KRASTA

Shiploader Small

Version 1

1 PREFACE 2

2 KRASTA BASICS 2

2.1 SCREEN LAYOUT 2
2.2 UNITS 2
2.3 LANGUAGES 2
2.4 SIZES OF TEXT AND GRAPHIC ELEMENTS, COLORS 3
2.5 CHANGING THE MODEL VIEW 3
2.6 SELECTING NODES AND BEAMS 3

3 THE EXAMPLE MODEL 3

3.1 OPENING THE MODEL 3
3.2 EXPLORING THE GEOMETRY 3
3.3 MASSES 5
3.4 LOADS 5
3.5 KINEMATICS, ORIENTATIONS, SITUATIONS 6
3.6 SOLVING THE MODEL 7
3.7 RESULT EVALUATION 8
3.7.1 Results for basic and combination load cases 8
3.7.2 Result controls and proofs 9

Kühne BSB GmbH
Stahlbau und Fördertechnik

Mina-Rees-Str. 5A
64295 Darmstadt
Deutschland
http://www.krasta.de
support@krasta.de

B
S
B

Date: 08.08.2014
Pages: 11

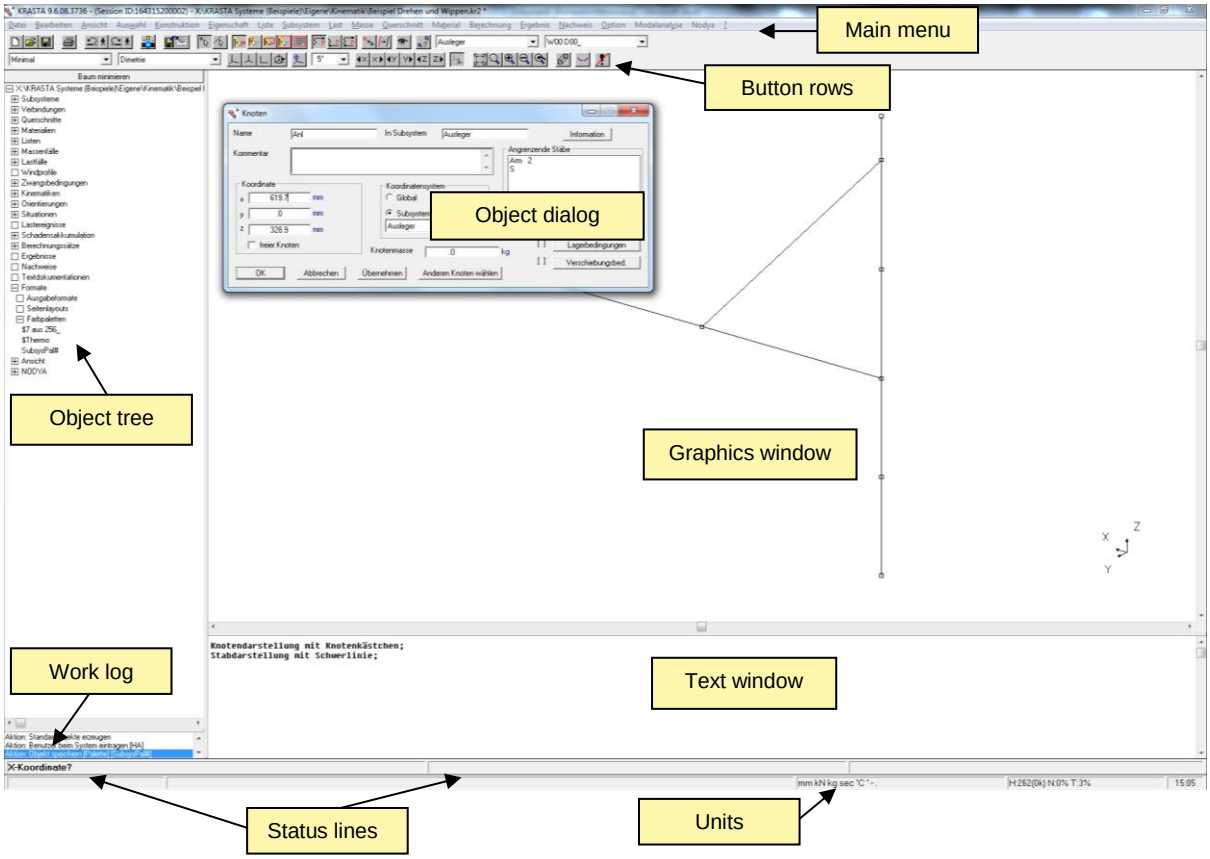
1 Preface

The provided KRASTA model „KRASTA small example“ is a virtual model of a shiploader. The only purpose of this model is to demonstrate selected capabilities of KRASTA. The model itself is highly simplified and essential loads and structural details are missing. The model is not meant as an example on how to map a real structure on a KRASTA model.

This example was created with KRASTA 9.6. Older KRASTA versions may lack features mentioned in this document.

