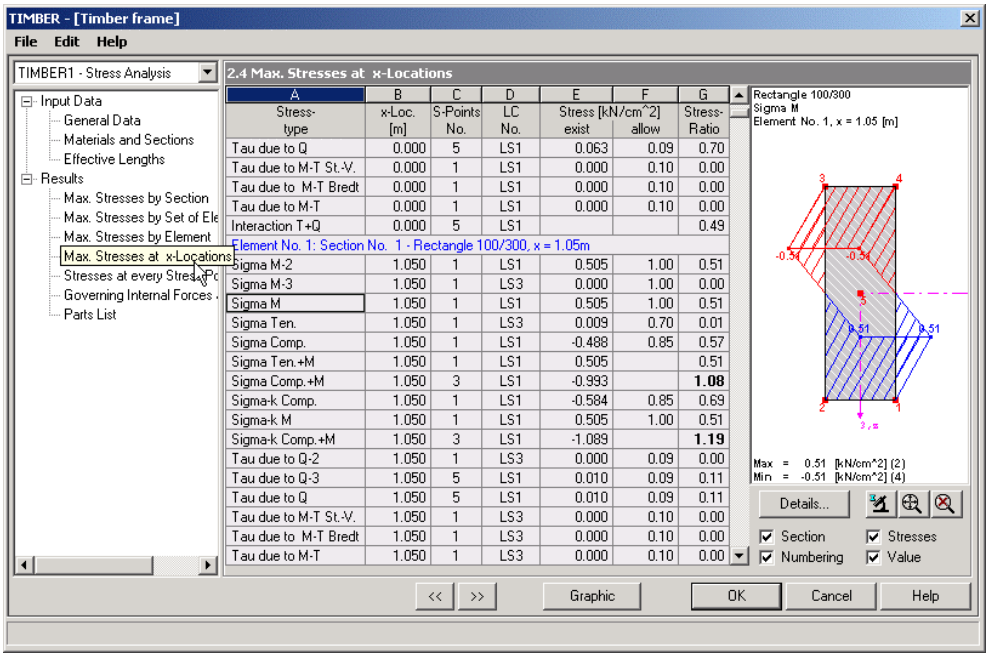
In the *2.3 Max. Stresses by Elements* mask, only the position of the element where the maximum stress occurs is given.



Mask 2.3 Max. Stresses by Element

3.5.4 Mask 2.4 Max. Stresses by X-location

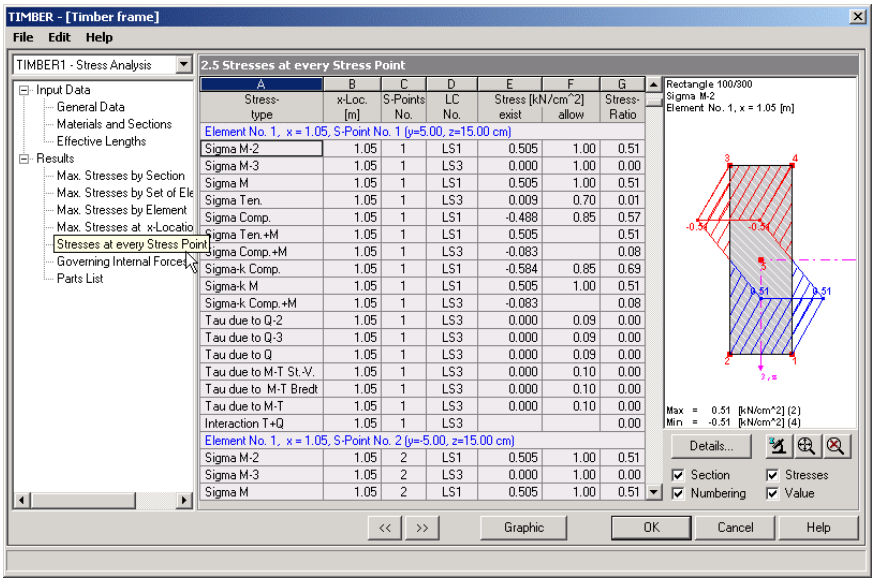
Mask ***2.4 Max Stresses by X-location*** lists all stresses in the RSTAB defined partition points. . If the corresponding maximum internal forces and moments have been computed in RSTAB, these locations will be identified additionally at the end of every element.



Mask 2.4 Max. Stresses by X-Location

3.5.5 Mask 2.5 Stresses at every Stress Point

All stresses for all defined stress points in all locations can be seen in the ***2.5 Stresses at every Stress Point*** mask. The mask will only appear if it is enabled in ***TIMBER*** from the ***Details*** function.

