WOODexpress™

Design of timber structures with Eurocode 5

Copyright © RUNET

USER's Manual

Table of contents

Updated 09/09/2011

License and Copyright

If you do not agree with the terms of the following Disclaimer and License Agreement, return the program disks before you install and activate it, to RUNET Norway as, within 30 days of purchase for a full refund of software cost and sales tax.

Disclaimer

This software should be used only from experienced and licensed professional engineers. The software must be considered as a helping tool for the designer engineer, and can never replace the knowledge, the experience and the judgment of a professional engineer. The user of this software must understand that no matter how advanced and well checked this software is, he should carefully check the results and take responsibility of their use.

Copyright

This software is owned by RUNET Norway as, and it is protected by EC (European Community) Copyright Laws and International Treaty Provisions. This software and the accompanying materials, must be treated like any other copyrighted material (e.g. book). It is allowed although to make one copy of the Software for backup or archive purposes. You may not copy and distribute the accompanying materials. It is strictly prohibited by law unauthorized reproduction or resale of this software product and the accompanying materials.

Software License

This is a legal agreement between the legal user of this software and RUNET Norway as. By installing this software you agree to be bound by the terms of this agreement. If you do not agree to the terms of this agreement then do not install this software and return within 30 days after purchase, for a fully refund of your payment.

Scope of License

Each licensed copy of WOODexpress, must be used either on a single computer, or installed on a single workstation used non simultaneously by multiple people, but not both. This is not a concurrent use license.

You may not rent or lease this software. You may not modify, adapt, translate, reverse engineer, decompose, or disassemble the software. Any violation of this agreement terminates your right to use this software.

Liability Limitations

WOODexpress, in no event shall be liable for any damages whatsoever (including without limitations, damages for loss of business profits, business interruption, or any other loss) arising of the use of this software. RUNET makes no warranties, either expressed or implied, as to the quality or performance of this software, that the results and calculations of this software will meet your requirements, or that the operation of this software will be error free.

This software is a helping tool to aid you in the design of timber structures. The results of this software must be reviewed and interpreted from experienced licensed engineers, and by no means constitute an acceptable engineering design.

WOODexpress and related documentation are provided "AS IS" and without warranties as to performance or merchantability or any other warranties whether expressed or implied. Because of the various hardware and software environment into which this software may be put, no warranty of fitness for a particular purpose is offered.

Under no circumstances shall RUNET Norway as and its personal be liable for any direct or indirect, incidental special or consequential damages resulting from the use or inability to use of this software or related documentation, even if RUNET Norway as has been advised of the possibility of such damages.

This agreement shall be governed by EC (European Community) laws. If for any reason a court or competent jurisdiction finds any provision of this agreement, or portion thereof, to be unenforceable, that provision of the agreement shall be enforced to the maximum extend permissible so as to effect the intent of the parties, and the remainder of this agreement shall continue in full force effect.

If this license is too restrictive with the laws of your country, do not use this software and return within 30 days after purchase, for a fully refund of your payment.

1   General

WOODexpress is a program for computations and dimensioning of timber components, and timber roofs according to Eurocode 5 (EC5). In a unified environment you can design in a simple way elements of timber structures, covering most of the needs of a structural design firm. It simplifies all the repetitive and time-consuming every day calculations for timber elements and timber roofs.

In a graphical added environment you specify the necessary dimensions, loads, design code parameters, of timber components, and the design is immediately performed. Default values and checks for erroneous input values, facilitate the input data process.

The report, which is created simultaneously, shows in detail all the calculations and the design steps with references to the corresponding design code paragraphs. It also warns you in case of inadequate design.

In one project you can create as many timber components (objects) as you desire, all the data are stored automatically in one file. A dedicated window helps you working with the design objects in a project. You can edit, delete, copy, include in report, design objects with a click of the mouse. Each design object is well marked with a name and an icon. You can select the objects to be included in the final project report.

You can view and print a detailed drawing of the roof truss in any scale, with the programs CAD modulus.

The report includes sketches, graphs and formulas, and with user specified title block, logos and fonts.

The material properties, the loads and the design code parameters can be adjusted by the user according to the requirements of the National application document or the country standards.

A context-sensitive Help system, guides you through the use of the program and the Eurocode provisions. On-line user's manual and frequently asked questions (F.A.Q.) are included in the program.

The **structure (timber) components** you can design are:
- **cross sections** under combined action in ultimate limit state
- **structural fire design** for all cases of ultimate limit state
- computation of **bolted, nailed and screwed joint** capacity of various types
- carpentry joints
- **floors**, **floor beams and composite concrete-timber floors.**
- a variety of **roofs** as
  - mono-pitch roofs
  - coupled roofs
  - close coupled roofs
  - collar roofs
  - trussed rafter roofs
  - attic roofs
  - trusses with carpentry joints

In addition various **engineering tools** are included as, unit conversion, section properties, area computations, reinforcing bar properties, lateral earth pressure coefficients.

From the **parameters** menu you can adjust the default dimensions and code parameters, according to the needs of your region and the Eurocode National Application Document of your country.

From this edition (01.11.2010) of User manual, the references to the pre version of Eurocodes, ENV, will be removed.

2   After program installation

The program is based on the structural Eurocodes. The application and the parameters of Eurocodes may be different from country to country. National Application Documents define the parameters, the supporting standards and provide national guidance on the application of Eurocodes.

After the installation of the program various parameters as material constants, safety factors, material properties, loads, must be adjusted according to the requirements of the region the program is used.

The appearance of the report is quite advanced, with user defined graphic and logo text, page margins, font selection, size of indentation etc.. The report appearance can also be adjusted to fit the requirements of the program user.

From the Parameters :

- Adjust the characteristic **material properties**, and loads according to the requirements of your region or country. For this it is advisable to consult the National Application Document of the Eurocodes 5 and 1.
- Select **default timber material**.
- Select **default loads for roof coverage, snow and wind loading**.
- Set the **default code values for material and load factors**. These are the action coefficients, the load combination coefficients and the material factors as they are defined in Eurocode 1 and Eurocode 5 or the National Application Document.
- Select the group of section sizes for truss members and purlins.
- You may want to define your own set of section sizes for truss members and purlins.
- From Eurocode and national Annexes, select the use of EN standards for timber design and loads.
- In [Parameters and Assumptions] you specify various design parameters and default values.
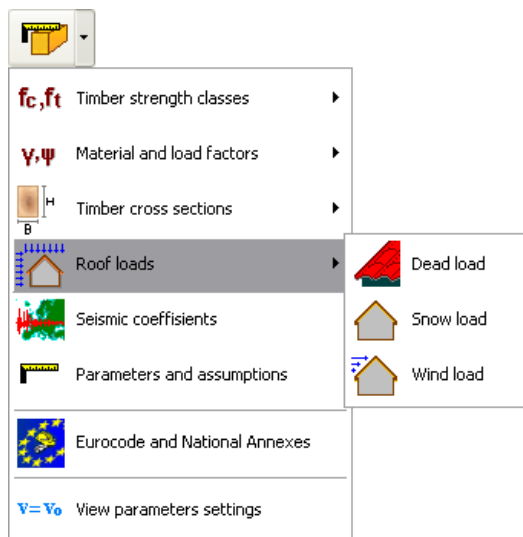
From the Report Setup

- Adjust the report appearance (margins, font, cover, logo, page caption, page footnote).

- From menu **Setup/Decimal point**, select type of decimal point symbol.

- Check the right appearance of Greek mathematical symbols in the report, and in case you do not get right appearance, activate Greek character support from the menu **Setup/Greek character Setup.**
- From menu **Help/Program user's manual**, you read or print the program user's manual.

3   Basic philosophy in program use

With WOODexpress you can design and manipulate various design objects, which are covering various parts of a timber structure. In a project you may create as many design objects, as you want. All the data are saved in a common project file. A common report is created for the timber objects, which are checked to participate in the report.

New timber objects can be created with the design buttons at the top. The design objects, with their name and a characteristic icon are shown in the [Design objects] window. From this window you regulate their appearance and the order of appearance in the reports. On the right side of the window shows the details of calculations of the selected design object.

By double clicking at a design object you enter its calculation window where you specify the dimensions loads and design code parameters. When the object is created all the values take the default values set from the parameters. All the data required are marked with a sketch, and the appropriate dimensions. Automatically a check is always made for wrong or erroneous input values.

In front of every design object is a check box. The objects that are checked will be included in the common report. If more than one roof trusses are checked for report, because the report for roof truss designs are very long, a warning message will show up, to inform you in order to reduce the included in the report design objects.

By right clicking on a design object you may choose from the popup menu to perform calculations, CAD drawing, report previewing or printing for the design object.

The basic steps in using the program are:

Open a Project File from menu [File].

- Select a design object, from the [Design objects] window, or create a new one from the design buttons on the top.
- Activate the computations of the object, by double clicking on the design object or by clicking the computations button. In a new object the computations are activated automatically.
- In the object's calculation window enter the necessary data for the particular design object and click on [Computations]. Click at the drawing button to enter the CAD drawing modulus for a roof truss.
- Mark the objects you want to appear in the report, and adjust their order of appearance in the [Design objects] window.
- Preview and Print the report.
- From the menu Parameters, you can specify the design and code parameters, and the default values.
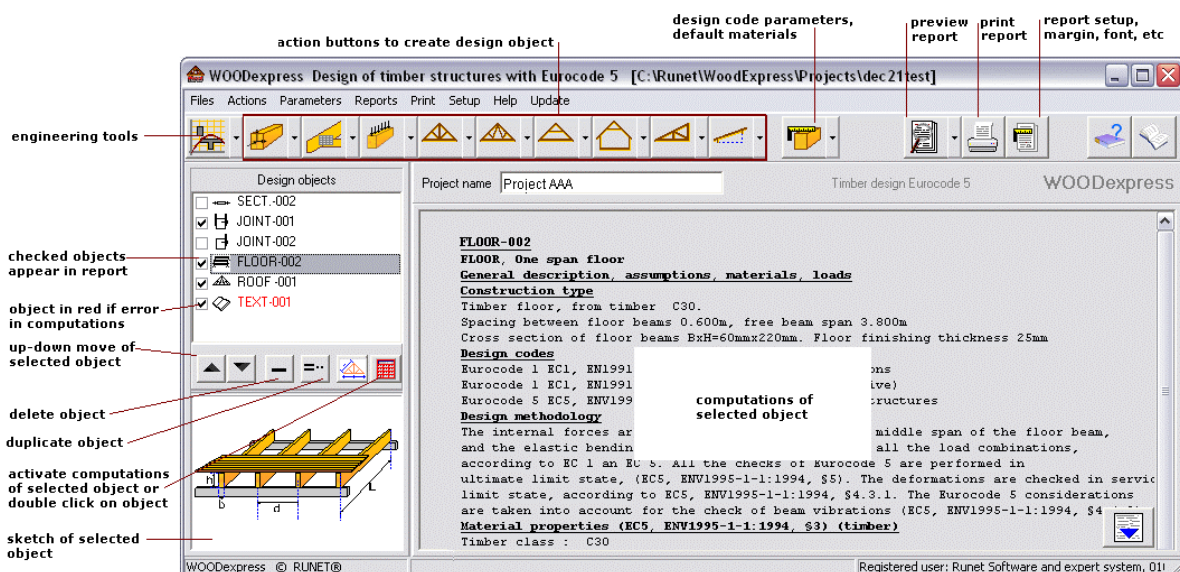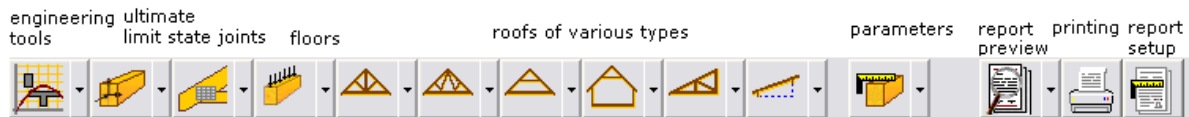
4   Design

The basic program actions are:

- create and manipulate design objects
- set design parameters
- preview reports
- print reports
- manipulate, preview and print the truss drawings

These actions are performed from the program menu or with the drop-down menu buttons (design buttons) at the top of the main program window. The [Design objects] window, is where all the design objects of the project appear with their names and their characteristic icons.

After creating a design object, you can activate and manage the object from the [Design objects] window.

5   Design objects

With the program you create and manipulate various design objects (or calculation objects). The design objects can be a variety of timber parts of a structure, timber connections and mainly timber roofs.

We will refer to these calculations as design objects, timber objects or structural objects.

You create the design objects with the design buttons on the top. In a project you may create as many design objects as you want. Automatically the program gives a default name to each object, (which you may change), and assigns a small characteristic icon in front to recognize the type of the design object.

The design objects are autonomous and each one has its own drawings, material properties and computations. All the design objects of the project are listed in the window at the left, which is the basic window in working with the design objects. By selecting (clicking at) an object, the corresponding computations appear on the right window. If the object appears in red colour, the computations have errors or are not satisfying the design code requirements. The sketch of the selected design object appears underneath.

The objects checked in front, ☑ are included in the report. A common report is produced from the selected objects. In the Report Setup you may specify the report of each design object to start in a new page.

With the button you activate the object design window, and with the button you activate the objects drawing window.

By right click on a design object you can choose to activate computation, or to see the report of the object. If the object is a roof truss you can enter the drawing of the truss.

The order of the objects (which is also the order of appearance in the report), is regulated with the two buttons .
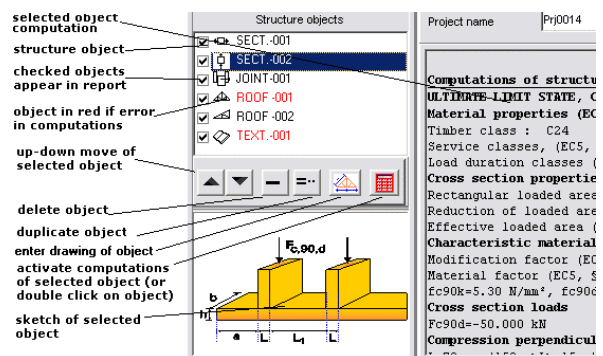
You can delete one or more selected design objects by clicking at Del key or , (multiple selection of design objects with [Shift] and mouse click, or [Ctrl] and mouse click).

You can duplicate a selected object by clicking at .

6   Calculation window

A calculation window has a typical sketch of the timber object that is to be designed. All the necessary input data are marked with their dimensions. By clicking [🖩 Computations] all the calculations are performed.
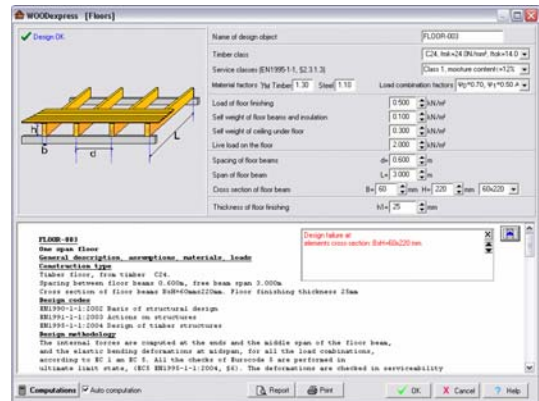
When the Auto-computation is checked, the calculations are performed automatically when you change the data.