2 KRASTA basics

2.1 Screen layout



2.2 Units

Units can be changed at any time even when object dialogs are open. Main menu:

Option > Units

2.3 Languages

The language can be changed at any time for the user interface (dialogs, menus etc.), result output and help file. Main menu:

Option > Languages

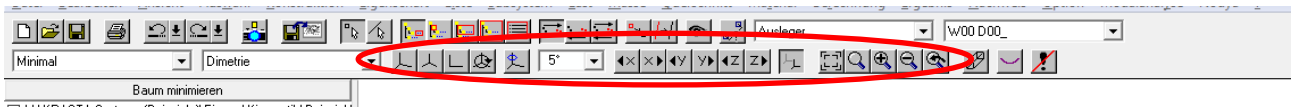
2.4 Sizes of text and graphic elements, colors

The graphic display can be customized. Main menu:

Display > Screen
Display > Colors

2.5 Changing the model view

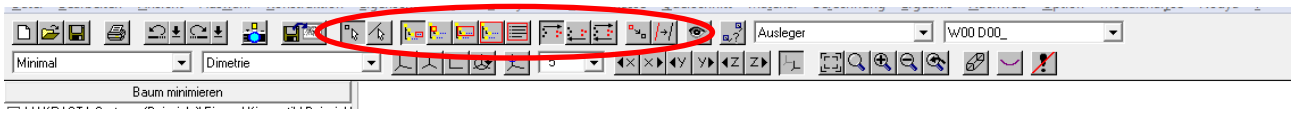
You may rotate the display of the graphics window with the display buttons in the second button row.



From KRASTA 9.6 on dynamic rotate and zoom is available, please check the manual for the keyword “Orbit”.

2.6 Selecting nodes and beams

Activate the according selection mode for beams or nodes. In KRASTA you may select, unselect or invert beams and nodes by clicking in the graphic window. The corresponding buttons to change the selection mode can be found in the first button row.



3 The example model

3.1 Opening the model

If the model is still in a zip file choose from the main menu: File > Unpack and open. Navigate to the zip file and press the open button. KRASTA unpacks the zip file and opens the model files.

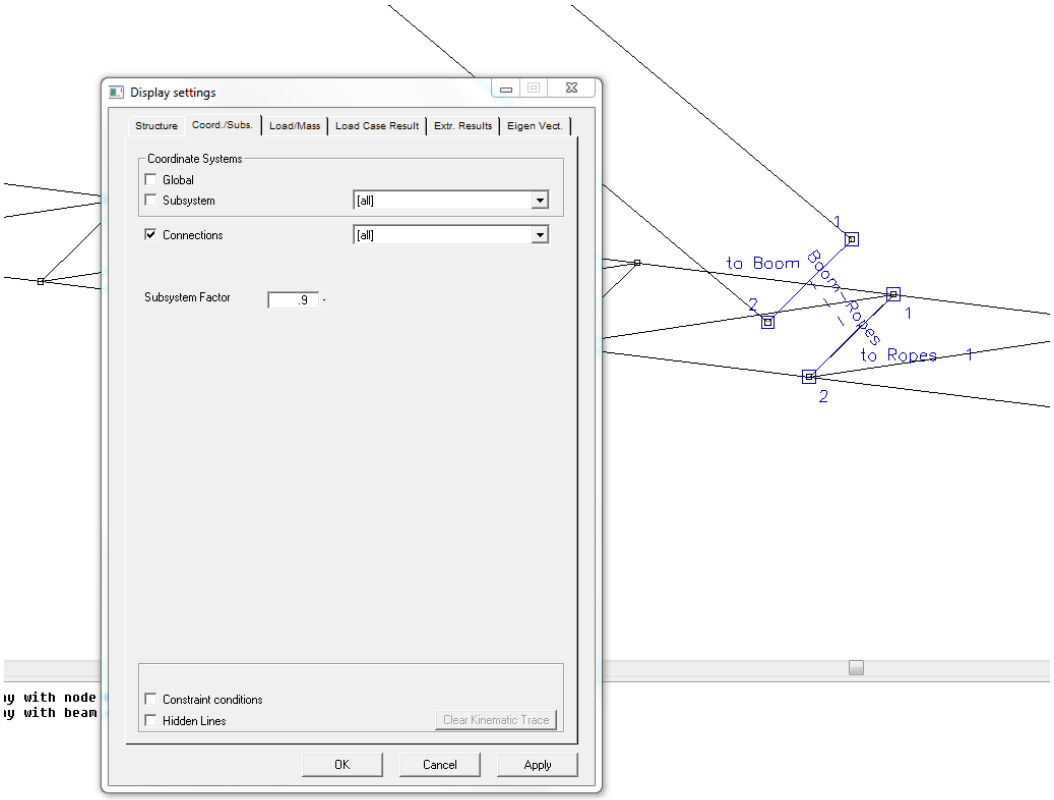
If you have already unpacked the zip file choose from the main menu: File > Open.