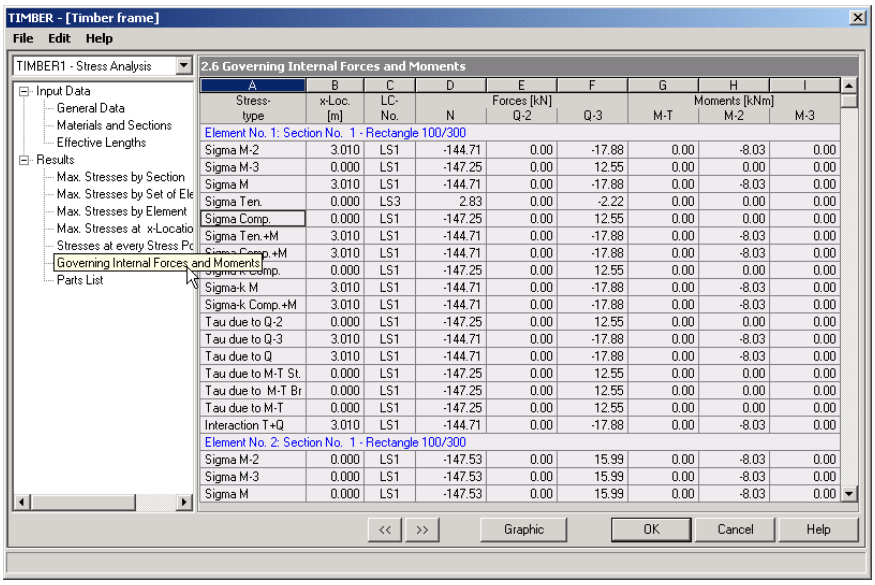


Mask 2.5 Stresses at every Stress Point

Please note that the amount of data can increase very rapidly if the stresses are computed at every point. It might be mentioned again, that the maximum stresses and therefore the governing stress points are determined automatically in the program, even if this has not been chosen separately in ***TIMBER, Details***. It is recommended to restrict the calculation of the stresses at every stress point only to a detailed examination of a particular element.

3.5.6 Mask 2.6 Governing Internal Forces and Moments

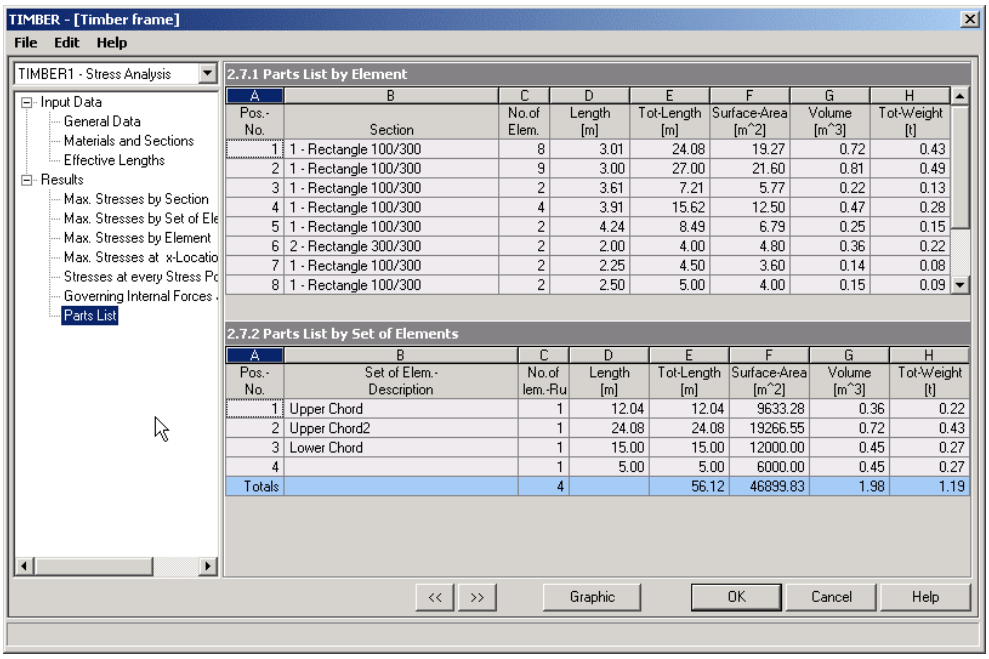
Mask ***2.6 Governing Internal Forces and Moments*** identifies for every element the governing design forces and moments, those which yield in each case the maximum tensile, compressive, shear or equivalent stress - depending on which criterion the stresses are based on.



Mask 2.6 Governing Internal Forces and Moments

3.5.7 Mask 2.7 Parts List

In the last Mask ***2.7 Parts list***, a summary of the necessary sections for the position is displayed. Provided that Element runs have been calculated, you will obtain,in additional to the mask **2.7.1 Parts List by Element** , Mask **2.7.2 Parts List by Sets of Elements**.