The Auto-computation mode can be set from [Setup/Auto computational option]. Depending on the speed of the computer the user can choose to have the computations performed simultaneously with the data input/change than when clicking the Computations button. In this case check the Auto Computation. [🖩 Computations ☑ Auto computation]

The calculations appear in the window underneath. This window is expanded by clicking [🔲]. Warnings and errors for inadequate design values are shown in red in a small textbox and in the calculations. You can view a CAD drawing of the timber roof or a sketch of the timber component by clicking [📐 Drawing].

When the object is created all the parameters take default values. A check is always made for wrong or erroneous input values. After the computations, OK or Error (in red) message is shown on top left.

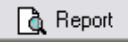You can see a report of this design object by clicking at [🔍 Report] or print the report with [🖨 Print].

When the Auto-computation is checked, the calculations are performed automatically when you change the data. The Auto-computation mode can be set from [Setup/Auto computational option].

7   Files

All the design (timber) objects are saved in the same unique file with an **extension [WoodExpressData]**. When you specify a new file name you don't have to type in the extension.

The data are saved automatically as you change them and do computations.

A folder with name [WOODexpress\Projects] is initially created to store the files of the various projects.

At the bottom of the [Files] menu is a list of the three most recent used project files.
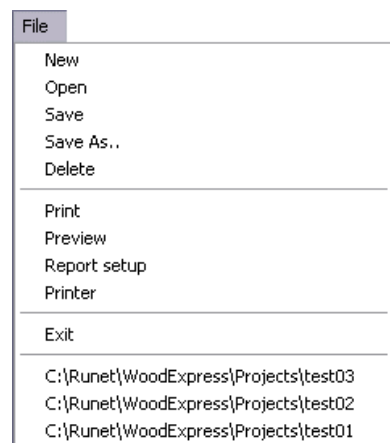
7.1   File commands

     New   Open Save   Save As   Delete

Printing commands

     Print   Preview   Report Setup Printer

Program settings

     Greek character support Language Setup   Decimal point symbol   Default main screen size
     Computational Option

8   Step by step, program use
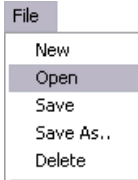
Open a Project File. Use New for new project and Open for an existing project file. All the data are saved in the same file. The data are saved automatically.

Create a new Design object, e.g. Roof type A, From the drop-down buttons on the top, automatically you enter the computation window for this object.

You may select an existing design object, from the [Design objects] window, and activate the computations by double clicking at the object, e.g. Roof-003, or by clicking at ▦ .

In the window with the computations, enter the necessary data for the particular design object and click on **Computations** .

When the Auto-computation is checked, the calculations are performed automatically when you change the data.

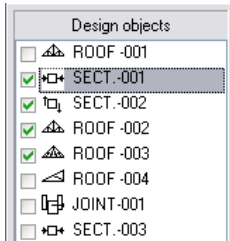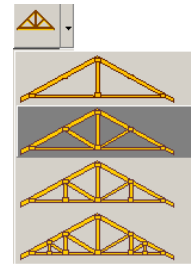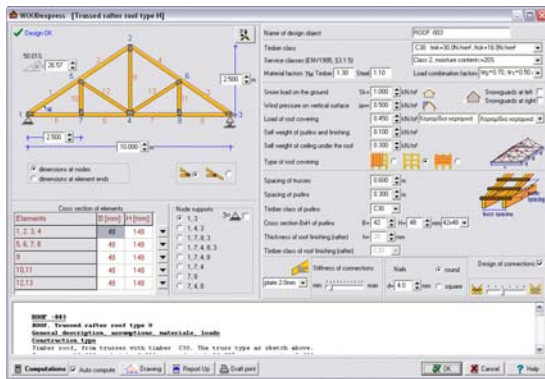Click to see more of calculations.

Computations   All the computations for the design object are performed.

✔ Design OK   A message appears if design is OK, the computations and the dimensions are adequate.

✘ Error, Inadequate design   If the design has problems due to inadequate dimensions this message will appear.

Drawing   Automatic generation of CAD drawings.

Report   Preview report.

Print   Print report.
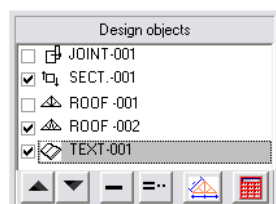
Select (check) the objects you want to include in the report. With the arrows you can adjust their order of appearance in the report. In the report only the objects checked in front will appear.

Report setup. Adjust the appearance of the report. You can adjust: font size, margins, captions and footnotes, line distances, character font, new page after each object printout, line thickness and paragraph indentation

9   Parameters

The parameters must be set for the region or country the program is used and according to Eurocode 0,1,5 and National Application Documents.

From the parameters you set:

- the default timber strength classes
- the default material and load factors, the action and load combination coefficients
- default sizes of timber cross sections
- the loads of roof covering, the snow load and the wind load
- seismic coefficients
- which Eurocode version to be used for the calculations
- different parameters and assumptions
- Show settings of parameters

9.1   Material and load factors

All the material, load factors and action coefficients have to be adjusted according to the Nation Application Document in every country. The values given below are recommended values in Eurocode 0 and 5.

The **action coefficients** for permanent and variable loads in unfavourable loading conditions, γG and γQ, which according to Eurocode 0 (EN 1990:2002) are $\gamma G=1.35$, $\gamma Q=1.50$.

In order to edit the values of the table you must first unlock by clicking at [Locked].
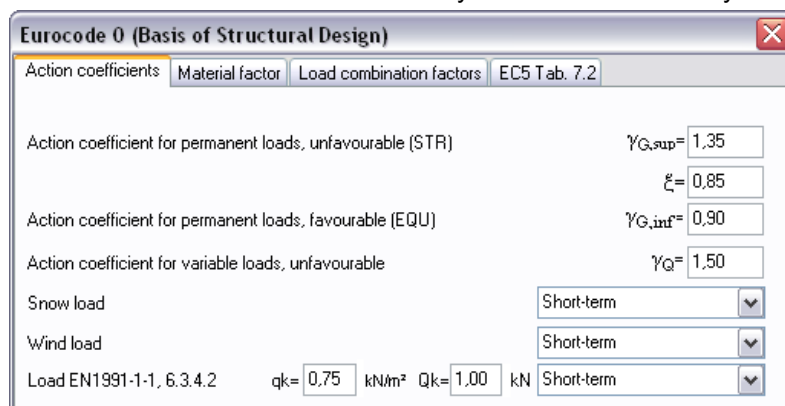
The **material factors** for timber based materials γM, which according to Eurocode 5 are 1.30 for timber based materials, 1.10 for steel in joints, and 1.00 for accidental situations and serviceability limit state.
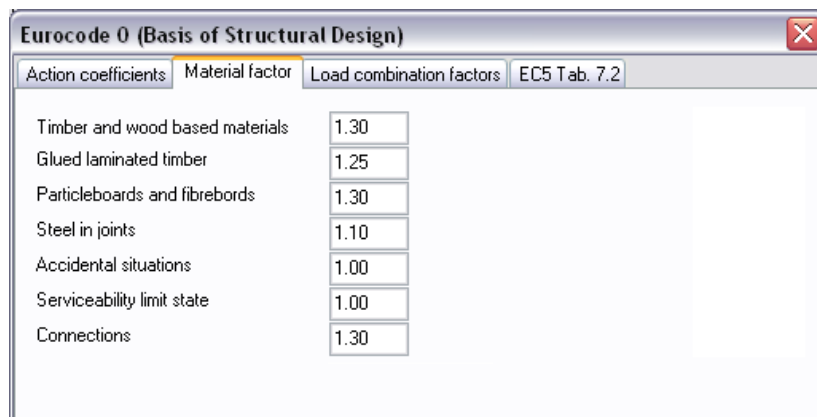
The **load combination factors** $\psi 0, \psi 1, \psi 2$, which are given in Eurocode 0 (EC 0) AnnexA1,A1.2.2 Table A1.1.

Action	ψ0	ψ1	ψ2
Category A (general)	0.70	0.50	0.30
Category B (offices)	0.70	0.50	0.30
Category C (common areas)	0.70	0.70	0.60
Category D (shops)	0.70	0.70	0.60
Category E (storage areas)	1.00	0.90	0.80
Category F	0.70	0.70	0.60
Category G	0.70	0.50	0.30

(Eurocode 0 (Basis of Structural Design) — tabs: Action coefficients | Material factor | Load combination factors | EC5 Tab. 7.2)

Recommended values of ψ factors for buildings

Action	ψ_0	ψ_1	ψ_2
Imposed loads in buildings, category (see EN 1991-1-1)			
Category A : domestic, residential areas	0,7	0,5	0,3
Category B : office areas	0,7	0,5	0,3
Category C : congregation areas	0,7	0,7	0,6
Category D : shopping areas	0,7	0,7	0,6
Category E : storage areas	1,0	0,9	0,8
Category F : traffic area, vehicle weight ≤ 30kN	0,7	0,7	0,6
Category G : traffic area, 30kN < vehicle weight ≤ 160kN	0,7	0,5	0,3
Category H : roofs	0	0	0
Snow loads on buildings (see EN 1991-1-3)*			
Finland, Iceland, Norway, Sweden	0,70	0,50	0,20
Remainder of CEN Member States, for sites located at altitude H > 1000 m a.s.l.	0,70	0,50	0,20
Remainder of CEN Member States, for sites located at altitude H ≤ 1000 m a.s.l.	0,50	0,20	0
Wind loads on buildings (see EN 1991-1-4)	0,6	0,2	0
Temperature (non-fire) in buildings (see EN 1991-1-5)	0,6	0,5	0
NOTE The ψ values may be set by the National annex. * For countries not mentioned below, see relevant local conditions.			

Serviceability limit state
In the program in menu [Parameters/Parameters and assumptions] you can define the values you want. Default values (L/300, L/250 and L/150). The limits for cantilever beams are taken as half of the above.

Eurocode EN 1995-1-1:2004/A1:2008, Table 7.2.

Eurocode 0 (Basis of Structural Design) — tabs: Action coefficients | Material factor | Load combination factors | EC5 Tab. 7.2

Winst=L/f1 (f1=300~500) f1= 300.00

Wnet,fin=L/f2 (f2=250~350) f2= 250.00

Wfin=L/f2 (f2=150~300) f2= 150.00

9.2   Service classes

In Eurocode 5 the service classes are defined from the mean moister content of the timber. In most cases National Application Documents, define this classification.

According to Eurocode 5

Class 1 In this class the mean moisture content of coniferous timber is below 12%

Class 2 In this class the mean moisture content of coniferous timber is below 20%

Class 3 Higher moisture content.

9.3   Timber strength classes

The timber strength properties can be changed after you unlock the data by clicking at [Change values]. The timber properties must be adjusted to the data of the region or country the program is used, and according to the National Application Document of Eurocode 5. By clicking at a line of the table you select the default timber class in the program.

The classification of timber in various strength classes as it is implemented in the program, are given in EN338 :2009, and EN 1194:2000, Structural timber-Strength classes.

|	Class	fmk (MPa)	ft0k (MPa)	ft90k (MPa)	fc0k (MPa)	fc90k (MPa)	fvk (MPa)	E0m (MPa)	E05 (MPa)	E90m (MPa)	Gm (MPa)	po (Kg/m³)
Poplar an softwood spieces	C14	14.00	8.00	0.40	16.00	2.00	1.70	7000	4700	230	440	290
	C16	16.00	10.00	0.50	17.00	2.20	1.80	8000	5400	270	500	310
	C18	18.00	11.00	0.50	18.00	2.20	2.00	9000	6000	300	560	320
	C20	20.00	12.00	0.50	19.00	2.30	2.20	9500	6400	320	590	330
	C22	22.00	13.00	0.50	20.00	2.40	2.40	10000	6700	330	630	340
	C24	24.00	14.00	0.50	21.00	2.50	2.50	11000	7400	370	690	350
	C27	27.00	16.00	0.60	22.00	2.60	2.80	11500	7700	380	720	370
	C30	30.00	18.00	0.60	23.00	2.70	3.00	12000	8000	400	750	380
	C35	35.00	21.00	0.60	25.00	2.80	3.40	13000	8700	430	810	400
	C40	40.00	24.00	0.60	26.00	2.90	3.80	14000	9400	470	880	420
	C45	45.00	27.00	0.60	27.00	3.10	3.80	15000	10000	500	940	440
	C50	50.00	30.00	0.60	29.00	3.20	3.80	16000	10700	530	1000	460
Hardwood spieces	D30	30.00	18.00	0.60	23.00	8.00	3.00	10000	8000	640	600	530
	D35	35.00	21.00	0.60	25.00	8.40	3.40	10000	8700	690	650	560
	D40	40.00	24.00	0.60	26.00	8.80	3.80	11000	9400	750	700	590
	D50	50.00	30.00	0.60	29.00	9.70	4.60	14000	11800	930	880	650
	D60	60.00	36.00	0.60	32.00	10.50	5.30	17000	14300	1130	1060	700
	D70	70.00	42.00	0.60	34.00	13.50	6.00	20000	16800	1330	1250	900
Glulam	GL24h	24.00	16.50	0.40	24.00	2.70	2.70	11600	9400	390	720	380
	GL28h	28.00	19.50	0.45	26.50	3.00	3.20	12600	10200	420	780	410
	GL32h	32.00	22.50	0.50	29.00	3.30	3.80	13700	11100	460	850	430
	GL36h	36.00	26.00	0.60	31.00	3.60	4.30	14700	11900	490	910	450
	GL24c	24.00	14.00	0.35	21.00	2.40	2.20	11600	9400	320	590	350
	GL28c	28.00	16.50	0.40	24.00	2.70	2.70	12600	10200	390	720	380
	GL32c	32.00	19.50	0.45	26.50	3.00	3.20	13700	11100	420	7850	410
	GL36c	36.00	22.50	0.50	29.00	3.30	3.80	14700	11900	460	850	430

9.4   Timber section sizes

From the menu Parameters you can choose group of cross sections for truss members, with cross section dimensions for the elements of the roof truss. The elements of the same cross section are in groups. The cross section is rectangular with width B and height H. The dimensions of the cross section are in [mm].
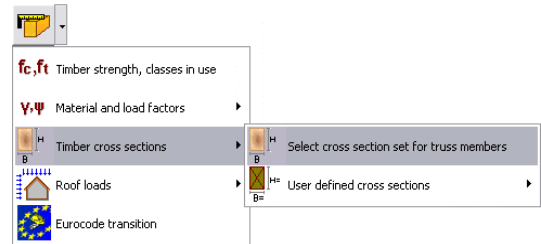
You can choose a set of cross section values according to your region. The most common values for the different countries in Europe are implemented in the program according to EN1313-1. You can also define two groups of user defined dimensions for truss members and purlins.