3.2 Exploring the Geometry

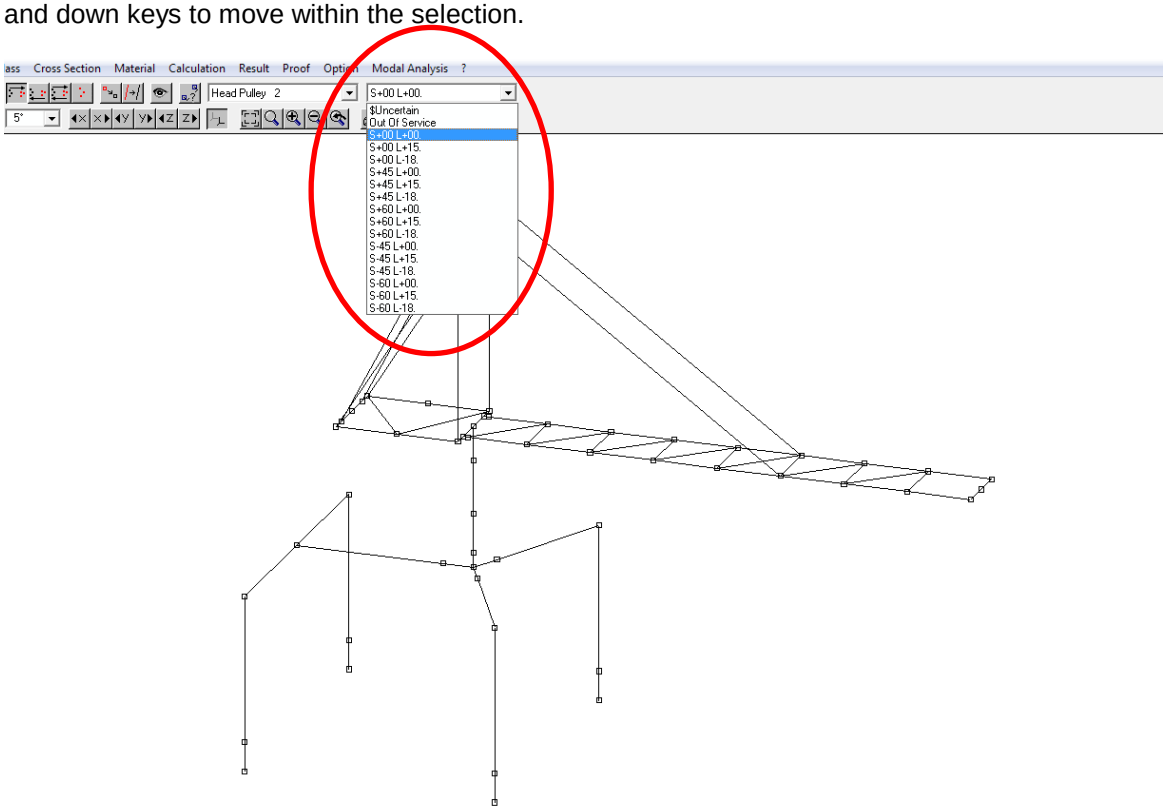
Change the model view (see 2.5) to get an impression of the global structure. You may switch on and off additional information for the graphics window, e.g. cross sections, beam and node names, support conditions and joints. Click on the button "display settings" (eye icon) and activate the desired options. “Display settings” may be saved and recalled for a faster workflow in the left selection box. Some “display settings” have already been saved during the modeling of the example.

Display settings in the example model:
Minimal – Standard display setting with nodes and beams
Cross sections: Cross sections are drawn on the beams
Jnts Supps Axis: Beam joints, node support conditions and beam local axes are drawn

The model consists of 7 different subsystems which can be moved relative to each other. To get an idea on the different subsystems select a subsystem factor smaller than 1 (0.9 will do in most cases) and switch on “Connections” on the “Coord./Subs.” Tab of the “display settings” dialog.

