KRASTA

Telescopic Crane System

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 4
3.4 LOADS 4
3.5 SUBSYSTEMS 6
3.6 ORIENTATIONS 9
3.6.1 General 9
3.6.2 Telescoping 10
3.6.3 Luffing 12
3.7 KINEMATICS 13
3.7.1 Luffing 13
3.7.2 Slewing 15
3.8 WORKING POSITIONS 16
3.9 SITUATIONS 17
3.9.1 Situation lists 18
3.9.2 Situation depended Mass Case 19
3.10 SOLVING THE MODEL 20
3.11 RESULT EVALUATION 21
3.11.1 Results for basic and combination load cases 21
3.11.2 Result controls 22

Kühne BSB GmbH
Stahlbau und Fördertechnik

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

Date: 08.08.2014
Pages: 22

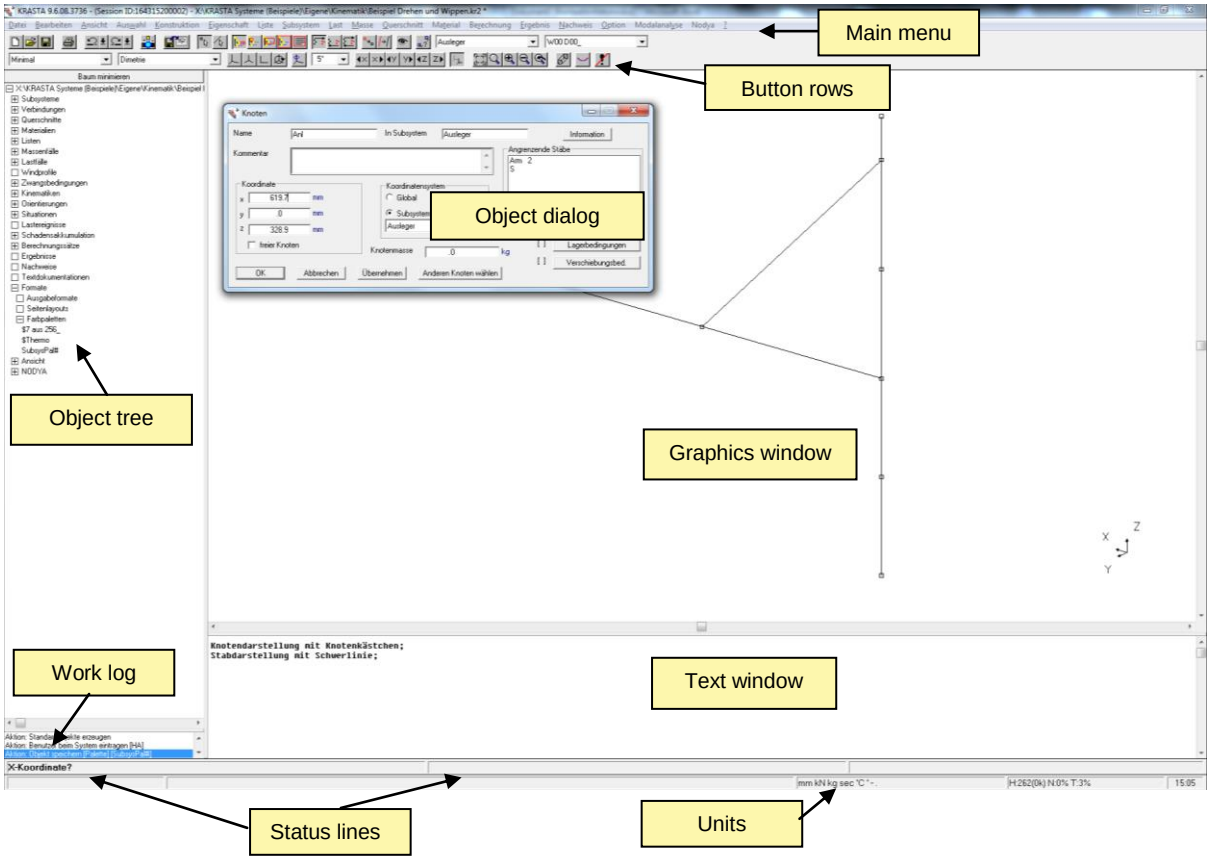
1 Preface

The provided KRASTA model „telescopic boom“ is a virtual model of a trailer crane. 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 9.7. 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