From the menu Parameters choose Timber cross sections/ User defined cross sections / Cross section for truss members-1. Enter the values for B and H in the empty white fields and your user-defined table is ready. You can change the values at any time. Remember to choose the table you want to use as default. From the menu Parameters choose Timber cross sections/ Select cross section set for truss members.
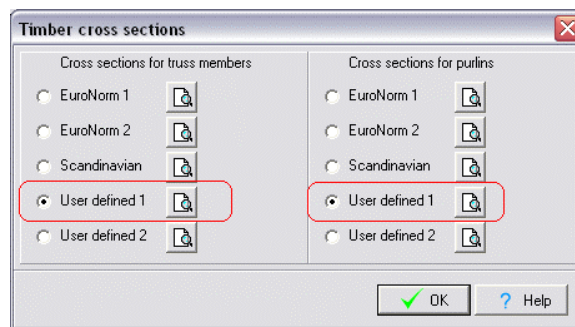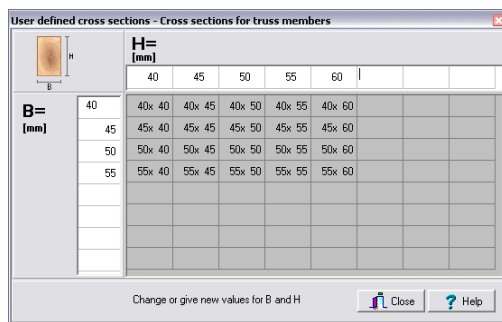
9.5   Roof loads
- Dead loads
- Snow loading
- Wind loads

9.6   Roofing

The top layer of the roof. Choose from the materials existing on the list or add your own. In order to change or add new values first click at Change values. To change the existing values simply click at the field and type the new values. To add or delete lines in the table click + and -. You specify the name of material and the values in kN/m^2 of the commonly used roofing in your area. At the button of the table you can change/specify the values for the dead load of ceiling under roof, live load on attic floors, and dead load of finishing of attic floors. These values will be used as default values in roof designs.
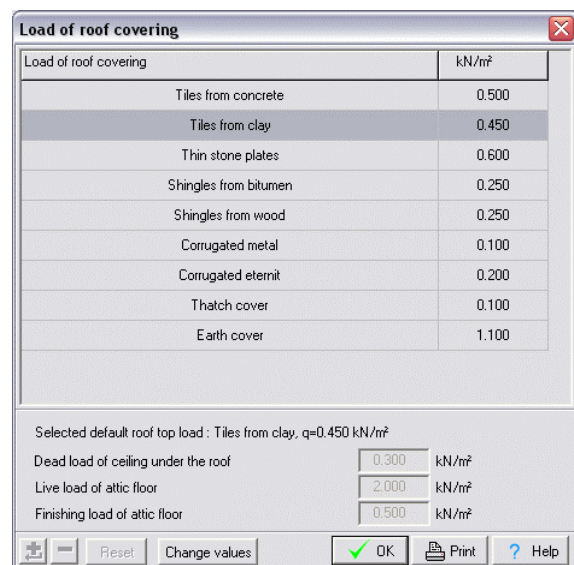
9.7   Snow loading

For the snow load you specify the characteristic snow load on the ground in kN/m^2. The snow loading on the structure is computed according to Eurocode 1, EN 1991-1-3:2003.

The snow loading on roofs according to Eurocode 1 (EC1) part 1 is:

s=μi.Ce.Ct.sk [kN/m^2]
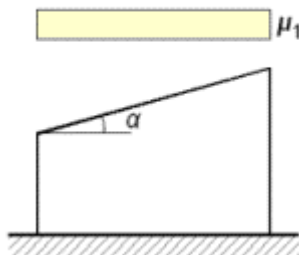
μi shape coefficient of the snow loading.

Ce and **Ct** coefficients depending on the exposure to wind and the thermal insulation of the roof correspondingly. Usually they have values =1.

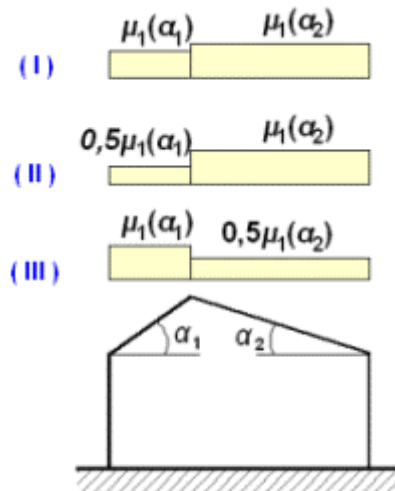**sk** Is the characteristic snow load value on the ground in kN/m^2.

For the EC countries the values of **sk** are given in Appendix C of Eurocode 1, EN 1991-1-1-3:2003.

The values of **μi** used in the program are according to EN 1991-1-1-3:2003, 5.3.2, 5.3.3.

Case of mono-pitch roof   **Case of double-pitch roof**
EN 1991: 2003 - 1-3, § 5.3.2   **EN 1991: 2003 - 1-3, § 5.3.3**)
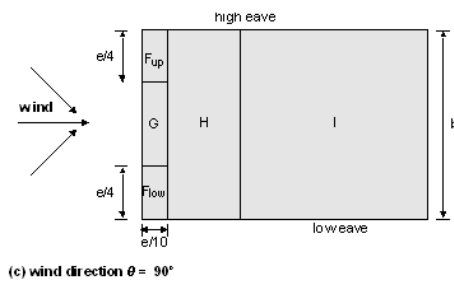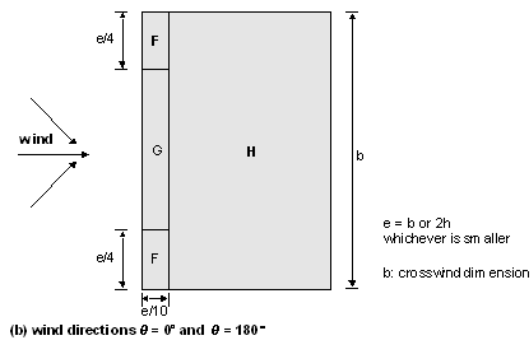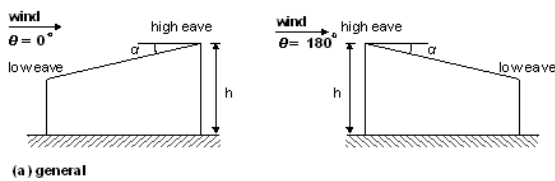
9.8   Wind loading

For the wind loading you specify the wind pressure on a vertical surface in kN/m^2.

The wind loading on the structure is computed according to Eurocode 1, EN 1991-1-4:2004.

For single-pitch roofs one loading (pressure) is considered.

For double pitch roofs two loading are considered, one with wind from left to right (pressure at left drag at right), and second with wind from right to left (pressure at right drag at left)

In the program the wind loading is computed as **we=qw.Cpe**, where **qw** is the wind pressure on a vertical surface in [kN/m²] given at the input data **qw=qref. Ce(ze).**

The wind loading according to Eurocode 1 EN 1991-1-4:2004 is : **we=qref. Ce(ze).Cpe.**

qref=(ρ.vref²)/2 [N/m²], ρ is the air density =1.25 kg/m³

vref is the wind reference velocity (m/s). **v1=qref. Ce(ze)**

Cpe is the pressure coefficient and is computed from the roof pitch according to EN 1991-1-4:2004.7.2.4 for mono-pitch roofs, and according to EN 1991-1-4:2004, 7.2.5 for double-pitch roofs.

Monopitch roofs (EN 1991-1-4:2004)   **Duopitch roofs** (EN 1991-1-4:2004)
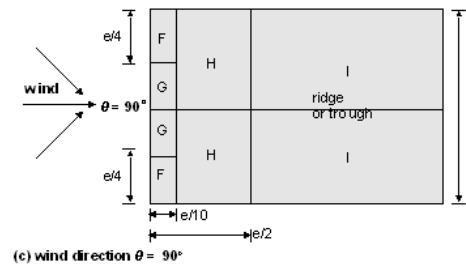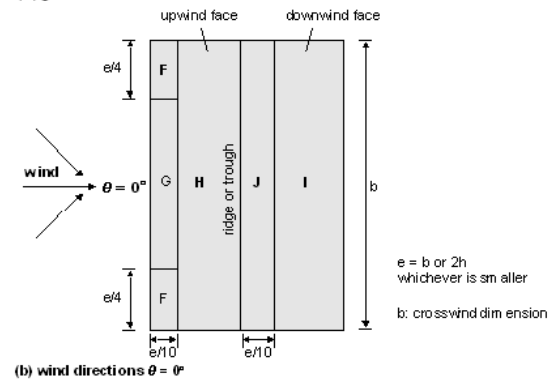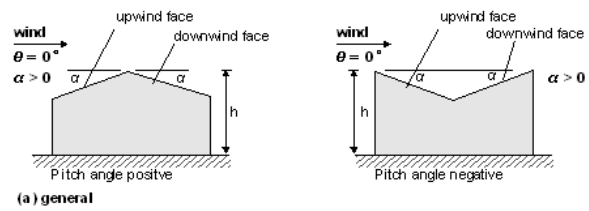
9.9   Seismic design

The seismic design of roofs is implemented in WOODexpress according to Eurocode 8, EN 1998-1, General rules seismic actions and rules for buildings.

The default values for the seismic analysis is selected from the menu [Parameters/seismic coefficients]
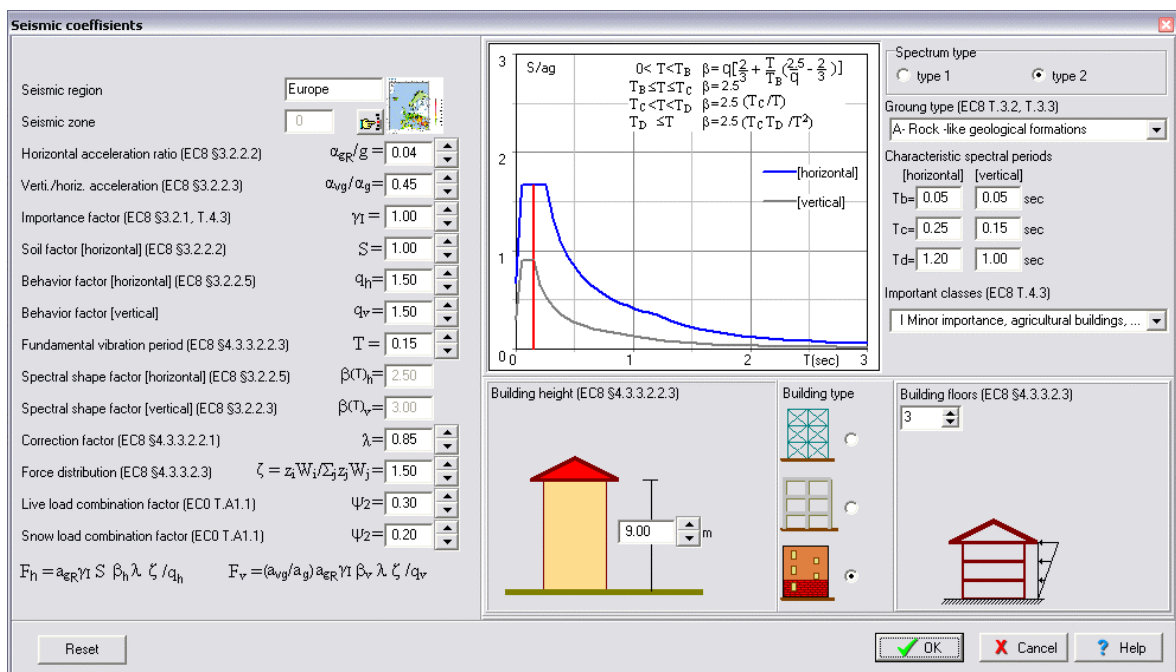

In the design window of roofs, check: Include seismic design, and the adjust the parameters for seismic design

You select the seismic zone from the map and automatically the Horizontal acceleration ratio is set.

By clicking on the map of Europe or specified country, choose seismic zone.

Choose spectrum type according to Eurocode 8, Table.3.3.

9.10   Eurocode and National Annex

Select the Eurocode and National Annex to apply in the design. The Eurocode parameters are set according to the National standards you choose. From the menu parameters, you can view the parameters.

If you select as National Annex- NA, the first option, Eurocode EN, then also ENV (prestandard versions of Eurocodes) are activated. These preversions of Eurocodes are expired, but are kept as an option in the program if someone wants to calculate with old standards for comparison.
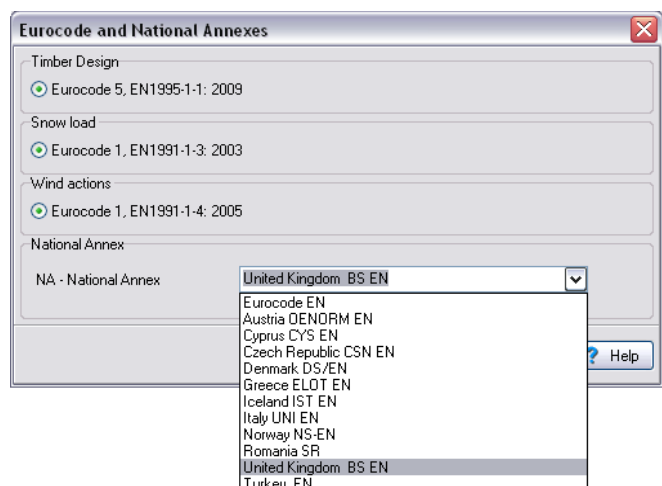
9.11 Parameters and assumptions

Design methods for timber connections

Elastic analysis. The forces on the fasteners due to a moment applied on the connection are proportional to the distance from the rotation center.

Plastic analysis. The forces on the fasteners due to a moment applied on the connection are all reaching the same value.

Yielding stress of plate steel in kN/mm2, used in the strength check of connection plate. A usual value for high strength steel is 500 kN/mm2. The net area of the plate is taken as 75% of the cross section area; (we assume 25% holes).

Tensile strength of nails, bolts and screws fuk in kN/mm2 is used to define the default characteristic yield moment of nails, bolts, and screws EN 1995-1-1:2004, eq. 8.14 and eq 8.30.

The **length of timber** is used to define the splices of the timber elements in the roof trusses.

Part of compressive design force. When the joint is in compression, the fasteners may be designed for only a part of the compressive force. This part can be defined in % here. Some comments on this subject you may find in Eurocode 5, 8.8.5.1 (3)

9.12 View all parameters

From the menu [Setup/ View all parameters] setting you can see the default values you have chosen for your designs. You can any time change the parameters from inside the calculation window. If you want to reset all your parameters to the original values of the program, press the button [Reset to original program values]

If you reset all parameters ALL your user defined values will be LOST. Program will close down and you must restart.

9.13 Units

The units used in the program are **SI (**System International Metric) units. The unit of any input value is marked next to the place you enter the data. The unit of every value in the report is also marked.

**Units used in the program**:

- length   [m]
- forces   [kN]
- moments [kNm]
- stresses [N/mm²] = [GPa]
- concentrated loads [kN]
- distributed loads   [kN/m²]
- line loads   [kN/m]

dimensions of timber sections, B[width]xH[height] in [mm]

10 Ultimate limit states

Design of various cases of single or combined action according to Eurocode 5, EN 1995-1-1:2004 §6. You specify the timber cross-section dimensions, the timber properties, the cross-section loads, and the design check is performed according to Eurocode 5.