Mask 2.7 Parts List

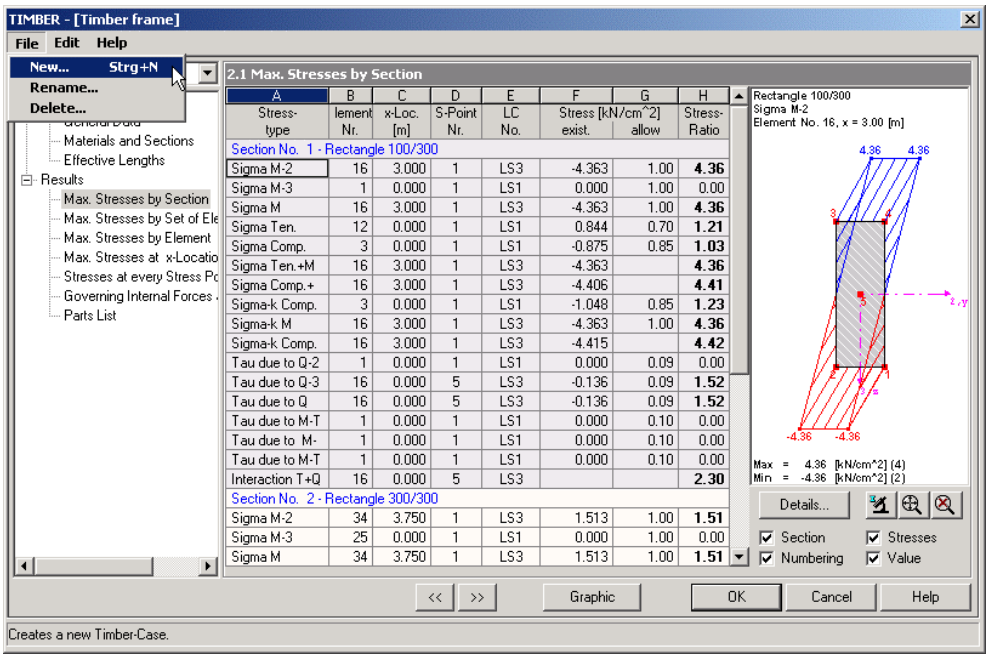
TIMBER assigns automatically in each case a *Pos.-No.* und displays in Mask 2.7.1 the corresponding
Section,
the number of elements,
the individual *element length [m]*,
the *total length* of all the elements of the Pos. *[m]*,
the *surface area* of all the elements of a *[m²],*
the *Volume* of all the elements of a position *[m3],*
and the *total Weight* of all elements of the Pos. No. *[t]*
Contained in Maske 2.7.2 are the same details as in Mask 2.7.1, should Sets of Elements have been defined.

3.6 The Menus

The menus contain all the necessary functions to handle the design cases and results. A menu can be activated by just clicking on it or by using the shortcut by pressing the [Alt] key plus the underlined letter in the menu title. For example, to use the *File* menu, just press [Alt+F] and the menu will roll down. This is applies to every menu.

3.6.1 File

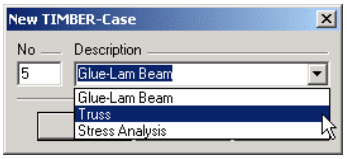
[Alt+F] ...handles the design cases.



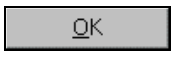
File Menu

New [Ctrl+N]

...creates a new *TIMBER design Case*.



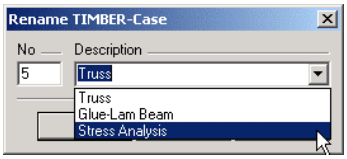
New TIMBER Case



To create a *New TIMBER Case*, you must enter a *No* and a *Description*. The [Arrow down] button lists all existing descriptions. [OK] creates the new case.

Rename

...renames the current TIMBER case by changing its *Description* or *No*.

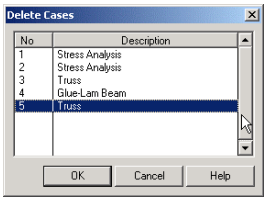


Rename TIMBER Case

If you change the number of the TIMBER case, be sure to pick one not already used by another TIMBER case.

Delete

...deletes the case selected from a list of all existing TIMBER cases.