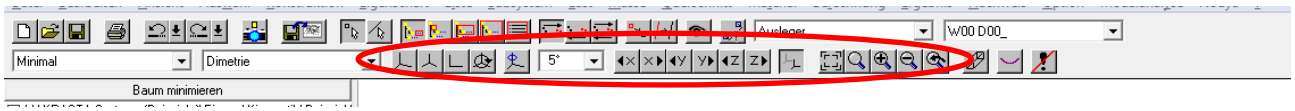
2.4 Sizes of text and graphic elements, colors

The graphic display can be customized. Main menu:

Display > Screen
Display > Colors

2.5 Changing the model view

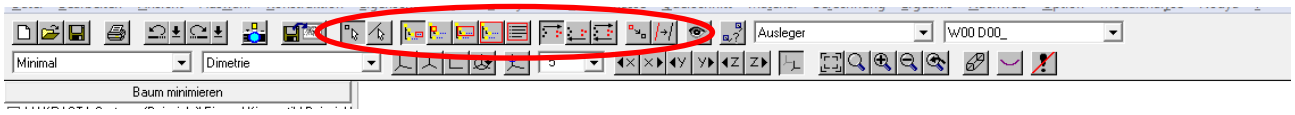
You may rotate the display of the graphics window with the display buttons in the second button row.



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.

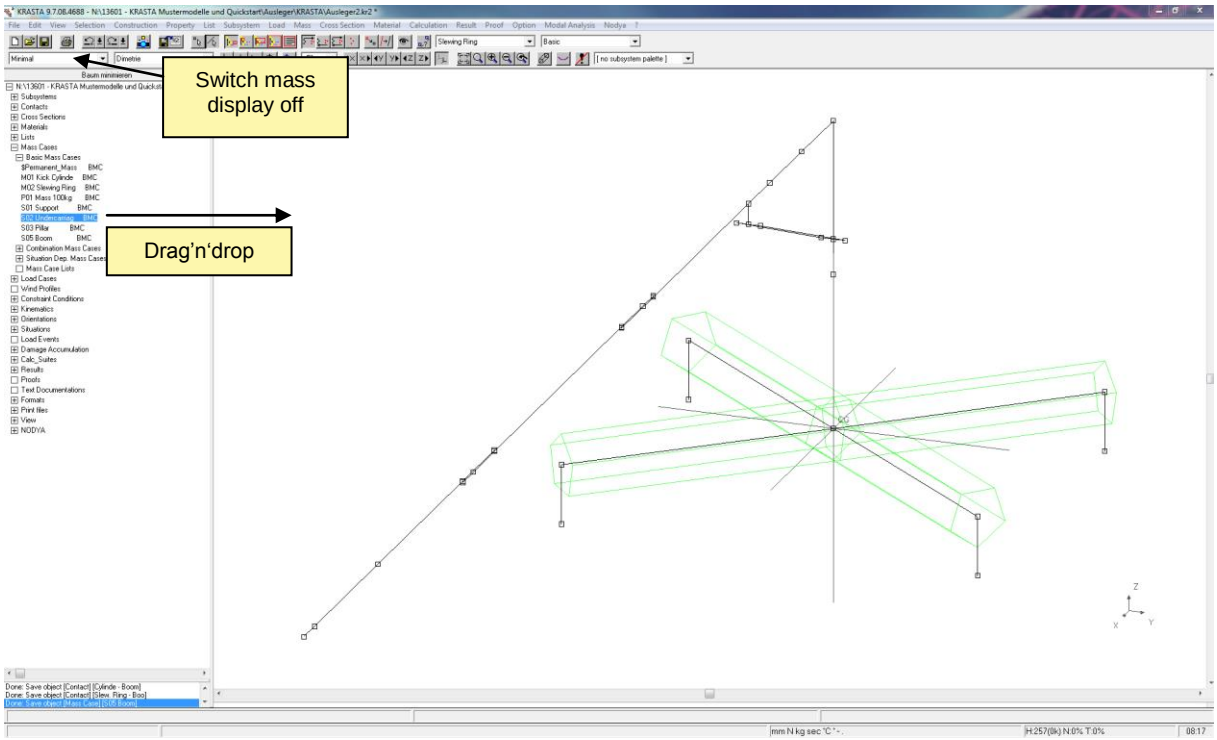
3.2 Exploring the Geometry

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
Arrow – Arrows from beam start to beam end are drawn
Arrow_Aux vect – Arrows and auxiliary vectors are drawn
Cross sections – Cross sections are drawn on the beams
Joints – Joints are drawn
Support Conditio – node support conditions are drawn