- Tension
- Compression
- Bending
- Shear
- Torsion
- Stability
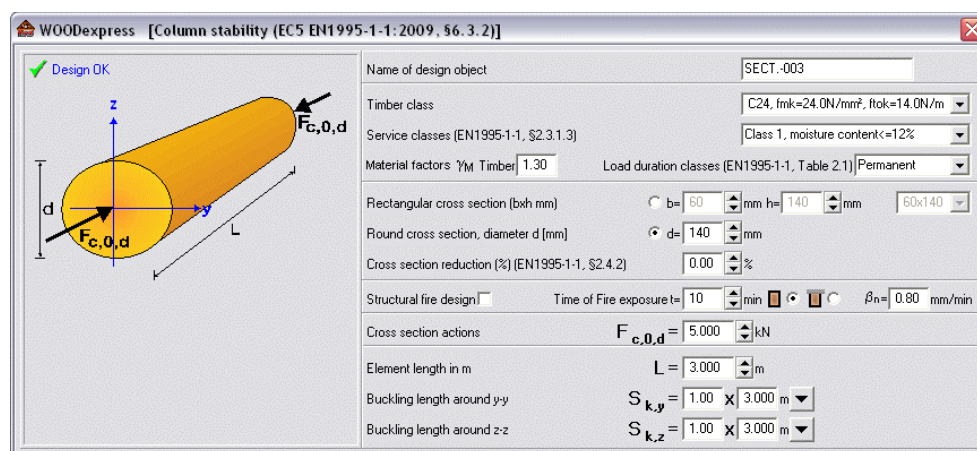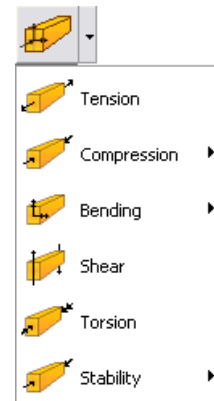
Many Ultimate limit cases can run automatically using the actions and section properties from other programs, using the Command language, Appendix 1.

11 Structural fire design

The structural fire design is performed for every stress case. Check the box structural fire design if you want to include this in your design.

Specify :   1. The time exposure to fire in minutes

2. The exposure of the cross section (all around or tree sides),

3. The notional design charring rate bn. This value is set automatically by the program with the choice of timber class, according to EC5 table 3.1.

The structural fire design is performed according to EC5 part 1-2, 4.2 reduced cross section method.
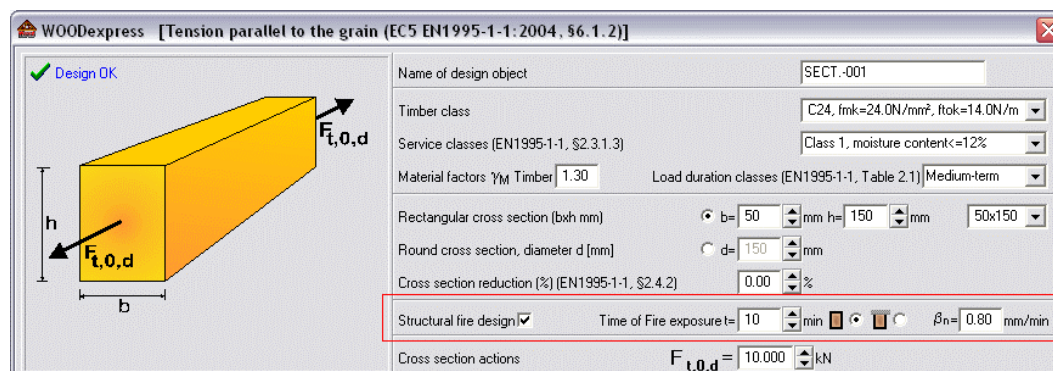
12 Connections

Design of nailed connections of various types. You specify the properties of the nails, (nail dimensions, round or square cross section), and the properties of the connection wood or metal plate. The Nail Capacity **Rd** is computed according to Eurocode 5, EN 1995-1-1:2004 §8.
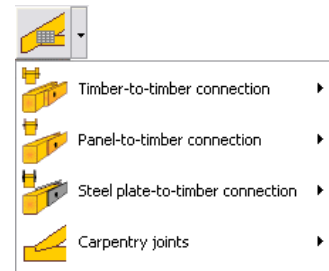
- Timber to timber connection
- Panel to timber connection
- Steel plate to timber connection
- Carpentry joints
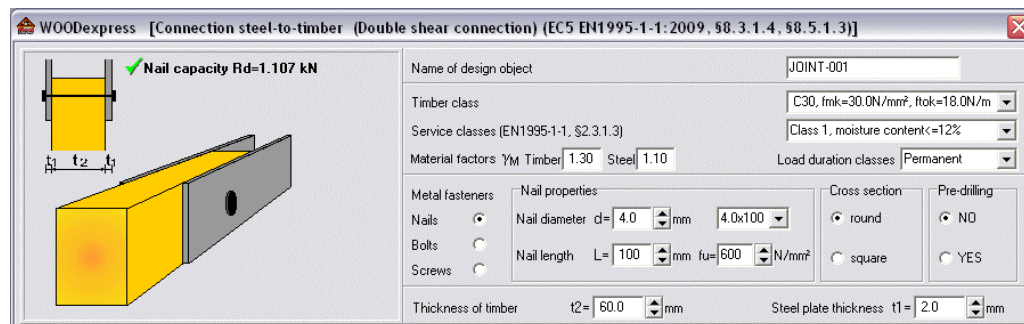
Example of Steel plate to timber connection
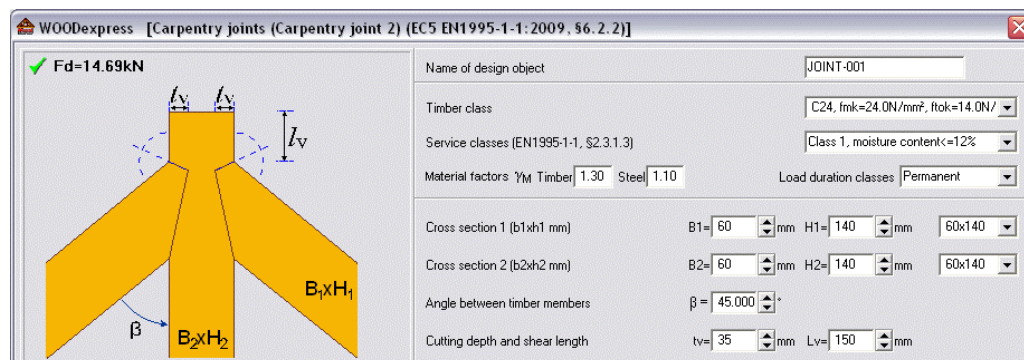
Example of Carpentry joint

13 Floors

In this session of the program you can design:

- Floor over one span
- Floor over two spans
- Simply supported beams
- Beam over two spans
- Timber-concrete composite floors
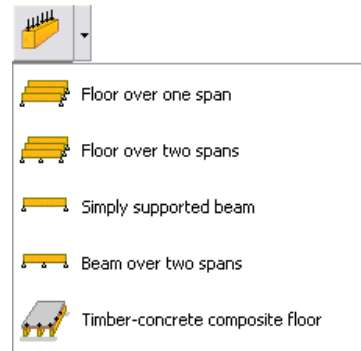
13.1   Floors over one or two spans

The internal forces are computed at the ends and the midspan of the floor beam, and the elastic bending deformations at midspan, for all the load combinations according to Eurocode 0,1 and 5. All the checks of Eurocode 5 are performed in ultimate limit state. The deformations are checked in serviceability limit state according to EN 1995-1-1:2004 §7.2. The Eurocode 5, EN 1995-1-1:2004 §7.3.3 considerations are taken into account for the check of floor vibrations.

The user has the option to select the way the moment of inertia of the floor beams is evaluated for the computation of deformations and vibrations.
a. as T cross/section with flange the floor finishing.
b as rectangular cross section.
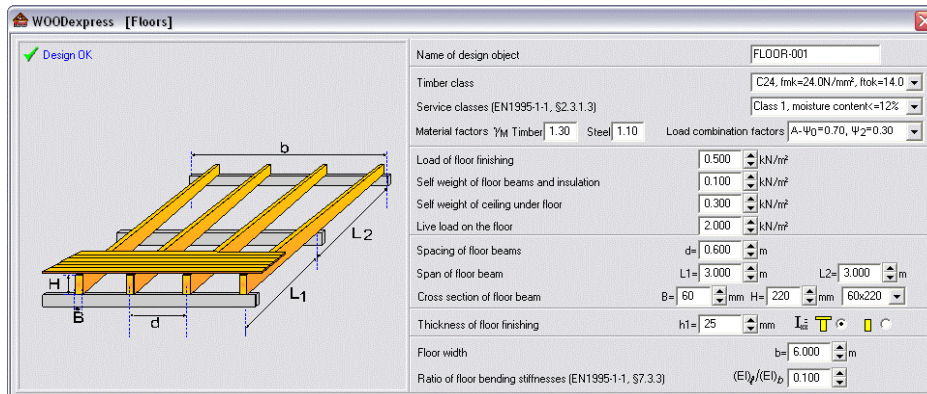
13.2   Simply supported beams and beam over two spans

Simply supported beams under line load. As in one span floors the internal forces are computed at the ends and the midspan of the floor beam, and the elastic bending deformations at midspan, for all the load combinations according to Eurocode 0,1 and 5.
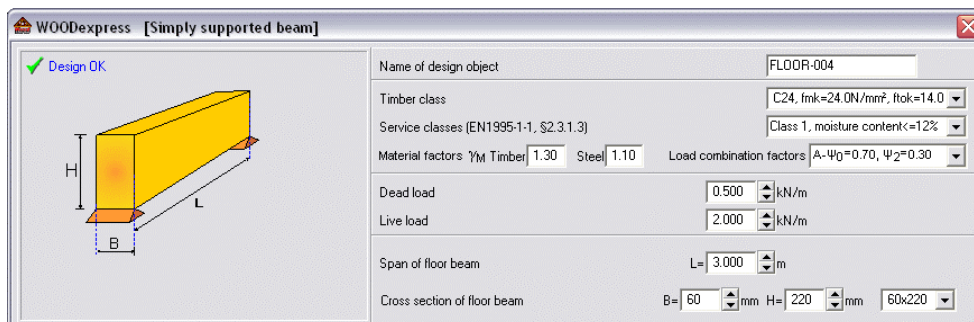
13.3 Timber-concrete composite floors

In this type of composite floors the concrete is only in compression, and timber is used in tension. The load carrying capacity and the out of plane rigidity of the floor is increased compared to a timber floor. The design of the composite cross section is based on EN 1995-1-1:Annex B.

In addition to the data supplied as in one span floors the data for the concrete/steel material, and slab thickness must be supplied, and the size, spacing and strength of the metal fasteners. If the spacing of the metal fasteners is set to zero (0) then the program selects an optimum value.

 If the concrete slab is quite thick then the neutral axis is inside the slab and a design failure message appears for the concrete slab. In this case you may reduce the thickness of the concrete slab or increase the gap between concrete and timber h3, so to have the concrete in compression only.

14 Roofs

From the programs main menu you can choose from various roof structures. Select the one you prefer, and perform a full design according to Eurocode 5. The loadings are evaluated according to Eurocode 1. The roof finishing (rafter), the purlins and the truss members are design for all the load combinations prescribed in Eurocode 1 and Eurocode 5. The internal forces of the trusses are computed with finite element analysis of the truss structure. The user can select the degree of stiffness of the connections (stiff or flexible joint connections). The truss is considered as a two dimensional frame structure, and the stiffness of the connections is adjusted according to the selected degree of stiffness.

In order to compute the design values for internal forces in various loading conditions, the internal forces are computed for unit loading, and from their combination the internal forces in various loading conditions are obtained.

All the load combinations according to Eurocode 0 and Eurocode 5 are taken into account and the checks are performed in the most unfavourable loading conditions, for combined action in ultimate limit state of design, according to EC 5 §6.

The deformations are checked in serviceability limit condition, according to EC 5 §4. The truss deflections are checked in **serviceability limit condition**, according to EN 1995-1-1:2004 §7.2.

The maximum deflection for finishing or purlins are computed as:
 1: Beams fixed at one end, when the span of finishing (distance of purlins) or the span of purlins (distance between trusses) is less than the half of the timber length defined in parameters and assumptions,
2: Simply supported beam when the span is larger than half of the timber length.

In EN the coefficients for checking the limit of allowable deflections are specified in the program as parameters. are defined in Table 7.2, as regions L/300~L/500, L/250~L350, L150~L/300. In the program in menu [Parameters/Parameters and assumptions] you can define the values you want. Default values (L/300, L/250 and L/150). The limits for cantilevering beams are taken as half of the above.

The natural frequencies of the roof trusses are also computed after a dynamic analysis.

The connections can be nailed, screwed or bolted joints with metal plates and they are dimensioned according to EN 1995-1-1:2004 §8.
A detailed report is produced, and shows analytically all the computations, design steps, design assumptions and references to the corresponding design code paragraphs. Automatic generation of truss drawings that can be manipulated with the CAD modulus. Warnings in red appear in the report in cases of inadequate design.
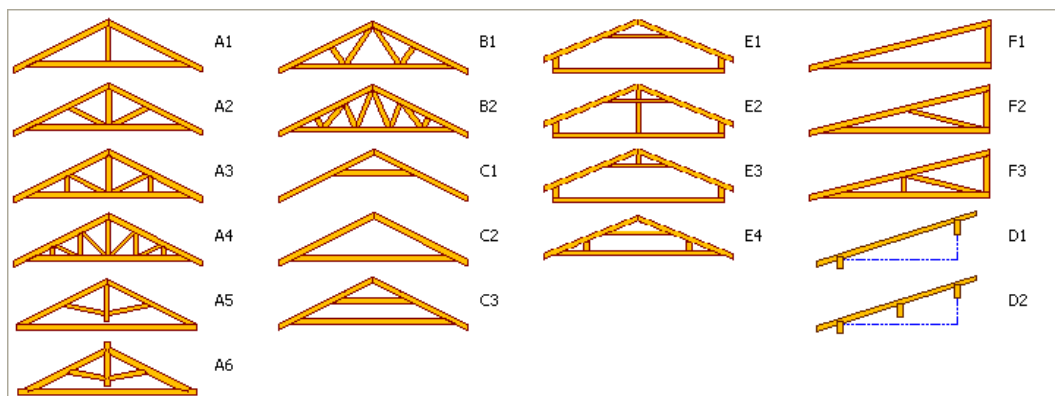
14.1 Roof dimensions

In the calculation window there is a sketch of the object. Specify the basic roof dimensions in meters [m], according to the roof truss drawing. The roof pitch (angle with the horizontal) may be

specified. If you give the roof height the roof pitch is obtained, otherwise, if you give the roof pitch the roof height is obtained.

You can specify dimensions at the nodal points (intersection of element axis) or the total dimensions. Select what dimensions you are giving.