Delete Case

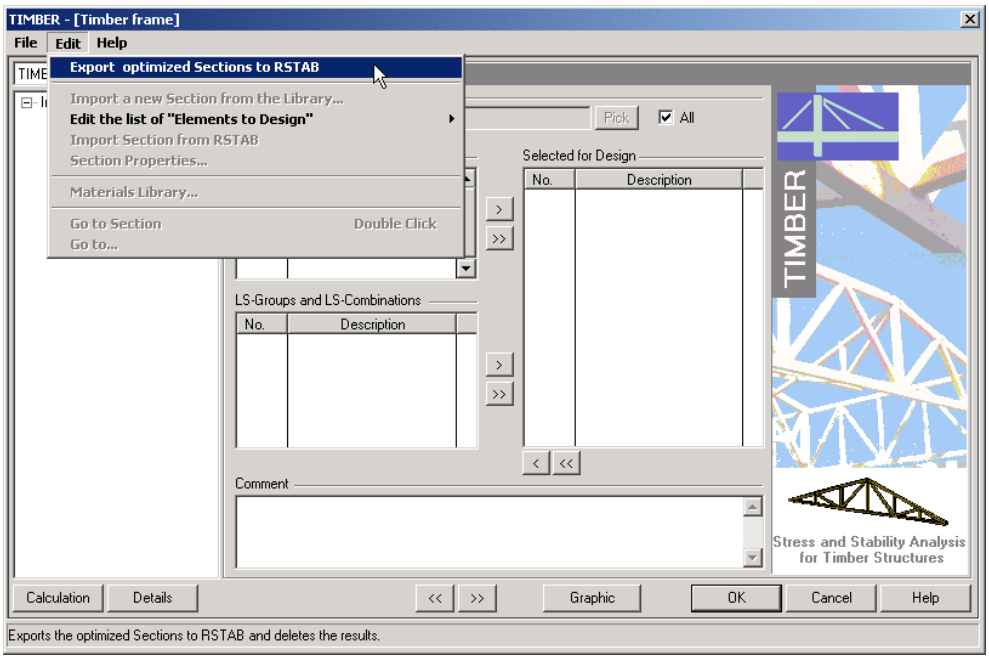
OK

Select the case that should be deleted and press [OK].

3.6.2 Edit

[Alt+E]

...offers many functions related to the handling of sections.



Edit Menu

Export Optimized Sections to RSTAB

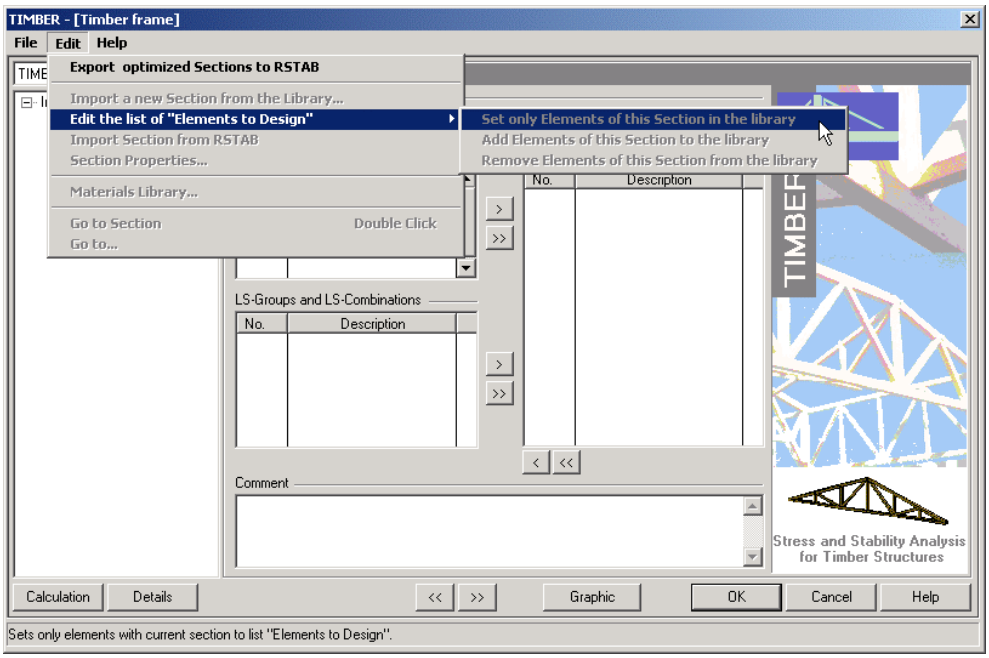
...exports edited and optimized sections to RSTAB and deletes existing RSTAB results at the same time.

Import New Section from Library

...opens the RSTAB sections library so that one of the sections can be imported to TIMBER.

Edit the list of "Elements to Design"

...offers a quick way to edit the *Elements to Design* list in mask 1.1. This sub-menu is available only when the *1.2 Materials and Sections* mask is open.



Edit→"Elements to Design" List Menu

Set Only Elements of this Section in Library

...sets only the elements of the current section in the "Elements to Design" list and deletes all others. The current section is the one where the cursor is located within mask 1.2.2.