3.3 Masses

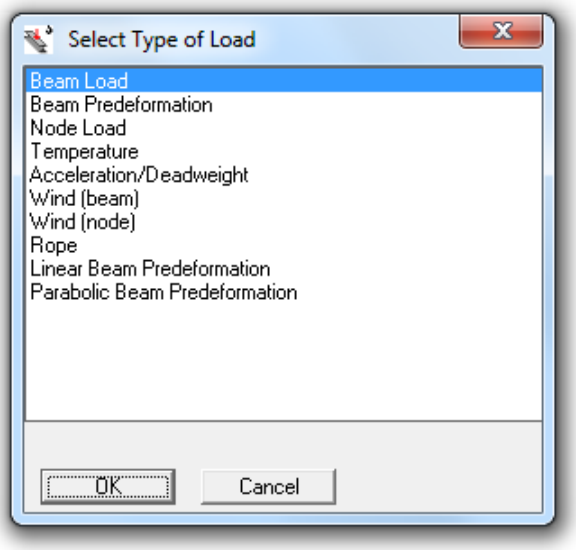
Masses are entered as “basic mass cases” where you can put node/beam masses on nodes/beams or apply a mass factor to the structural mass of (parts of) the structure. The masses are usually taken from parts lists or are estimated from earlier projects. To see the mass distribution and the sum drag the 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.



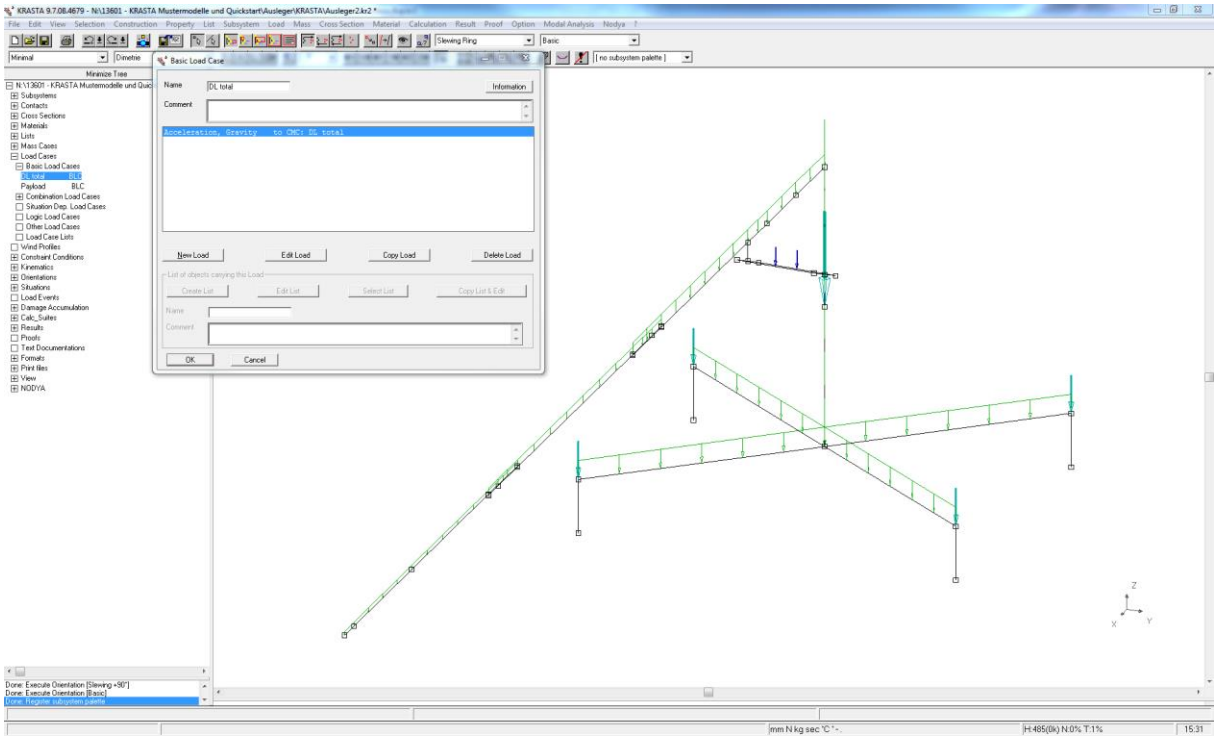
Basic mass cases are combined into combination mass cases which depend on the required combinations for load cases.