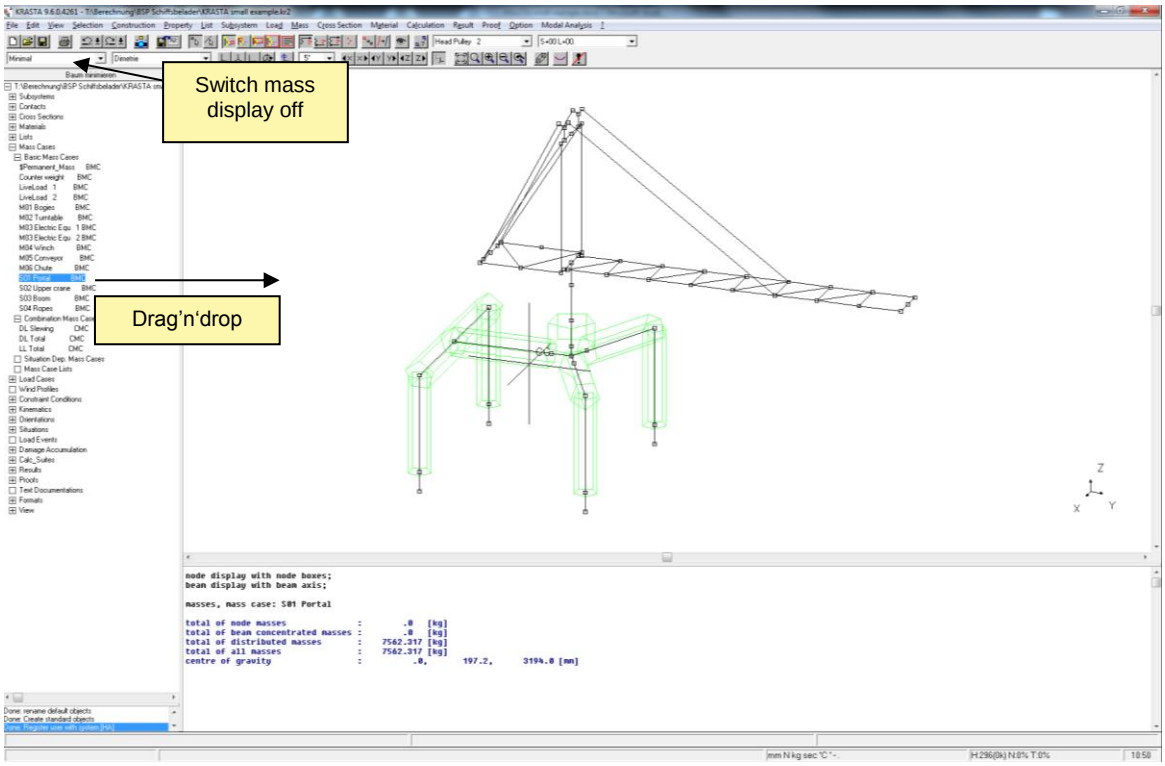


Various working positions have been included in the model as “Situations”. You can switch through the situations with the top right selection box. Once this selection box has focus you may use the arrow up and down keys to move within the selection.



3.3 Masses

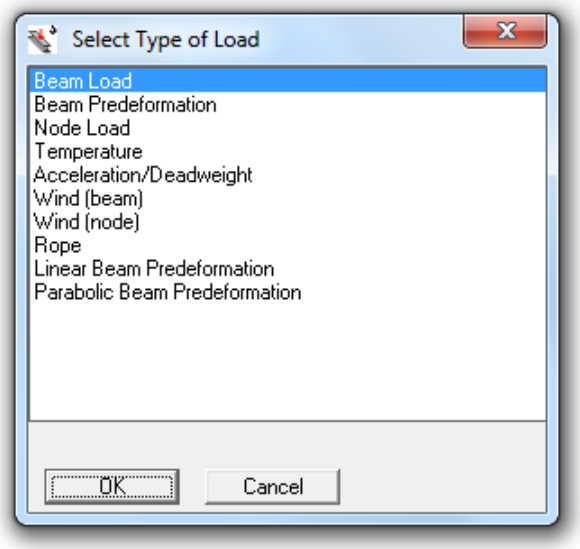
Masses are entered as “basic mass cases” where you can put node/beam masses on nodes/beams or apply a mass factor to the structural mass of (parts of) the structure. The masses are usually taken from parts lists or are estimated from earlier projects. To see the mass distribution and the sum drag the mass case from the tree and drop it on the graphics window. To switch off the mass display select “minimal” from the saved display settings.



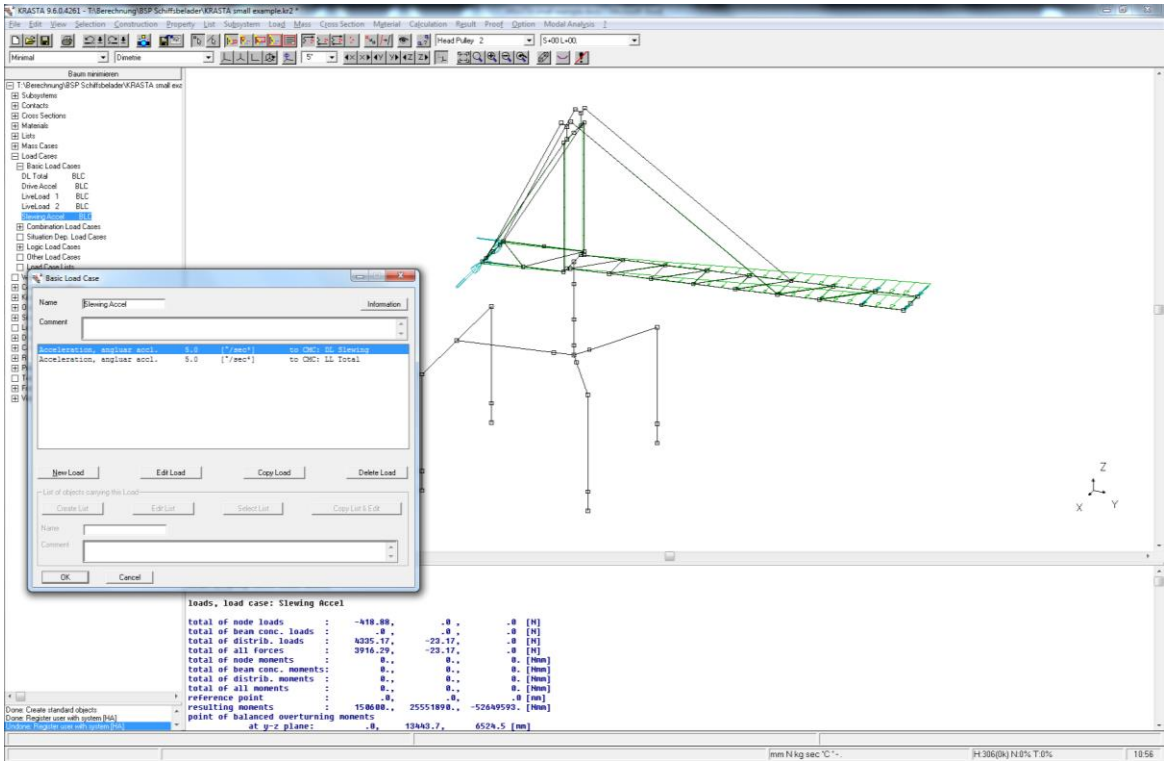
Basic mass cases are combined into combination mass cases depending on the required combinations for load cases.