Add Elements of this Section to Library

...adds the elements of the current section to the "Elements to Design" list. Elements from other sections already listed remain in the list.

Remove Elements of this Section from Library

...removes the elements of the current section from the "Elements to Design" list. Elements from other sections already listed remain in the list.

Import Sections from RSTAB

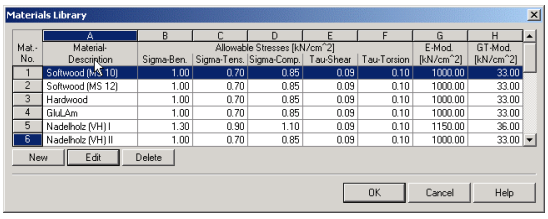
...imports the current section from RSTAB according to the entered S*ection No.*

Section Properties

...shows the details of the current section..

Materials Library

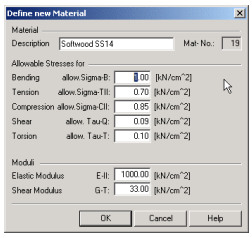
...opens a mask with a tabular index of all existing materials.



Materials Library

The *Materials* are identified by a *No.* and a *Description.* The *Allowable Stresses [kN/m²]* for *Sigma* and *Tau* are listed in columns B to F.

A new material and its resulting allowable stresses in the design case can be used by clicking on the desired row and then on [OK]. Use [New] to open a dialog where you can define a new material . The new material remains in the library for further use.



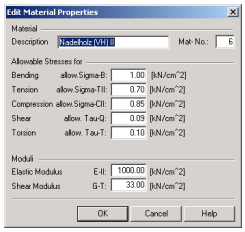
Set New Material in Library.

OK

Type in the new material name in the *Description* field. The program automatically enters the *List No.* the *Sigma* and *Tau* values are entered in their respective fields. [OK] confirms the input and closes the dialog.

The [Edit] button in the *Materials Library* opens a similar dialog after marking the material to be edited.

Edit



Edit Material Properties

All previously described input, including the *List No.* can be edited here.

Delete

The [Delete] button removes the material selected from the library. Before the data is deleted, a message asks you to confirm the deletion.

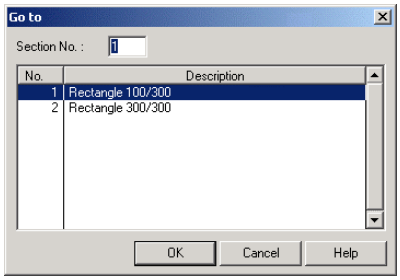
TIMBER Query to Delete

Go to Section

...goes to the current Section in mask 1.2.2.

Go To

...goes to the location entered. Depending on the mask, you can *Go To* an *Element* or a *Section.* You can also select an item from the list displayed and then [OK].



Go To

3.6.3 Help

[Alt+H]

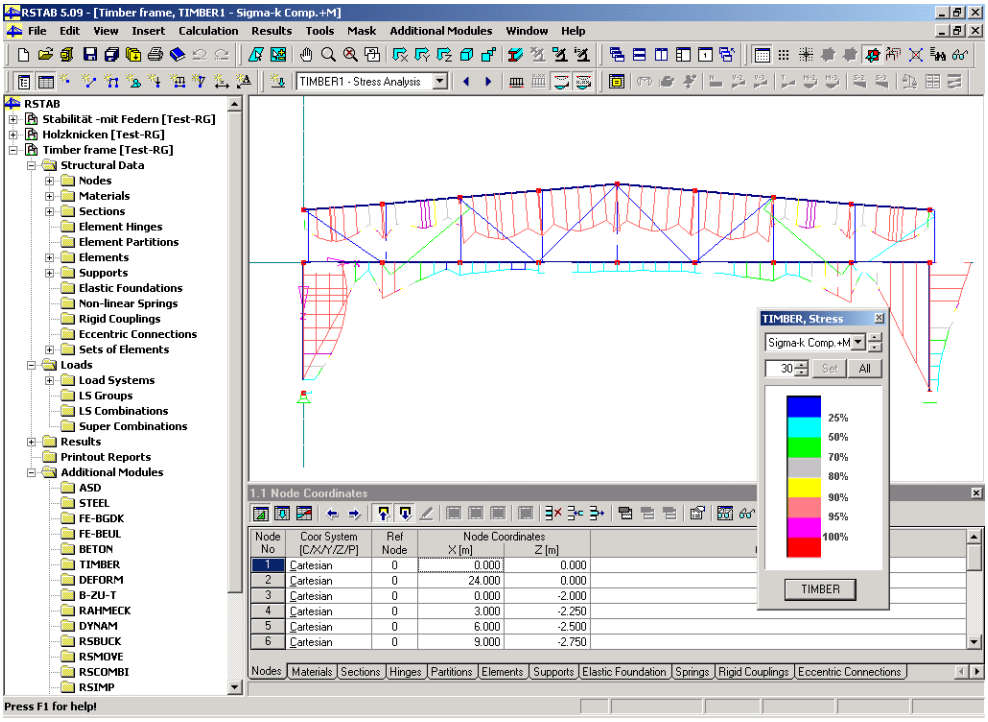
...opens the online help system.