You can also choose if you want the truss to be extended at left and right ends with cantilever elements or if you want an asymmetric roof..
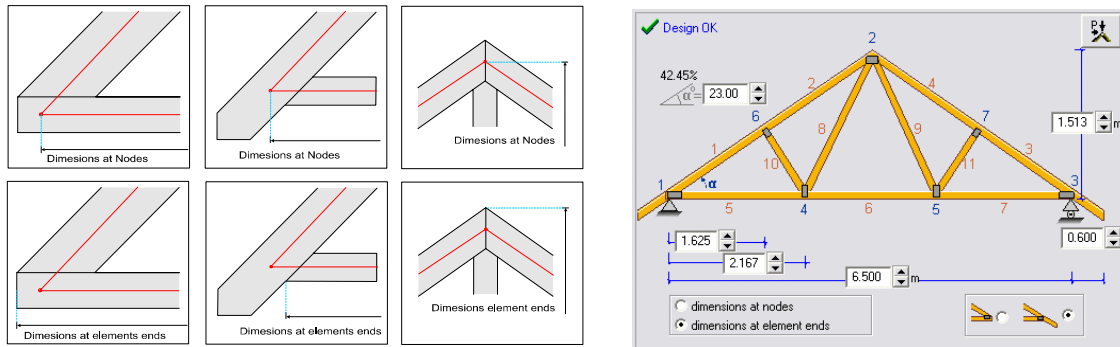
14.2 Element cross sections

For the elements of the truss you specify the cross section dimensions. The elements of the same cross section are grouped according to the most common sizes in different countries, EN1313-1. The cross section is rectangular with width B and height H. The dimensions of the cross section are in [mm].

In the program there are included user defined groups of timber sections for the truss elements and the purlins. The user can define two(2) groups for timber sections for truss elements and two (2) groups for purlins. From [parameters/timber sections] you can select the group of timber sections you want to appear in the design of roofs.
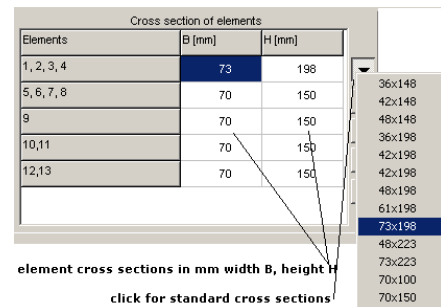
14.2.1 Reduction of timber sections

Specify the cross sections of the elements used in the truss. If the timber of the trusses are old, damaged or suffered from deterioration, you can specify a reduction of cross section in %.
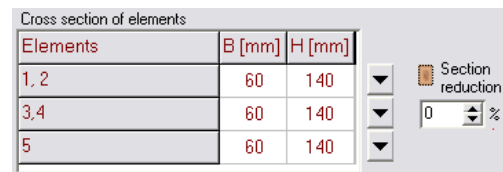
14.3 Roof secondary structure

You can select between three types of roofing.
- Roof consisting of purlins and timber finishing on top. The finishing may be rafters laid on top of the purlins. The tiles are laid on top of the finishing.
- Roof consisting of purlins and no rafters on top. The tiles are laid on top of the purlins.
- Roof with the timber finishing (rafters) nailed directly on the roof truss.
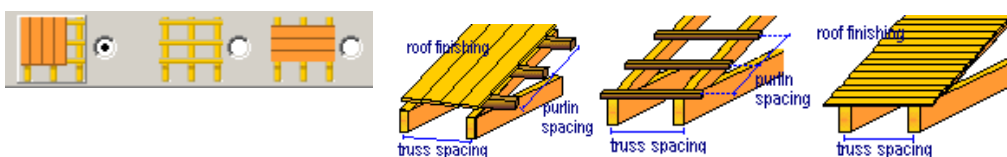
14.4 Roof loads

You specify the loads:
- dead load of the roof covering materials [kN/m^2],
- snow load on the ground [kN/m^2]
- wind load as wind pressure on a vertical surface in [kN/m^2] .
- self weight of purlins and roof finishing materials (insulation, water proofing) in [kN/m$^{2]}$
- dead load of the ceiling under the roof in [kN/m$^{2]}$
- dead and live loads on the attic floor in [kN/m^2]

The default values for these loads are set in the menu [Parameters].
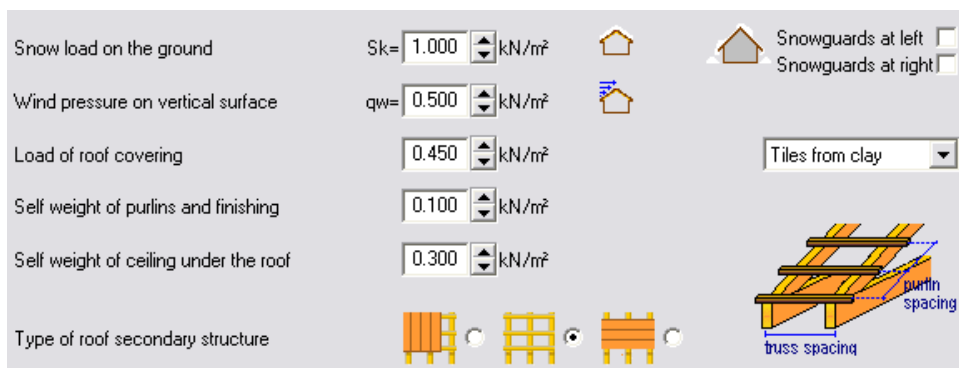
14.4.1 Concentrated loads

Enter the nodal loads in [kN]. Node (first column) is the nodal point where the concentrated force is applied. Fy, vertical nodal force positive downwards. Fx horizontal nodal force positive towards the right side. #Load in the last column is the load case for the nodal loads.

Nodal loads			
Node	Fy kN	Fx kN	#Load
2	5.2	0	0
2	2	0	1
2	3.2	1.2	5

Permanent loads,   Load case =0
Snow loads   Load cases 1 to 4 (see fig pg. 10)
Wind loads   Load cases 5 and 6 (pg. 11)

14.4.2 Snow and wind loading

For the snow load you specify the characteristic snow load on the ground in kN/m^2. The snow loading on the structure is computed according to Eurocode 1, EN 1991-1-3:2003. For the wind loading you specify the wind pressure on a vertical surface in kN/m^2.
The wind loading on the structure is computed according to Eurocode 1, prEN 1991-1-4:2004.

14.5 Spacing of trusses   Spacing of purlins
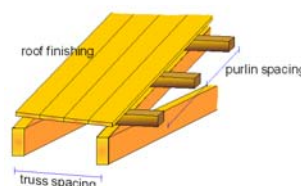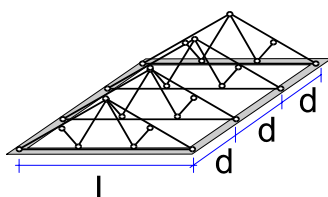
14.6 Truss connections

The connections are considered as bolted, nailed or screwed joints with single or double steel plate and they are dimensioned according to Eurocode 5, EN 1995-1-1:2004 §8.
- The user can select or specify :
- The type of connection, nailed, bolted or screwed connections.
- The diameter of the nails, bolts or screws, as well as the type of nails (round or square). The tensile strength of the steel for the nails, screws or bolts can be specified.
- The plate arrangement. Double plates on both sides of the truss surface or inside the truss member, single plate on the outside surface or inside the truss member.
- The thickness of connection plates.
- The yielding stress of plate steel.
- The degree of stiffness of the connection to perform the truss analysis.
- Multiple or single plates for each joint. Selecting multiple plates reduces the plate size in connections with more than two members.

14.6.1 Truss connections, Steel plates

Select the thickness of plates in mm. In the case of nailed joints you can select standard plates as (BMF plates 1.50 or 2.00 mm). When you choose BMF plates the program selects the optimal plate size from the available BMF plates, and marks it in the drawings details of the connection. If the BMP plate is getting too large, or no reasonable dimensions for BMF plates are available, the program will choose ordinary steel plates and this will also show in the drawing and reports.

You specify also the characteristic value of yielding stress of the connection plates fy in kN/mm². The default value is defined in parameters and assumptions

When the steel plates are getting very large, try multiple plates.

14.6.2   Truss connections, Nails

Specify the nail diameter (in mm), as well as the type of nails (round or square). In case of selecting BMF plates automatically the nails are set to 4mm shank nails. Specify also the tensile strength of the nails fu kN/mm², (EN1995-1-1:2004, Eq. 8.14). The default value of the tensile strength of nails is specified in parameters and assumptions.

The minimum nail length will be specified automatically by the program according to EN1995-1-1 §8, and shown in the report and drawings.

14.6.3   Truss connections, Bolts

When you are using bolts in connections, specify the following:

The diameter of the bolts in mm.

The position of the connection plates:
- double plate on both faces of the timber members.
- single plate on one face of the timber members.
- single plate in the middle of the timber members.
- double plate inside the timber members.

 The tensile strength of the bolts, fu in kN/mm², (EN1995-1-1:2004, Eq. 8.30). The default value of the tensile strength of bolts is specified in parameters and assumptions

 The steel plate thickness t in mm .

 The yielding stress of the steel of the connection plate, fy in kN/mm². The default value of yielding stress of the connection plates is defined in parameters and assumptions.
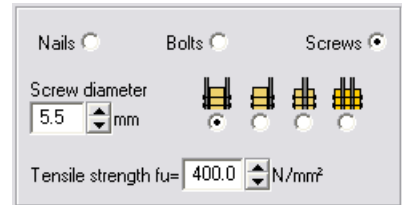
14.6.4   Truss connections, Screws

When you are using screws in connections, specify the following:

The diameter of the screws in mm.

The position of the connection plates:

- double plate on both faces of the timber members.
- single plate on one face of the timber members.
- single plate in the middle of the timber members.
- double plate inside the timber members.

The tensile strength of the screws, fu in kN/mm², (EN1995-1-1:2004, Eq. 8.30). The default value of the tensile strength of screws is specified in parameters and assumptions

The plate thickness t in mm.

The yielding stress of the steel of the connection plate, fy in kN/mm². The default value of yielding stress of the connection plates is defined in parameters and assumptions

The minimum screw length will be specified automatically by the program according to EN1995-1-1 §8, and shown in the report and drawings.

14.6.5   Carpentry joints

In carpentry joints you specify the cutting depth (tv) and shear surface length (Lv) for each joint.

When you select timber section dimensions tv and Lv are set to the default values which are tv=h/4 and lv= h. The minimum value if Lv is 150 mm.

When the [Alllow optimisation] is checked, the values the user is specifying for *t*v and *l*v, may change if from the solution comes out that larger values are needed. If this option is unchecked then tv and lv are kept and if the joint is inadequate an error message shows. By pressing the reset values *t*v and *l*v are reset to the default values (h/h and h or 150 mm)
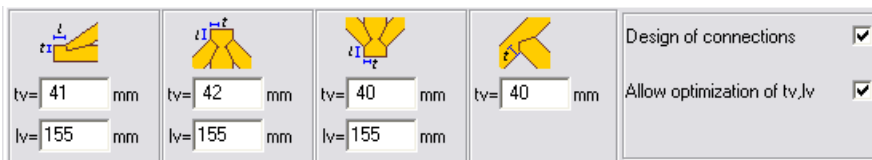
14.6.6   Stiffness of connections

You select the stiffness of connections. By moving the bar at left the truss is solved with very flexible (almost pin) connections. By moving the bar to the right the truss is solved with very stiff connections. The connections are considered as bolted, screwed or nailed joints with single or double steel plate and they are dimensioned according to EN 1995-1-1:2004 § 8.
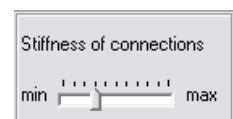
14.6.7   Truss connections, Plate arrangement

The user can select the connection plates to be multiple plates instead of single plate in order to reduce the plate sizes. If the pointer is moved to the left the program selects multiple plates in the connections. If the pointer moves to the right the program selects single plates in the connections. If the pointer is placed between the program depending on the size of the plate selects multiple or single plate automatically depending on the connection type and the angle of the connected members.
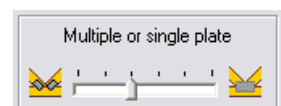
The user is advised to perform a solution with single (right position) and with multiple (left position), and select the preferable configuration.

14.7   Asymmetric roofs

From the truss types, the following can be designed as asymmetric.

A1, C1,C2,C3, E1,E2,E3.
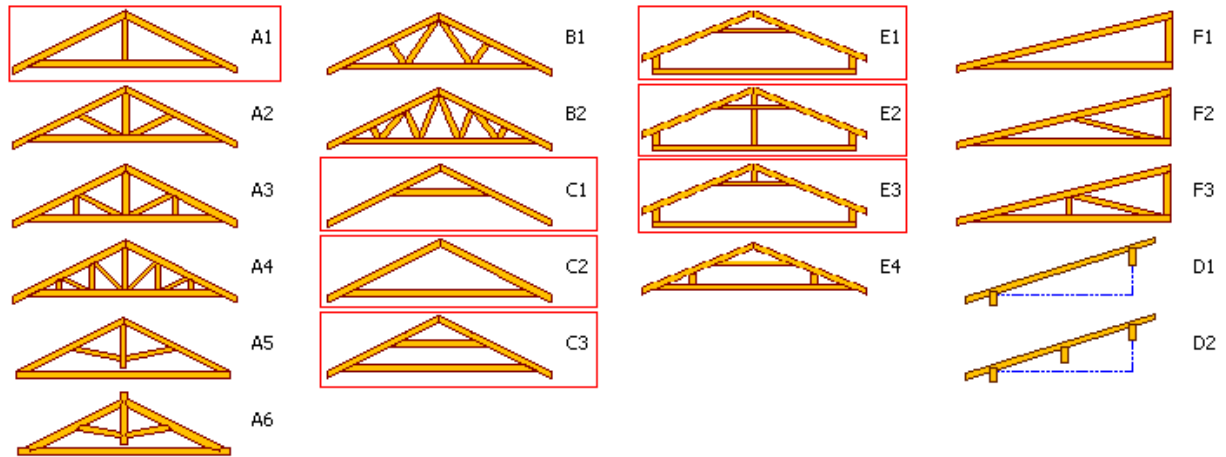
In design window, choose asymmetric and adjust the dimensions.
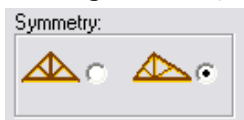
14.8   Hipped roofs

In the hipped roof analysis the program automatically sets the data for the 2-dimensional trusses which are used for the design of the roof. These data are the dimensions of the trusses and the loadings. The analysis of all the 2-d trusses are performed automatically.

These (2 or 4) trusses are numbered on the sketch of the roof, and they are grouped according to their dimensions and loading.

The data and the design of Each individual 2-d truss may be seen by clicking at Truss (1), Truss (2) etc. .

The user may want to override the automatic data generation of the 2-d trusses. In this case check

the [ ] , (do not change data). If you have the [auto computation] checked in the design window, all the computations of the 2-d trusses are performed automatically, and it may show a small delay of the cursor, due to many calculations.

15 CAD drawing of roof trusses

The CAD modulus of the program automatically creates a detailed truss drawing. You can adjust the scale of the drawing, and you can choose the visible layers. The properties of the drawing components, (line thickness, colour, text size) can also be adjusted. You can also specify the dimension units that are shown on the graph.