3.4 Loads

Various load types are available for basic load cases. Most common are acceleration load cases where you accelerate (gravity, drive or slewing acceleration) mass cases. You can also apply node and beam loads as well as other loads. See image below for an overview of load types.



Drag the load cases from the tree and drop them to the graphics display to visually check the loads. To switch off the loads display select “minimal” from the saved display settings.



Basic mass cases can be combined to combination mass cases.

Logic load cases apply a permutation logic to a set of load cases, such as “exactly one”, “one or none”, “all possible combinations” or “all”.

In the example the logic load case “LC H” is the topmost load case that we use for result evaluation. It includes dead load (always), drive acceleration (+/-), Live load (all possible permutations) and slewing acceleration (+/-). This load case expands to 12 different permutations.

```
LC H
   1.0 * DL Total  BLC
   1.0 * Driving +/-  LLC  Exactly One
      (+/-)1.0 * Drive Accel  BLC
   1.0 * LiveLoad  LLC  One or None
      1.0 * LiveLoad   1  BLC
      1.0 * LiveLoad   2  BLC
   1.0 * Slewing +/-  LLC  Exactly One
      (+/-)1.0 * Slewing Accel  BLC
```

3.5 Kinematics, Orientations, Situations

With “kinematics” you can move subsystems relative to each other, e.g. luffing of the boom. The defined kinematics are used in “orientations” to achieve a certain working position. “Orientations” are the geometric definition of the working position.

“Situations” are important objects as they define the working state of the structure for the solver. “Situations” include an “orientation” and eventually a list of active load cases and a list of inactive beams. “Situations” are calculated by the solver and are used in evaluation pattern for results and proofs.

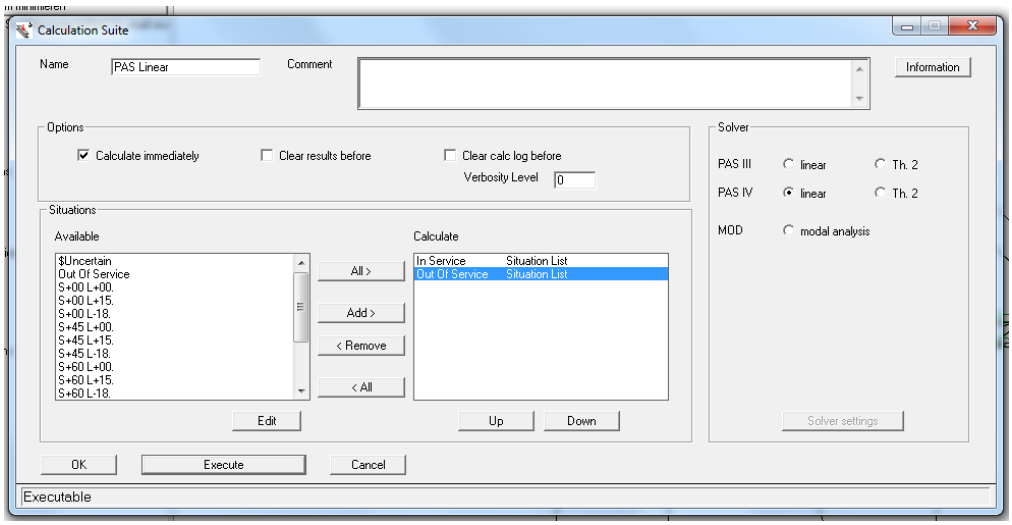
The following table shows the situations used for the model.

situation	slewing [°]	luffing [°]
Out of service	0	80
S+00 L+00	0	0
S+00 L+15	0	15
S+00 L-18	0	-18
S+45 L+00	45	0
S+45 L+15	45	15
S+45 L-18	45	-18
S+60 L+00	60	0
S+60 L+15	60	15
S+60 L-18	60	-18
S-45 L+00	-45	0
S-45 L+15	-45	15
S-45 L-18	-45	-18
S-60 L+00	-60	0
S-60 L+15	-60	15
S-60 L-18	-60	-18

3.6 Solving the model

“Calculation suites” manage the solvers. In calculation suites you can select the situations to solve, the solver and the theory. In the example the calculation suite “PAS linear” is prepared to solve all situations with solver “PAS IV” with linear theory.