4. Results

4.1 Graphic Display

Graphic

After the calculation, the results can be displayed by clicking on [Graphic]. The current TIMBER Case is shown.



Graphic Display

Set All

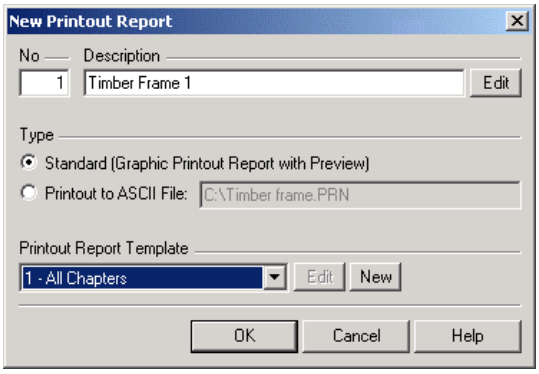
TIMBER

If the [Display Results] button was already activated when TIMBER was started, you will immediately see the stress-contour diagram and the ***Stress*** window. In the stress window, it's possible to select the stress to be displayed and the magnitude of the stress diagram with respect to the structure. With [Set] the changed settings are applied to the display. [All] applies the changed factor to all windows displaying stresses. [TIMBER] takes you back to the TIMBER masks.

The [Print] button integrates the graphic results into the printout report or prints the graphic immediately.

4.2 Printout

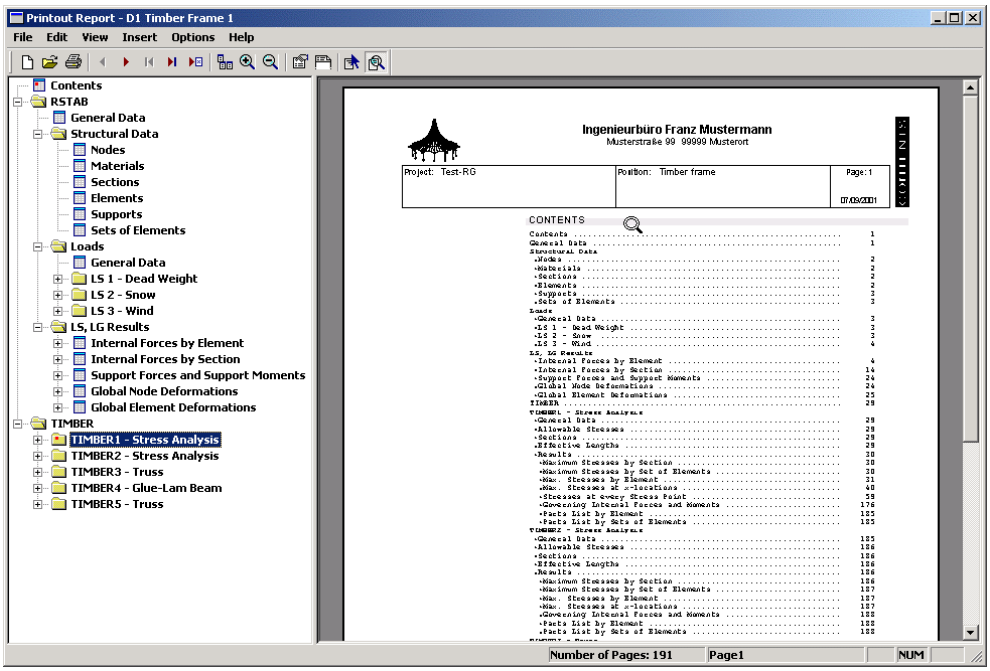
To print the numerical results of TIMBER, a [Printout Report] must first be opened or created in RSTAB.