Before previewing or printing the drawing you can select the printing paper size, and move the drawing to the desired position on the paper.
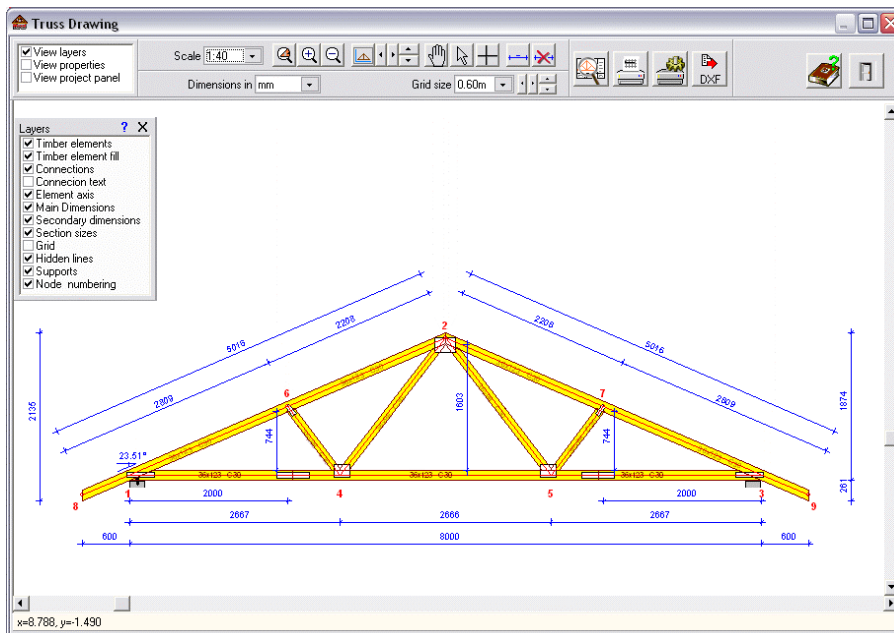
15.1 CAD Features

Scale of Drawing	Zoom	Layers	Dimension units	Grid

Scale/Move/Zoom If you cannot see all or parts of the truss on the screen, you can scale or move your drawing. You activate/deactivate the move command (hand) by double clicking on the drawing.

By right click you can change cursor.

Layers Choose the layers you want to be visible and printed. The properties of the layers are defined of the Properties of drawing components.

15.1.1   Dimension units

Choose unit for dimensions appearing on the truss drawing. This will be the default unit until you change it.

Grid If you want the grid to appear, from the layers panel, check the grid and choose the size from the pull down menu. By clicking on the small arrows on the right, you move the grid in relation to the drawing.

15.1.2   Line thickness, colour and font sizes

By using this panel you can adjust the appearance of the truss drawing.

Turn on or off the layers from the panel with Layers.

For the line type of **Axis and nodes**, choose line thickness **1 for dashed line**, line thickness 2 for the thinner solid line etc.
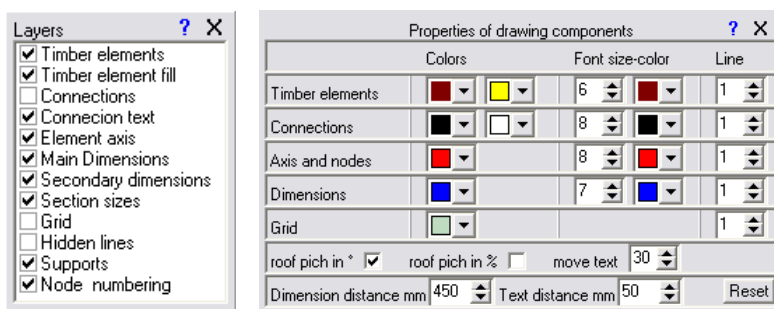
There are three levels of dimensioning. By adjusting the dimension distance you move the dimension lines further or closer to the truss element.

By adjusting the Text distance you move the connection text further or closer to the truss connections.

The values you are setting are maintained automatically. By clicking at Reset you restore the original default values of the program.

15.1.3   Add extra dimensions

If you want to add extra dimensions on the drawing, use the . Click on the point at beginning and the end of the distance you want to insert. Stop the process by right click.

If you want to remove all the extra dimensions added, use the .

For the standard dimensions, use the layer function to turn the dimension on or off.

The extra dimensions added are not maintained in the data file.

15.2   Print - preview drawing

Before you print your drawing it is advisable to preview the contents of you drawing first.

  Click on the Preview Button and set the parameters of printing.

Choose Paper size and orientation, Scale and check for Black and White **according to your printer.**

You move (click on the drawing and move the mouse) the drawing to place it at the desired position inside the drawing paper.
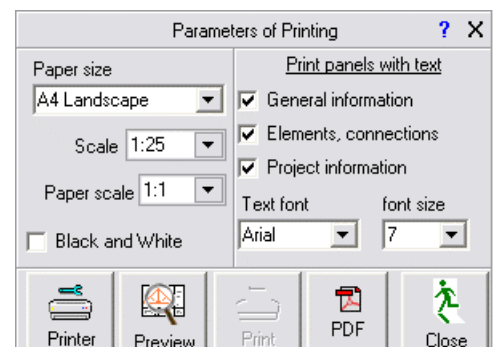
In case your screen size does not allow you to see all the drawing paper by choosing another Paper scale you scale down the screen image.

Choose the text panels you want included in your drawing.

When you check/uncheck a text panel you can see the area available for the truss drawing is changing.

You can change text font and size. Be aware if you increase the text size in A4 paper. The text may become too large for the text area.
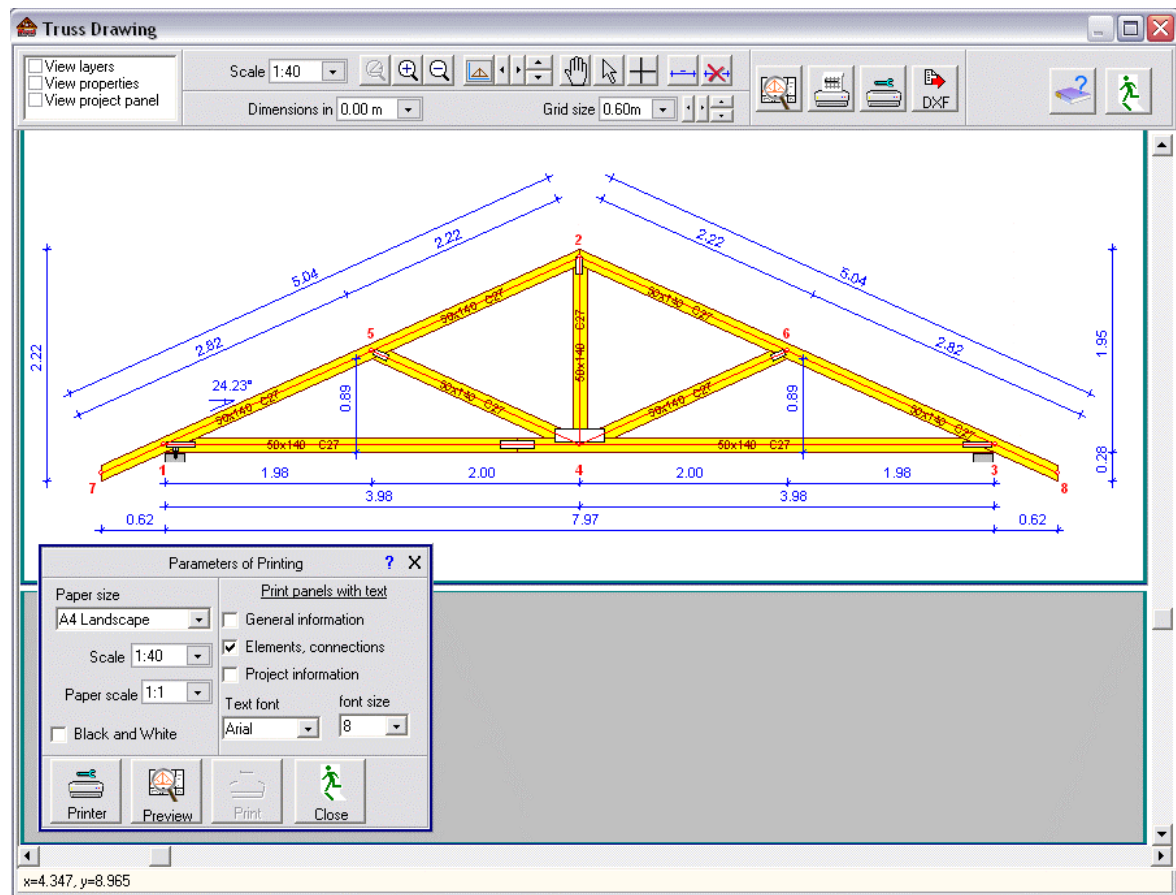
Print preview drawing.
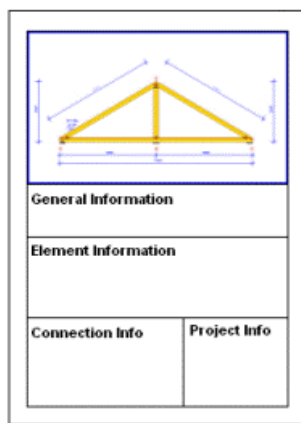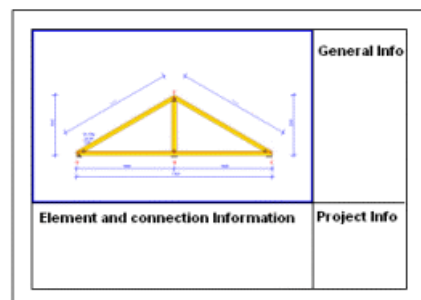
Page orientation for drawings.

15.3 Project panel

To edit appearance of the text panel for the drawings check the fields you want to be included and type the wanted text.
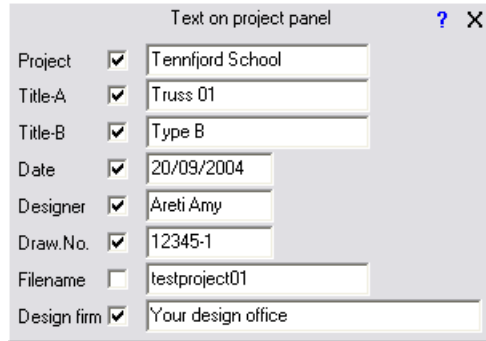
The project title is automatically taken from the name of the project.

The title A is automatically taken from the name of the design object.

The Design Firm title is automatically taken from the settings of the report parameters, see pg. 28, report page footer.

15.4 Export drawing to PDF format

From the CAD modulus of the program you can save you truss drawing in PDF format
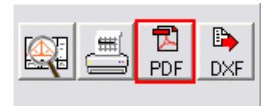
15.5 Export drawing to dxf format

From the CAD modulus of the program you can save you truss drawing in .dxf format. This file can be read from Autocad In the window that appears specify the file name and adjust the text size and decimal symbol in the new file.
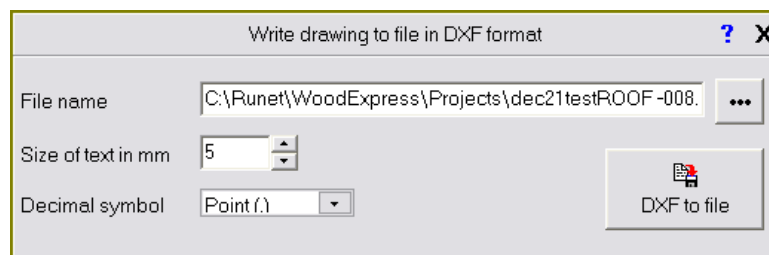
16   Program settings

16.1 Greek character setup

According to the notation used in the Eurocodes the report contains many Greek mathematical symbols. In most cases the system takes care of the language settings by itself. But depending on the Window installation the Greek mathematical symbols may or may not appear right. If you have older Windows versions you can add Greek language support in your Windows. Go to [Settings/Control Panel/Regional and Language Options/Advanced].

If your Windows do not support Greek mathematical symbols, then from [Setup/Greek character support] or [Set UP/Greek character support] select NO. The Greek characters will appear as: alpha, beta etc. , in the report.

16.2 Language Set Up

You can choose the language of the program from the menu [Setup/Language Setup]. The design objects have to be recalculated to show the new language in the calculations and report.

 *Not available for all versions.
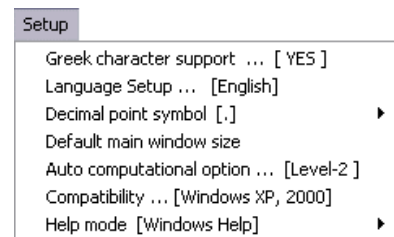
16.3 Decimal point symbol

You specify (.) or (,) for the decimal point appearing in the input data and the reports.

16.4 Screen sizes

You can resize the main screen, and its size is maintained. The size of the main screen is automatically set to the size the last time you opened the program. You can reset the main screen to the default size by clicking at [Setup/Default screen size].

Most of the program windows have a height limited by the height of the main screen. If you want to have these windows larger, simply enlarge the main program screen.

16.5 Default screen dimensions

By clicking at this menu option you reset the screen size to the original set in the program. Each time you change the main screen size its size is saved and when you reopen the program it appears with the new size. The size of the main window is also the default size for the calculation windows that are opened afterwards.

16.6 Auto computational option

Depending on the speed of the computer the user can choose to have the computations performed simultaneously with the data input/change or when clicking the button .

Options from the menu Setup/ Auto computational option:

NO: Computations only when press Computations.

Level - 1: In objects with large output (as roofs) computations only when press Computations, and in objects with small output else when you change input data';

Level - 2: Computations performed simultaneously when you change input data for all objects.

If you have chosen the option Level -2, and see that it is bothering slow when you change data in the roof trusses, uncheck the box Auto computation and the mode will turn into Level -1 mode.

For new machines use Autocompute checked as the machines are very fast. In the case of hipped roofs because the computation part involves the computation of all the sub-trusses, probably it is preferable to uncheck the Autocomputation mode.
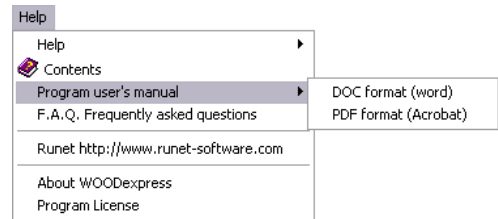
16.7 User's guide

You can preview, or print the program user's manual. You select to view it as a Word(doc) or as an Acrobat (pdf) document.

17 Reports

After designing the desired timber objects they can be printed into a report. The report will contain all the objects that are checked in the [design objects] window. The order of which the objects will appear in the report can be adjusted with the two arrows at the bottom of the design objects window. Adjustments for the report, font, margins, logo of caption or footnote, etc. can be done from [Report Setup].
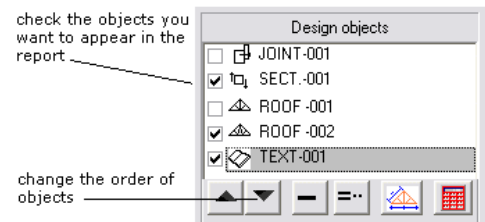
If more than one roof truss is checked for report, the report for roof truss designs becomes very long. A warning message will show up to inform you. You can either continue or make the report smaller. To reduce the size of the report, uncheck one or more of the trusses in the design object window.