To solve the model expand the “Calculation suite” branch in the object tree, click with the right mouse button on “PAS Linear” and choose “Apply”.



After the solver run is done please check the calculation log for warnings and errors. You can find the calculation log in the main menu: “Calculation” > “Show log file…”

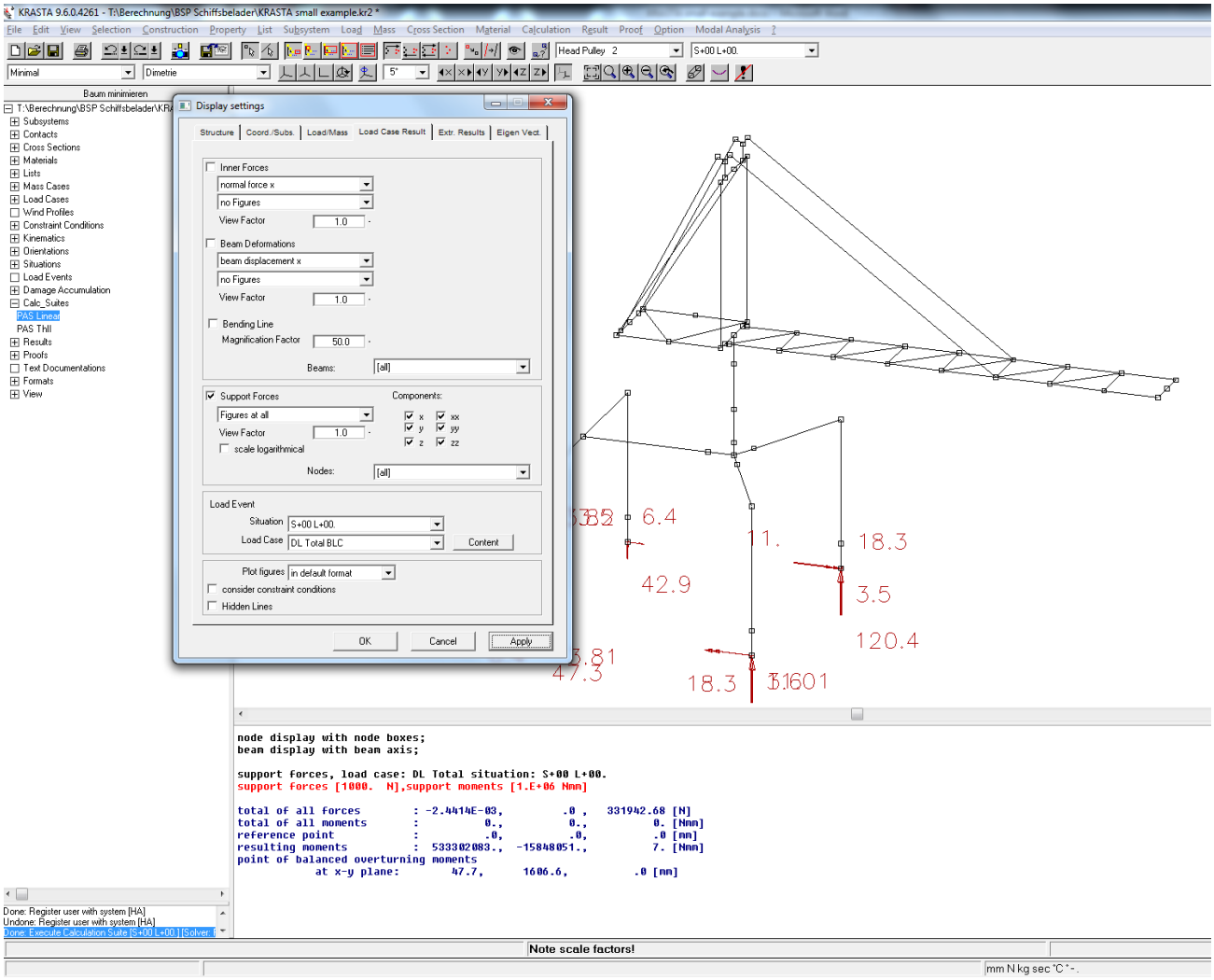
3.7 Result evaluation

3.7.1 Results for basic and combination load cases

Results for basic and combination load cases can be switched on in the “display setting” dialog. The following results are available.

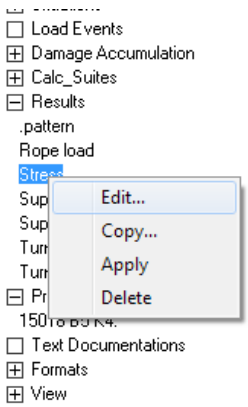
For beams: Inner forces, beam deformations, bending line
For nodes: support forces

Logic load cases are not available in this dialog, as they usually expand to more than one load permutation.