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.



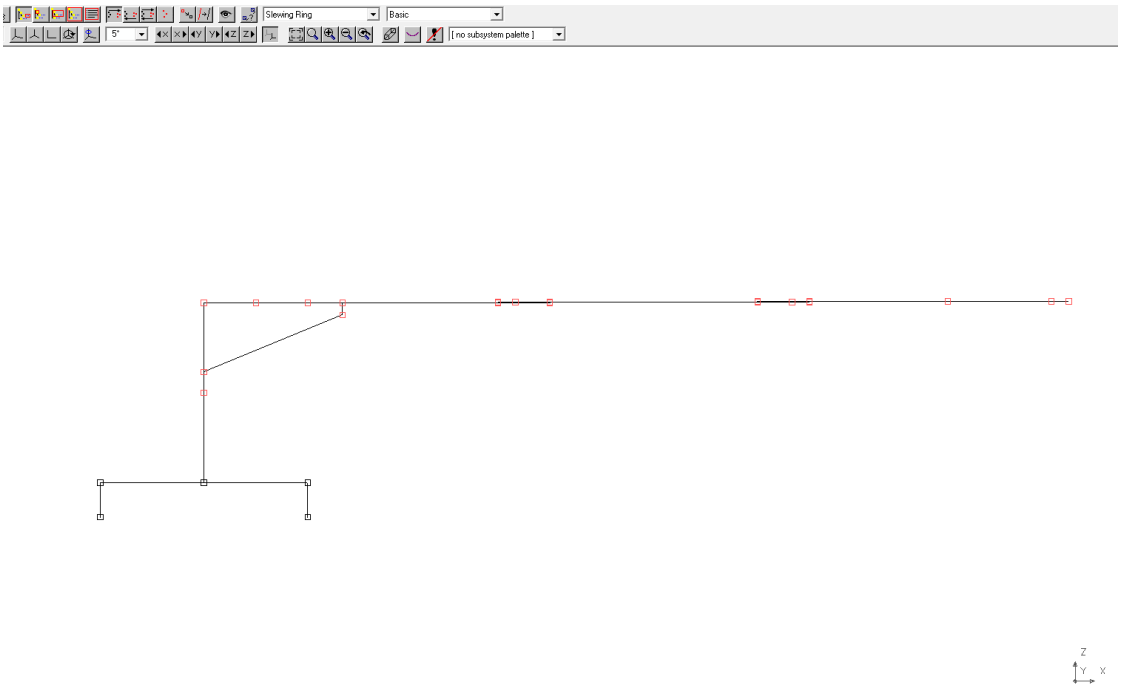
Basic load cases can be combined to combination load cases.

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”.