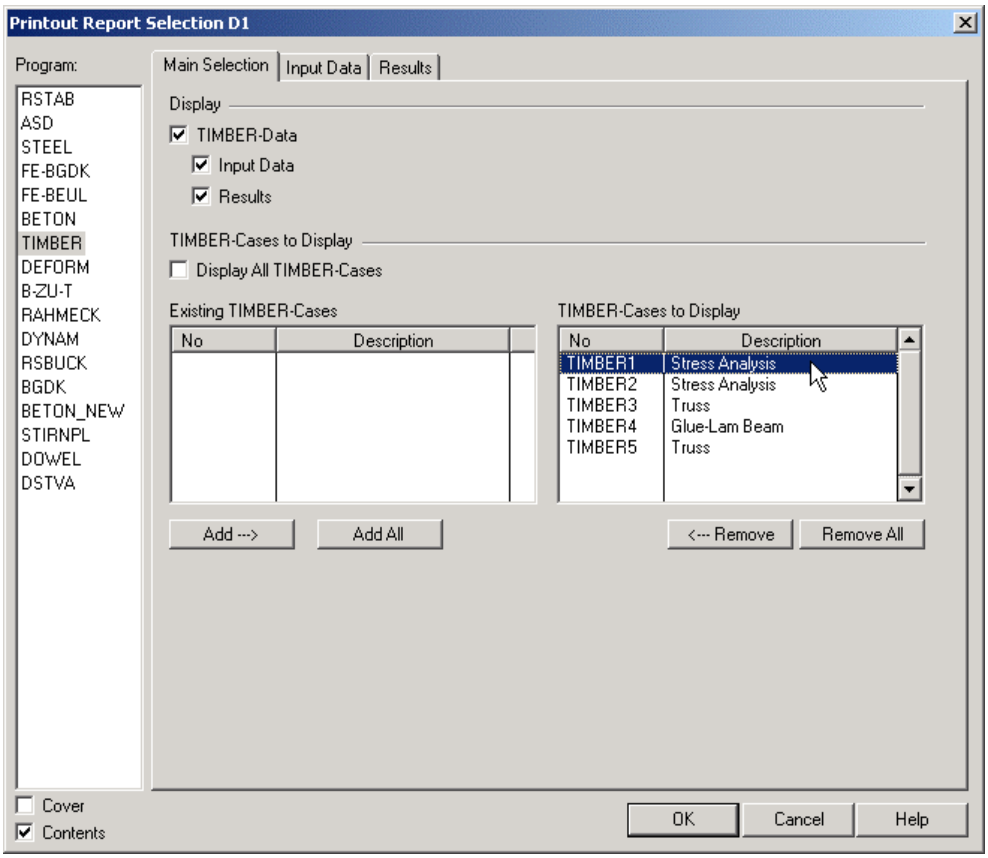
New Printout Report

Set up the printout report as described in RSTAB. It is created through the options ***Printout Report, Printout Report Template,*** and ***Create Printout Report***. The report includes the TIMBER results and is edited by clicking on [OK].

The printout report is a compilation of RSTAB, TIMBER and all other additional modules. It is made so all data can be handled in one document. Since the data may be very large, it is important to work with selections and printout report templates. This speeds up the creation and display of the printout report and protects your computer from running out of memory.



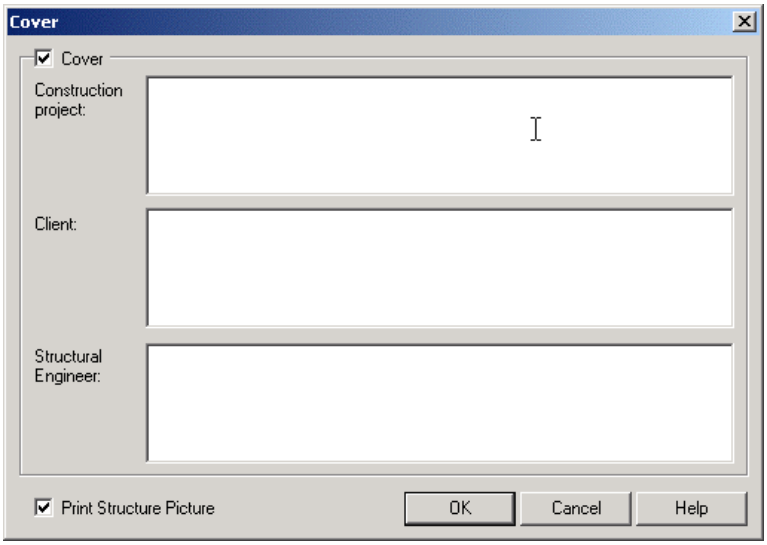
TIMBER Data and Results in the Printout Report