If hipped roofs are included the report contains a large number of pages. In this case is advisable to see or print the report of each subtruss separately from the window of the calculations.

17.1 Preview report

The report preview contains all the objects that are checked in the [design objects] window. You can adjust the order in which the object appears in the report by using the two arrows at the bottom of the [design objects] window.
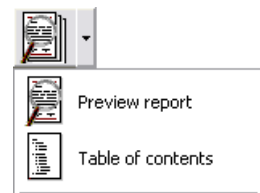
In order to preview the report you must have a valid printer installed in your system. If you work in a network there must be installed a network printer. Otherwise the system will report – invalid printer. Simply connect/add a printer, or select another printer as default.

Two different preview types are available and can be selected from

[Report Setup/Page-Preview/Single-Composite]
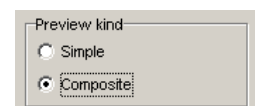
From the [Report Setup] you can adjust the appearance of your report such as font, margins, logo of caption or footnote, etc. In [Report Setup/Various/Change page for each chapter], you may choose to start each design object in a new page.

The report preview and printing for the roofs contain detailed drawings of the connections.

It has been noticed that the report preview does not appear right at the connection section, in Windows 98 and Me, although it is printed right. To avoid this you may check the compatibility to be Windows 98 and Me, from [Setup/Compatibility Mode]. In this case you must print the report using the print command and not from inside the preview window. This problem does not exist in windows 2000 and XP.

17.2 Print report

The report contains all the objects that are checked in the [design objects] window. The order of the objects appearing in the preview can be adjusted with the two arrows at the bottom of the design objects window.

In order to print a report you must have a valid printer installed in your system. If you work in a network there must be installed a network printer. Otherwise the system will report – invalid printer. Simply connect/add a printer, or select another printer as default.

Adjustments for the report, font, margins, logo of caption or footnote, etc. can be done from Report Setup. In [Report Setup/Various/Change page for each chapter], you may choose to start each design object in a new page.
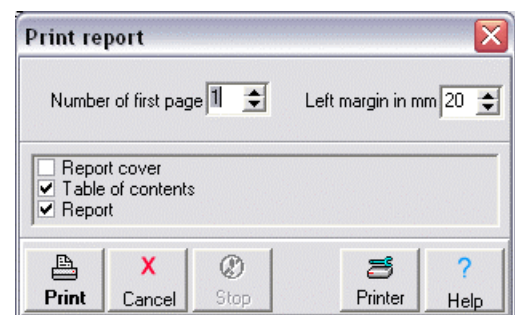
From the printing dialog you can adjust the page number of the first page and the left margin in mm. More adjustments for the report, font, margins, logo of caption or footnote, etc. can be done from Report Setup.

In [Report Setup/Various/Change page for each chapter], you can choose to start each design object in a new page.

17.3 Report to file

You can export the report to DOC file, PDF file and to a RTF file (text only), which can be opened by another text editor or Microsoft's Word.

In DOC and PDF format, the report is exported as it looks, with the drawings and paragraph settings. In the RTF format only the text is exported.
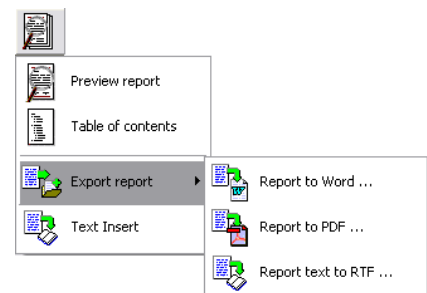
17.3.1 RTF files

To make the text appear right in the new rtf file, when you first open the file in Word, do the following:
- select all the text
- expand the margins to 16 cm or 6.5 inc.
- set the font to courier new
- set the font size to 8.

If your windows do not support Greek character set, the Greek mathematical symbols will not appear right.

Depending on the Window installation the Greek mathematical symbols may not appear right. If you have Windows XP or 2000 you may add Greek language support in your Windows, from Windows [Settings/Control Panel/Regional and Language Options/Advanced].

If your windows do not support Greek mathematical symbols, then from [Setup/Language Set-Up] select the language without the support of Greek mathematical symbols. Thus the Greek characters will appear as alpha, beta etc.

17.4 Text insert

You can insert your own text in the, with the [Preview/Text Insert] command.

In the window which opens, type in the text or read it from a ***.rtf** file. This text object can be treated like all the other objects of the program.

text object ─
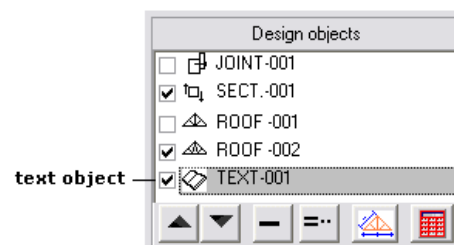
17.5 Report editing

If you want to edit the report, save the file to word or rtf format and do the changes from the new document.

17.6 Printer Setup

Select printer, and adjust printer properties. Standard Windows dialog.

17.7 Troubleshooting

Greek Mathematical symbols

According to the notation used in the Eurocodes the report contains many Greek mathematical symbols. Depending on the window installation the Greek mathematical symbols may not appear right. In case you have windows XP or 2000 you may add Greek language support in your windows, from Windows [Settings/Control Panel/Regional and Language Options/Advanced].

In case your windows do not support Greek mathematical symbols, then from [Setup/Language Setup] select the language without the support of Greek mathematical symbols. In this case in the report the Greek characters appear explicit e.g. alpha, beta etc.

18 Report parameters

From the main menu you can adjust the appearance and the printout of the reports by using the [report parameters setup].

18.1 Report setup

Header, page footer, paper size, orientation, line distance, margins etc.
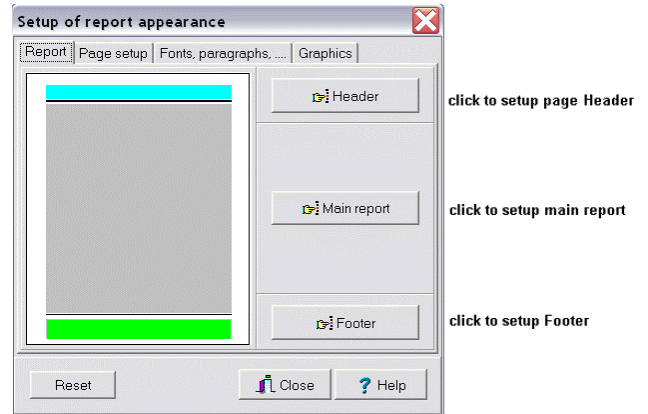
18.1.1 Report Page Header

On the page's header it may appear, a small picture (bitmap), at the project title, the chapter title, the page number and a horizontal line underneath. By checking the corresponding boxes you can choose which of the above objects you want to appear on the caption. The position of these objects is regulated from the numbers in mm you specify in the boxes in columns 2 and 3. In the last column you can set the font, or select a bitmap for the icon, or the thickness and colour of the line. At the page place you can specify the letters you want to appear before the page number e.g. Pg. With the buttons at the bottom you can preview or print a sample of the header.
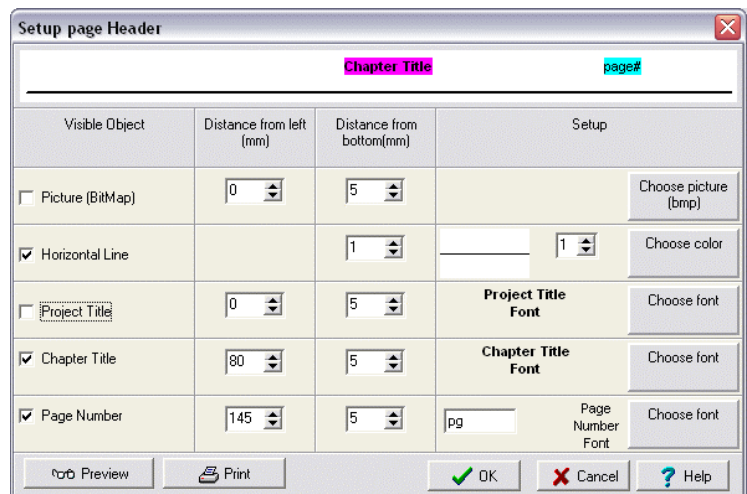
18.1.2 Main report

You select the font type, as well as the size of the font. For the font type it is **wise to select non proportional fonts, such as Courier, Courier new, Lucida Console, so that the report formulas and tables to be aligned properly**.

You may also specify the page margins (left, right, top, bottom) in millimetres (mm).
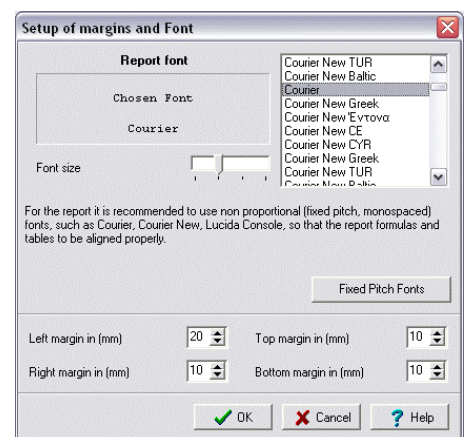
18.1.3 Report page footer

On the page's footer it may appear, the logo of the design firm, the file name of the project, the report subtitle or chapter title, the report date, and a horizontal line on top. By checking the corresponding boxes you can choose which of the above objects you want to appear on the caption. The position of these objects is regulated from the numbers in mm you specify in the boxes in columns 2 and 3. In the last column you can set the font, or the thickness and colour of the line.

With the buttons at the bottom you can preview or print a sample of the page footer.
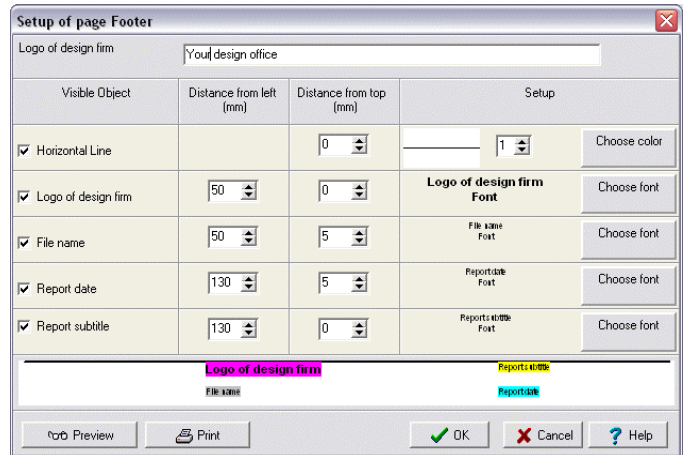
18.2   Page setup

18.2.1 Report cover

You can design your own front page of the report. From [Report Setup/Page Preview/Report Cover]

You can edit the features on the cover of the report. The cover may be displayed with an outline, a picture (from bitmap file) and two text lines. You can adjust the contents with the checkboxes.

The outline's colour and thickness be changed.

If you wish a picture on the cover, you can choose from the examples or choose your own bitmap.

The style of text in the two text lines from the font style editor box.

You can Preview your new report cover and also do test print.
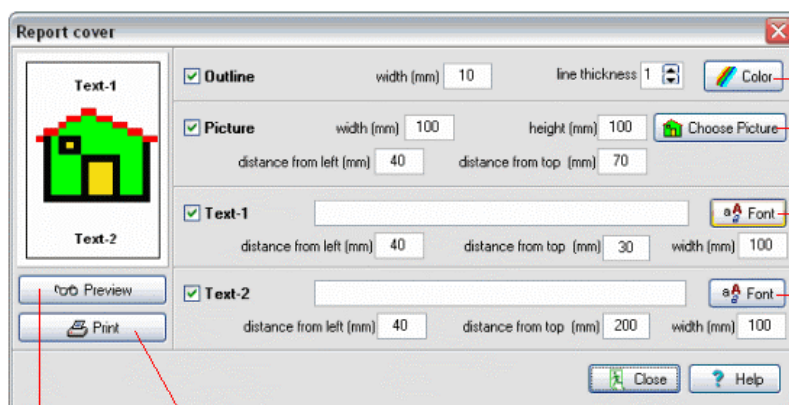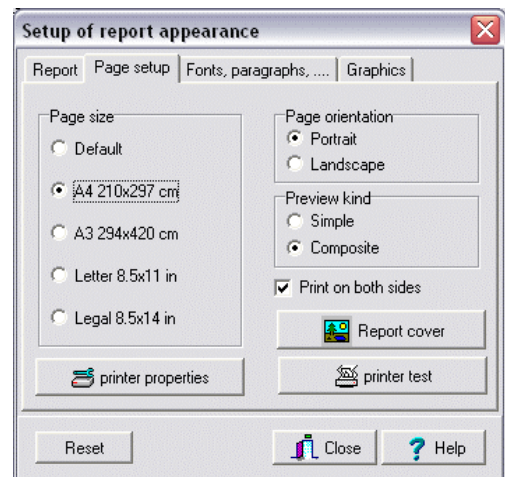
18.2.2 Report setup, Various

Paragraph numbering in report. Check this if you want your chapters numbered.

If you check [**Change page for each chapter**], the computations of every design objects will start on a new page.

If you check [**Print Errors in red colour**], warnings will be printed in red when computations are not satisfying the codes or standards.