3.5 Subsystems

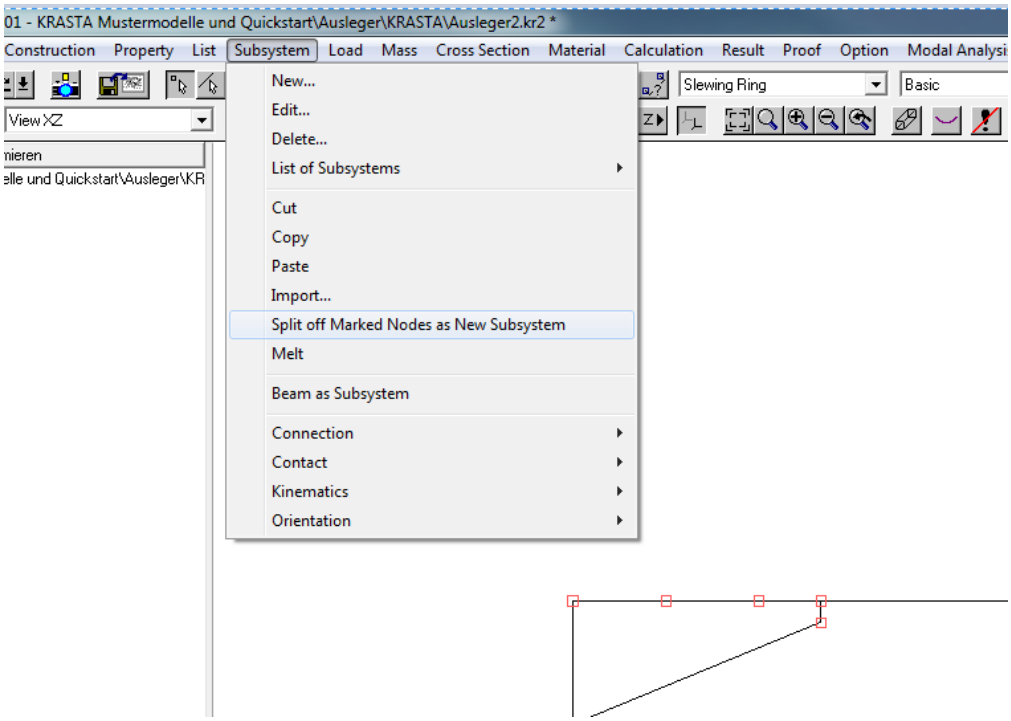
The model is divided into different subsystems. Subsystems are individual subsets of beams and nodes which can be moved relative to each other by using kinematics or switching contacts.

To create a subsystem mark all nodes which should belong to the new subsystem. The correct control hierarchy in the “Subsystems”-object tree is automatically created, if the subsystem is split in the relevant order which depends on the telescopic crane movement.

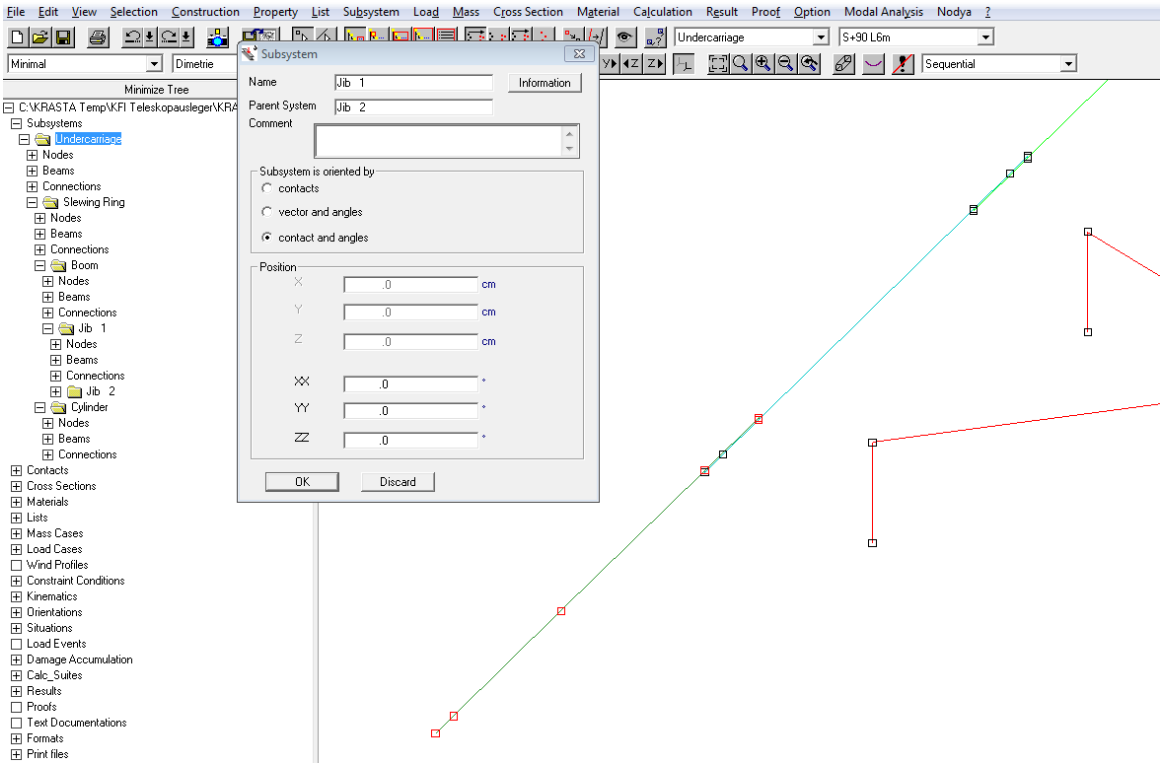


For separating select “Subsystem” in the Main Menu and choose “Split off Marked Nodes as New Subsystem”.