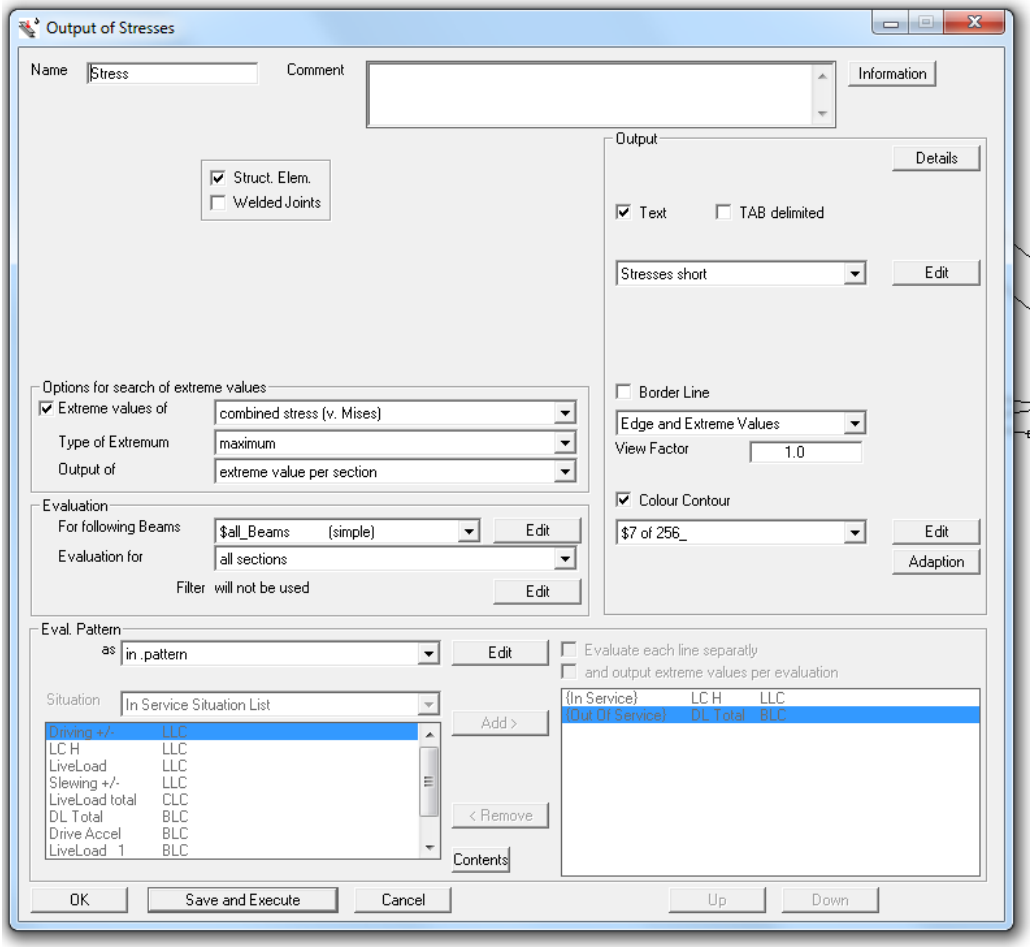
3.7.2 Result controls and proofs

Result controls and proofs are available in the object tree. Some have already been prepared. Edit one to open the dialog.