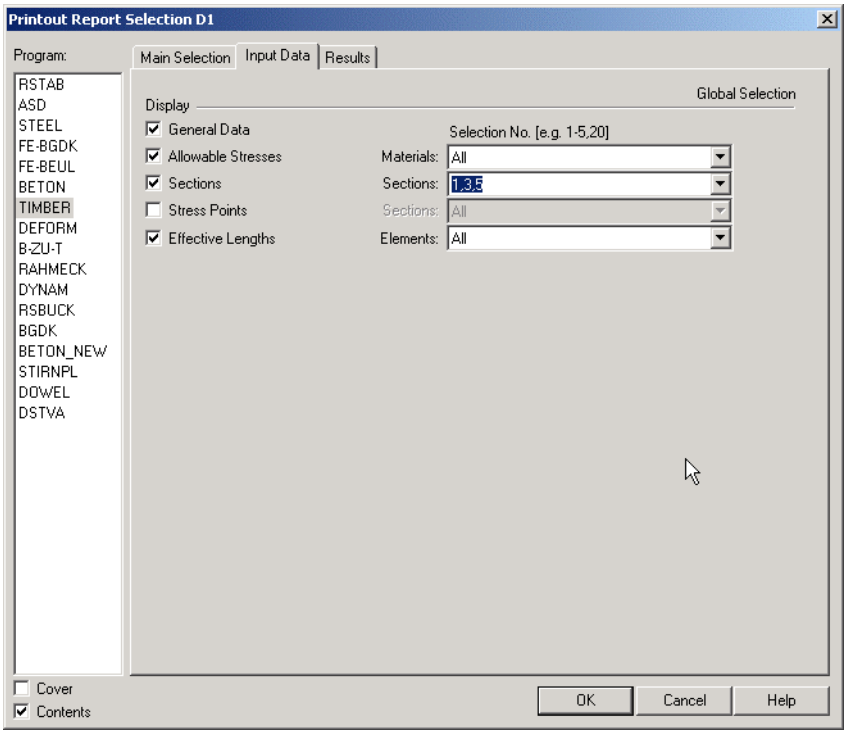
Selection TIMBER – Main Selection

In the *Main Selection* folder, decide whether to *Display TIMBER Data, Input Data* and/or *Results.* If *Display All TIMBER Cases* is not checked, particular *Existing TIMBER Cases* can be selected for *TIMBER Cases to Display.* To move cases from one list to the other, use the [Add], [Add All], [Remove] and [Remove All] buttons.

Click on *Contents* in the lower left-hand corner to include the TIMBER table of contents in the printout report. Click on *Cover* to create a cover sheet for the graphic.

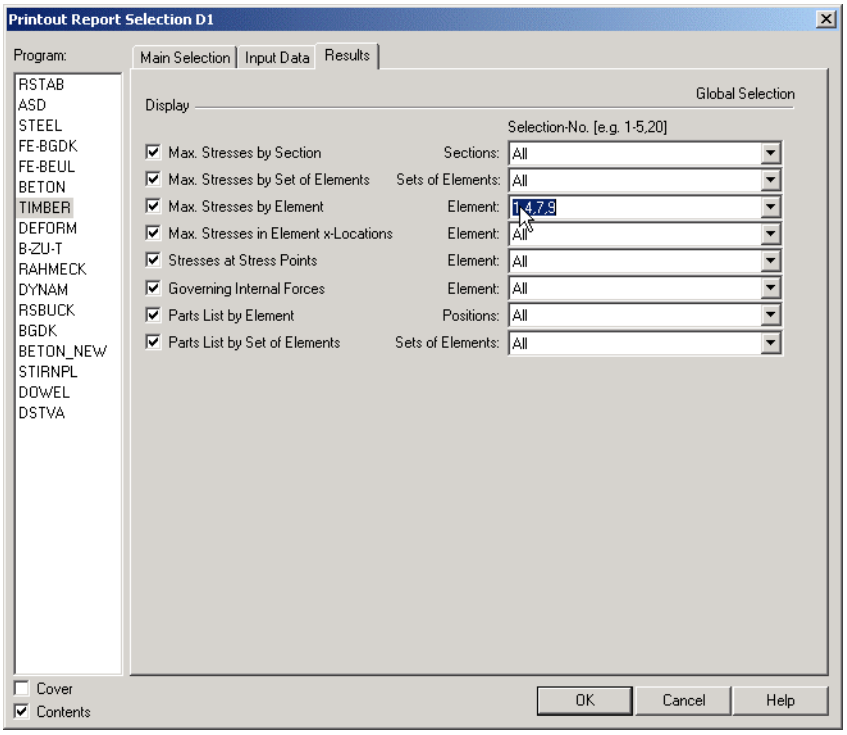


Cover



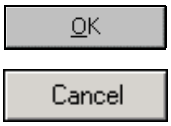
Selection TIMBER – Input Data

The *Input Data* folder contains *Display* options for *General Data, Materials, Sections* with their *Stress Points* and *Effective Lengths*. A detailed *No. Selection* is possible for *Materials, Sections* and *Effective Lengths.* Simply click on the [Arrow Down] button in each field and make a selection from the list.



Selection TIMBER - Results

Within the *Results* folder, select which results to *Display* and whether *All* or particular numerical *No. Selections* will be included.



In every folder, [OK] confirms the settings and closes the window. The printout report is automatically updated when editing is confirmed. [Cancel] closes the dialog without editing included.