Subsystem > Split off Marked Nodes as New Subsystem

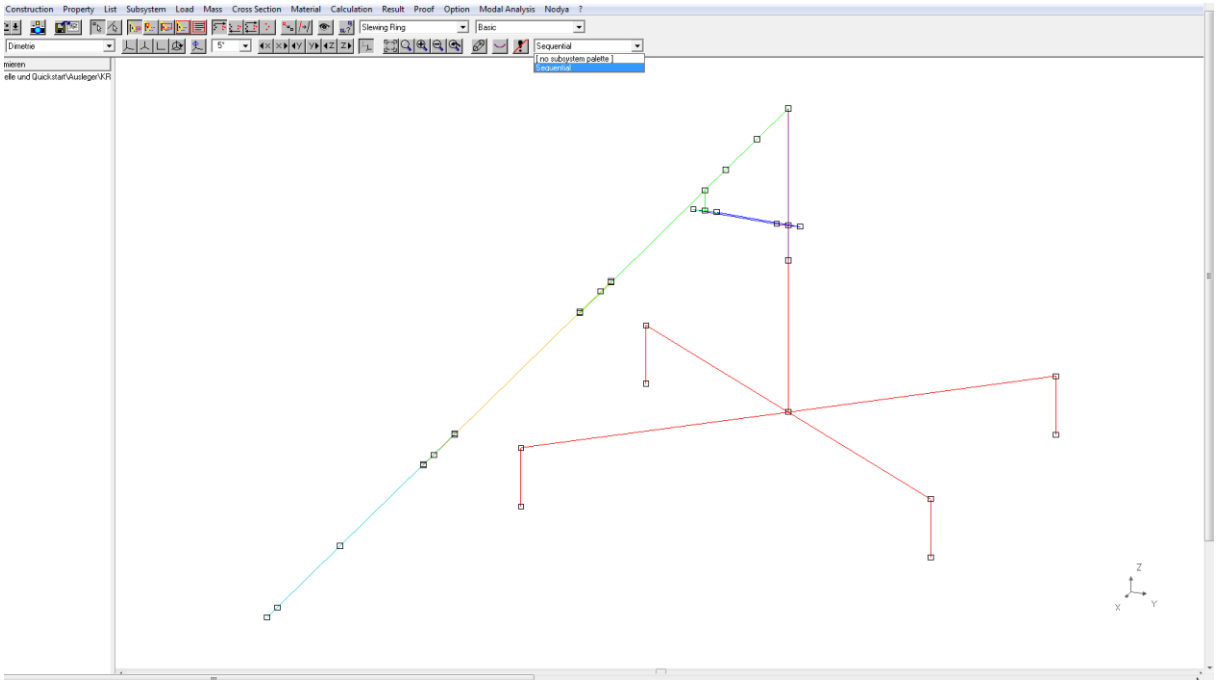


In the window “Subsystem” name the new subsystem for better navigation and orientation. The Parent System is automatically filled with the previous subsystem.