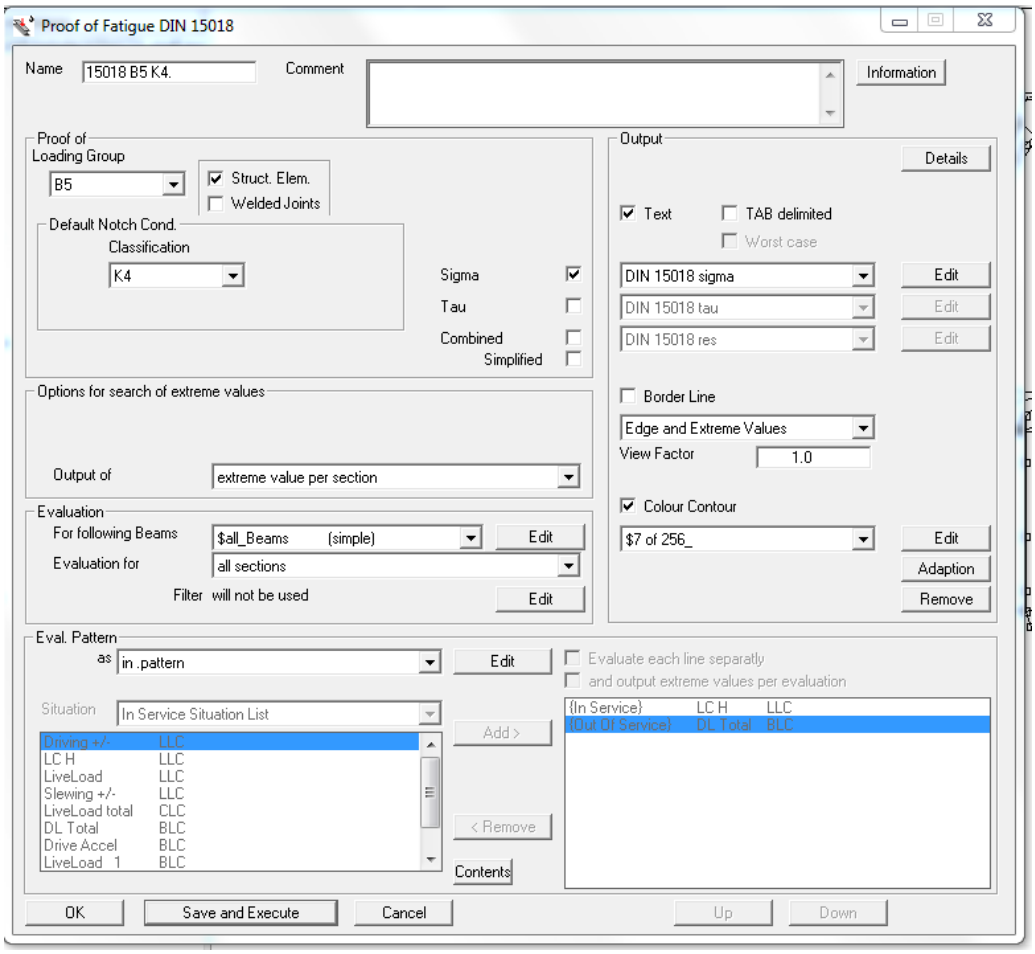


On the left side of the dialog you can switch on the search for extreme values and select a list of beams/nodes for which the results should be evaluated. On the right side you can control the output. Output is possible as text results, border line or color contour.

In the lower part you define the evaluation pattern. The selected situations/loads are displayed on the right side. Other result controls can be selected as a prototype.



The dialogs for proofs basically look the same. As proofs are standard specific, standard dependent controls can be found in the top left of the dialog. For proofs of fatigue you usually find “loading groups” and “notch cases” here.

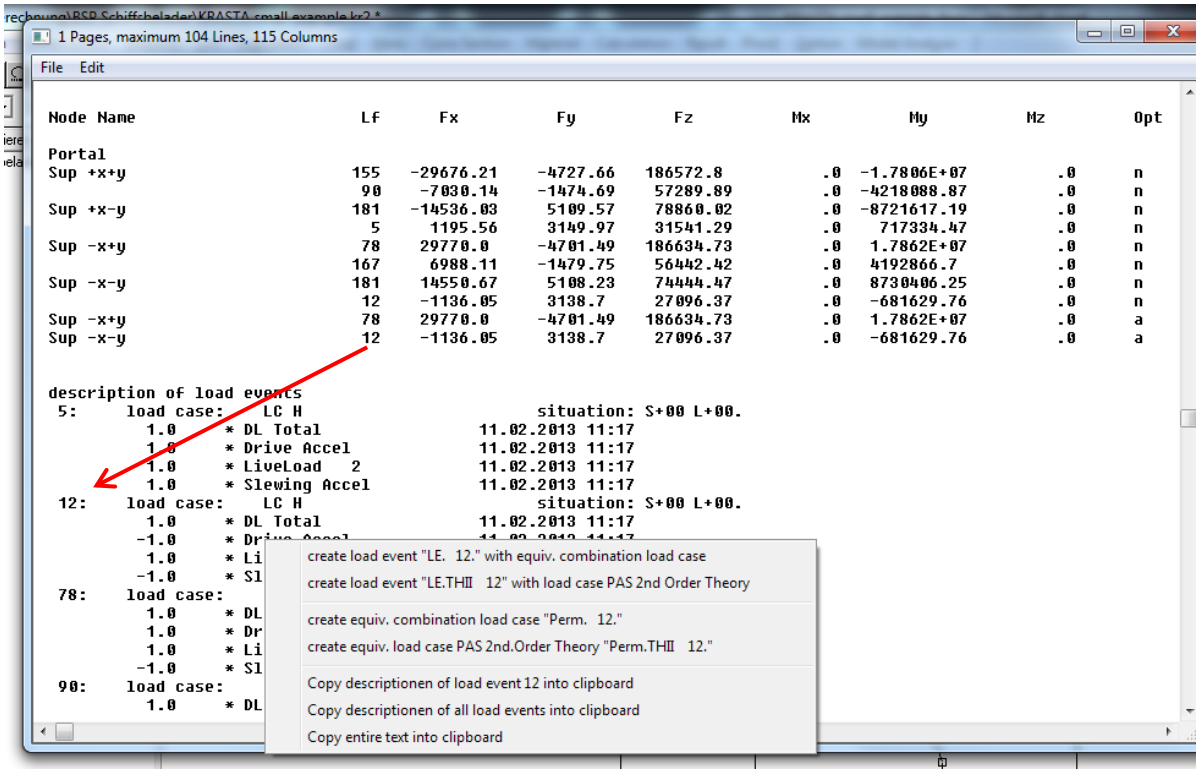


In the example we evaluate load case “LC H” in 15 “in service” situations and dead weight only for the “out of service” situation. Altogether we evaluate 15*12+1 = 181 load permutations.

The text output usually consist of

- a header, with description of options and result items
- a result table
- a list of permutations with which you can identify governing load permutations

When you “right click” on a permutation you can create according load events or combination load cases for further evaluation of this permutation.



If you choose “TAB delimited” text output you can copy and paste the text output to a spread sheet live MS Excel to process the results externally.