You can adjust the line distance in mm and the paragraph left margin in characters.
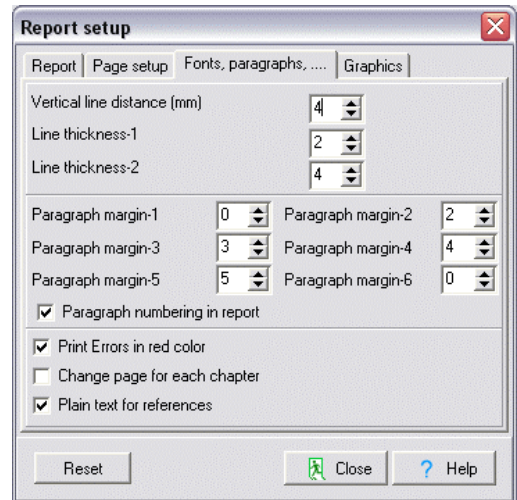
The indentation of paragraphs can be adjusted from the margin already set in [Report setup/Page-setup/main report]. The indentation can be adjusted in characters (not mm), and

margins are according to the figure.

```
paragraph margin-1 ———————|Computations of structure object : COLUMN-001

paragraph margin-2 ———————|Column cross section in biaxial bending

                            EC 2, Table 2.3 (ac=1.50, as=1.15)
paragraph margin-3 ———————|Concrete-Steel class: C25/30-S500, Concrete cover (EC 2, §4.1.3.3)

                            Column of rectangular cross section b=0.300 m, h=0.300 m
paragraph margin-3 ———————|Loads, axial Nsd=100.00 kN, moments Msdxx=0.00 kNm,Msdyy=0.00 kNm

                            Msdxx=    0.0 kNm isdxx=(Msdxx/bh²fcd)=    0.0/(0.300x0.300²x16700
                            Msdyy=    0.0 kNm isdyy=(Msdyy/hb²fcd)=    0.0/(0.300x0.300²x16700
                            Nsd  =- 100.0 kN      vd=(Id/(bhfcd)   =- 100.0/(0.300x0.300 x16700
                            from biaxial bending with compression diagrams utot=0.10
```

Mark **Plain text for references** if you don't want the references to be bold and underlined.

```
1.1.4. Material properties  (EC5 EN1995-1-1:2004, §3)    ☑ Plain text for references


1.1.4. Material properties  (EC5 EN1995-1-1:2004, §3)    ☐ Plain text for references
```

19 Engineering tools

19.1.1 Unit conversion   Cross sections
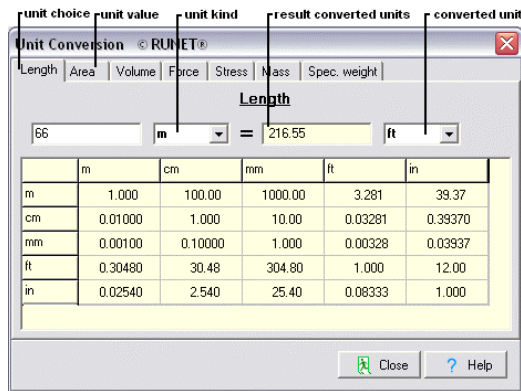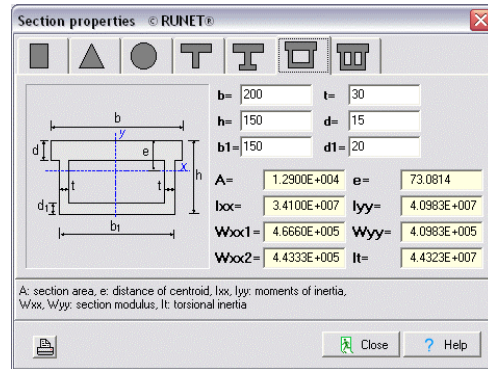
Cross section properties. Give the cross section dimensions b,h,...etc, and the cross section properties (area, moments of inertia, and section modulus), are computed.

19.1.5 Areas (x,y coordinates)

To find the area of a more or less complicated shape you can use the area of the region.

Give the points of the surrounding of an area, in polar (r , theta) coordinate. The area and the centroid of the region are computed. On the right of the window appears a sketch of the region, and the centroid is marked in red. With the buttons at the bottom left you may save the data in a file and read them back again later.
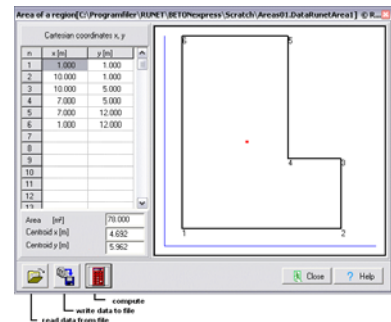
19.1.6 Area (polar coordinates)

Give the points of the surrounding of an area, in polar (r , theta) coordinate. The area and the centroid of the region are computed. On the right of the window appears a sketch of the region, and the centroid is marked in red. With the buttons at the bottom left you may save the data in a file and read them back again later.
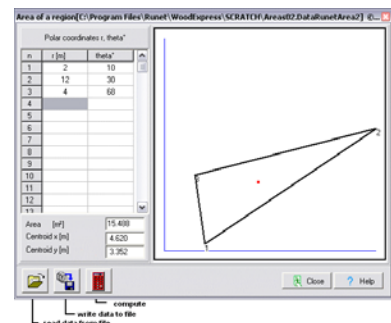
19.1.7 Areas (sum of triangles)
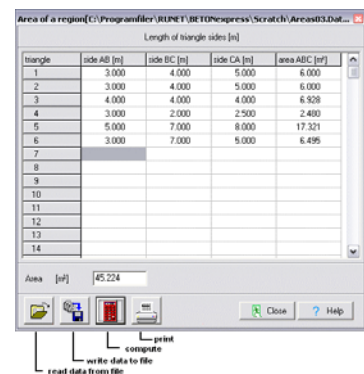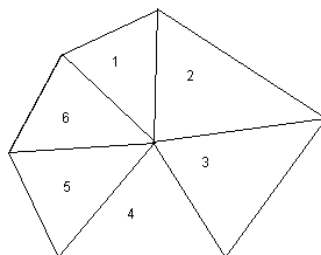
20 Eurocodes

Group of standards for the structural and geotechnical design of buildings and civil engineering works.

These standards are a set harmonized technical rules for civil engineering works, in the members of the European Community. National Application Documents are national standard for adapting the Eurocode to native requirements.

The structural Eurocodes are:

Eurocode 0 (EC0) EN 1990 Basis of structural design

Eurocode 1 (EC1) EN 1991 Actions on structures

Eurocode 2 (EC2) EN 1992 Design of concrete structures

Eurocode 3 (EC3) EN 1993 Design of steel structures

Eurocode 4 (EC4) EN 1994 Design of composite steel and concrete structures

Eurocode 5 (EC5) EN 1995 Design of timber structures

Eurocode 6 (EC6) EN 1996 Design of masonry structures

Eurocode 7 (EC7) EN 1997 Geotechnical design

Eurocode 8 (EC8) EN 1998 Design provisions for earthquake resistance of structures

Eurocode 9 (EC9) EN 1999 Design of aluminium structures.

21 References

Eurocode 0

1990:2002 Basis of structural design

Eurocode 1

EN 1991-1-1:2002 Actions on structures – general actions – Densities, self-weight and imposed loads.

EN 1991-1-2:2002 Actions on structures – general actions – Actions on structures exposed to fire

EN 1991-1-3:2003 Actions on structures – general actions – Snow loads

prEN 1991-1-4:2004 Actions on structures – general actions – Wind actions

Eurocode 5

EN 1995-1-1:2004 Design of timber structures – General – Common rules and rules for buildings

EN 1995-1-2:2004 Design of timber structures – General –Structural fire design

EN 338:1995 Structural timber – strength classes

EN 338:2003 Structural timber – strength classes

Eurocode 8, EN 1998-1, General rules seismic actions and rules for buildings.

1. STEP/Eurofortech, Timber Engineering, Centrum Hout, Netherlands, 1995 Werner, Holzbau Verlag GmbH, Dusseldorf 1991
2. W.M.C. McKenzie, Design of structural timber, Macmillan press Ltd. 2000.
3. Ozelton & J.A. Baird, Timber Designer Manual, Blackwell Science Ltd, 2000.
4. BMF Bygningsbeslag a/s, Danmark.
5. Timber Engineering Step 1, Centrum Hout, The Netherlands
6. Timber Engineering Step 2, Centrum Hout, The Netherlands
7. Atlante del Legno, Natterer Herzog Volz, UTET, 2003

WOODexpress Command Line

WOODexpress can run as a post processor of various Finite Element Programs (ANSYS, SAP2000,) to perform the concrete element design. The communication of WOODexpress with other programs can be done with a command file in simple text format. Each line of this <u>Command line file</u> describes an object that is going to be created in WOODexpress. Commands and data can be read in WOODexpress and the design objects are automatically created. The format of the command text file is given below.

How to import the command file

Click at menu File/ <u>Read Command Line File</u>
   Browse and [Open] the file with the command lines (.TXT)
   Enter the name of the new project file as .WoodExpress data.
   … and the Design objects are created from the commands and the data of the text file.

Example of command text file

```
MATER TC=C24, SC=1, LC=0, gM=1.30, gS=1.10, gG=1.35, gQ=1.50


S-TENSION    NM=section-001, TP=0, B=58,  H=150,  Rs=12, Na=15.45
S-TENSION    NM=section-002, TP=1, D=150, Rs=12, Na=25.45


S-COMPR-PAR NM=section-011, TP=0, B=60,  H=150,  Rs=16, Na=17.35
S-COMPR-VER NM=section-012, TP=0, B=60,  H=150,  Rs=5, Na=12.55, La=60, LL=70, L1=180
S-COMPR-ANG NM=section-013, TP=0, B=70,  H=140,  Rs=5, Na=15.45, Aa=20


S-BEND     NM=section-021, TP=0, B=60,  H=150,  Rs=0, My=12.45, Mz=3.5
S-BEND+T   NM=section-022, TP=0, B=60,  H=150,  Rs=5, Na=11.34,  My=12.45, Mz=3.5
S-BEND+C   NM=section-023, TP=0, B=60,  H=150,  Rs=6, Na=9.46,   My=11.45, Mz=2.5


S-T+C+B   NM=section-031, TP=0, B=60,  H=150, Rs=10,  My=7.45, Mz=3.5
S-T+C+B   NM=section-032, TP=0, B=60,  H=150, Rs=10, Na=9.46,   My=11.45, Mz=2.5
S-T+C+B   NM=section-033, TP=0, B=60,  H=150, Rs=10, Na=-9.46,  My=11.45, Mz=2.5


S-NM NM=section-034, TP=0, B=60,  H=150, Rs=10, Na=-10.46,  My=11.45, Mz=2.5


S-SHEAR   NM=section-041, TP=0, B=90,  H=140, Rs=10, Vs=15.45
S-SHEAR   NM=section-042, TP=0, B=140, H=90,  Rs=12, Vs=6.45
S-SHEAR   NM=section-043, TP=1, D=200, Rs=10, Vs=16.45


S-TORSION  NM=section-051, TP=0, B=120,  H=140, Rs=5, Mt=15.45
S-TORSION  NM=section-052, TP=1, D=120,  Rs=0, Mt=15.45


S-STAB-N   NM=section-061, TP=0, B=120,  H=140, Rs=5, Na=15.45, L=3.20, SKy=1.00,SKz=2.0
S-STAB-NM  NM=section-062, TP=0, B=120,  H=140, Rs=5, Na=15.45, My=2.1, Mz=3.2, L=3.20, SKy=1.00,SKz=2.0
S-STAB-M   NM=section-063, TP=0, B=120,  H=140, Rs=5, My=2.1, Mz=3.2, L=3.20, SKy=1.00,SKz=2.0


S-STABILITY  NM=section-071, TP=0, B=120,  H=140, Rs=5, Na=15.45, My=2.1, Mz=3.2, L=3.20, SKy=1.00,SKz=2.0
S-STABILITY  NM=section-072, TP=0, B=120,  H=140, Rs=5, Na=15.45,  L=3.20, SKy=1.00,SKz=2.0
S-STABILITY  NM=section-073, TP=0, B=120,  H=140, Rs=5, My=2.1, Mz=3.2, L=3.20, SKy=1.00,SKz=2.0
```

Command Line explanations

Every part of a command must separated with comma (,)
Code words (first word and words with =) must be exactly the same
Capital and small letters are the same

MATER	Materials and partial safety factors
TC=C24	Timber class
SC=1	Service class (1,2,3)
LC=0	Load duration class
	0:permanent,
	1:long term,
	2:medium term,
	3:short term
gM=1.30	Material factor for timber
gS=1.10	material factor for steel (if needed)
gG=1.35	γ_G Partial factor for permanent loads
gQ=1.50	γ_Q Partial factor for variable loads

If Material Command is omitted, then the <u>default values</u> that are set in the program the moment you read the command file are taken.

Many material cards may be included. Each one affects the set of following commands.

S-TENSION	Cross section tension parallel to the grain
NM=Section-001	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=58	Cross section width in mm
H=150	Cross section height in mm.
Rs=10	Cross section reduction (%)
Na=15.45	Cross section axial load in kN

S-COMP-PAR	Cross section compression parallel to the grain
NM= Section-011	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=16	Cross section reduction (%)
Na=17.35	Cross section axial load in kN
If TP=1 then D=150 cross section diameter in mm.	

S-COMP-VER	Cross section compression perpedicular to the grain

NM= Section-012	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm
Rs=5	Cross section reduction (%)
Na=12.55	Cross section axial load in kN
La=60	Loaded length in mm
LL=70	Loaded length in mm
L1=180	Loaded length in mm
S-COMP-ANG	Cross section compression perpedicular to the grain
NM= Section-013	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=70	Cross section width in mm
H=140	Cross section height in mm
Rs=5	Cross section reduction (%)
Na=15.45	Cross section axial load in kN
Aa=20	Load angle in degrees.

S-BEND	**Cross section Bending**
NM= Section-021	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=0	Cross section reduction (%)
My=12.45	Cross section bending moment in kNm
Mz=3.50	Cross section bending moment in kNm

S-BEND+T	**Cross section Bending and Tension**
NM= Section-022	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=5	Cross section reduction (%)
Na=11.34	Cross section tension in kN
My=12.45	Cross section bending moment in kNm
Mz=3.50	Cross section bending moment in kNm

S-BEND+C	**Cross section Bending and Compression**
NM= Section-023	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.

Rs=6	Cross section reduction (%)
Na=9.46	Cross section compression in kN
My=11.45	Cross section bending moment in kNm
Mz=2.50	Cross section bending moment in kNm

S-T+C+B	**Cross section with Axial load (tension or compression) and bending moments**
NM= Section-031	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
Na=9.46	Cross section compression in kN
My=11.45	Cross section bending moment in kNm
Mz=2.50	Cross section bending moment in kNm

(any of the forces or moments can be omitted)

S-NM same as S-T+C+B

S-SHEAR	**Cross section with shearing force**
NM= Section-041	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
Vs=15.45	Cross section shearing force in kN

S-TORSION	**Cross section with torsion**
NM= Section-051	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	Section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
Mt=15.45	Cross section torsional moment in kN

S_STAB-N	**Cross section stality in Axial load**
NM= Section-061	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
Na=15.45	Cross section axial load in kN
L=3.20	Element length in m
Sky=1.00	Buckling length around y-y axis (Ly=sky.L)
Sky=2.00	Buckling length around z-z axis (Lz=skz.L)

S_STAB-NM	Cross section stality Axial load and moments
NM= Section-062	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
Na=15.45	Cross section axial load in kN
My=2.1	Cross section bending moment My in kNm
Mz=3.2	Cross section bending moment Mz in kNm
L=3.20	Element length in m
Sky=1.00	Buckling length around y-y axis (Ly=sky.L)
Sky=2.00	Buckling length around z-z axis (Lz=skz.L)

S_STAB-M	Cross section Lateral stability
NM= Section-063	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
My=2.1	Cross section bending moment My in kNm
Mz=3.2	Cross section bending moment Mz in kNm
L=3.20	Element length in m
Sky=1.00	Buckling length around y-y axis (Ly=sky.L)
Sky=2.00	Buckling length around z-z axis (Lz=skz.L)

S_STABILITY	Cross section stability (Axial load and moments)
NM= Section-071	Name object (any name up to 16 characters)
	*** NOTE object names are <u>unique</u> must not repeated *****
TP=0	section type 0:rectangle, 1:round cross section
B=60	Cross section width in mm
H=150	Cross section height in mm.
Rs=6	Cross section reduction (%)
Na=15.45	Cross section axial load in kN
My=2.1	Cross section bending moment My in kNm
Mz=3.2	Cross section bending moment Mz in kNm
L=3.20	Element length in m
Sky=1.00	Buckling length around y-y axis (Ly=sky.L)
Sky=2.00	Buckling length around z-z axis (Lz=skz.L)

General case of stability.

If Axial load is missing = lateral stability,

if moments are missing=buckling with only axial load.