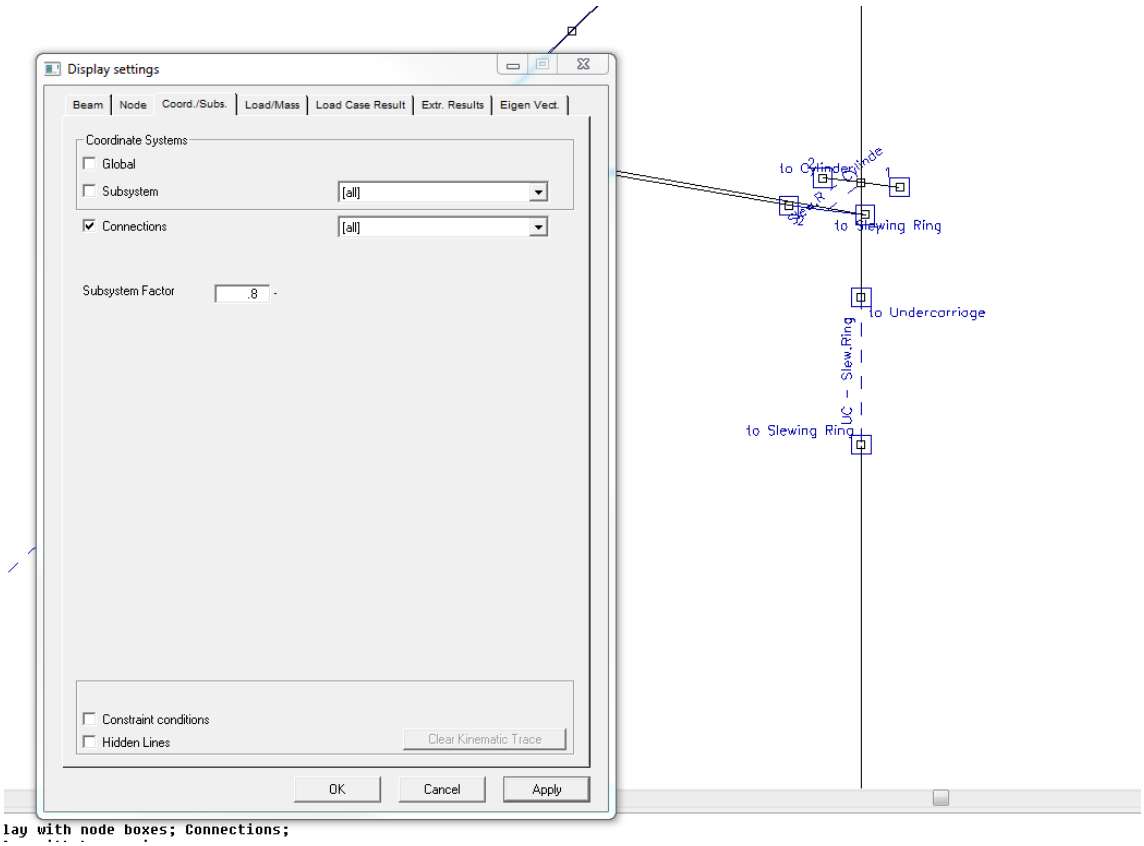
To see the subsystem in different colors choose “Sequential” in the right selection box in the second button row.

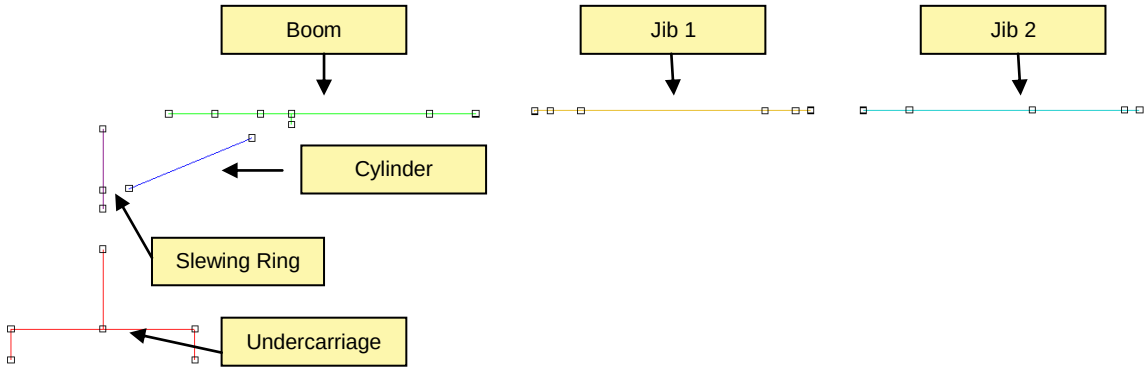
The subsystems are now displayed in different colors.



After “Split off Marked Nodes as New Subsystem” automatically contacts and connections are formed. They can be edited in the object tree. It is recommended to name them immediately after separating and according to their purpose.

The model consists of 6 different subsystems. To get an idea on the different subsystems open the display settings dialog and choose the “Coord./Subs.” register. To space the subsystems select a “Subsystem Factor” smaller than 1 (0.9 will do in most cases) and apply. To take a look at the contacts and the connections take a check mark on “Connections” and apply.





3.6 Orientations

3.6.1 General

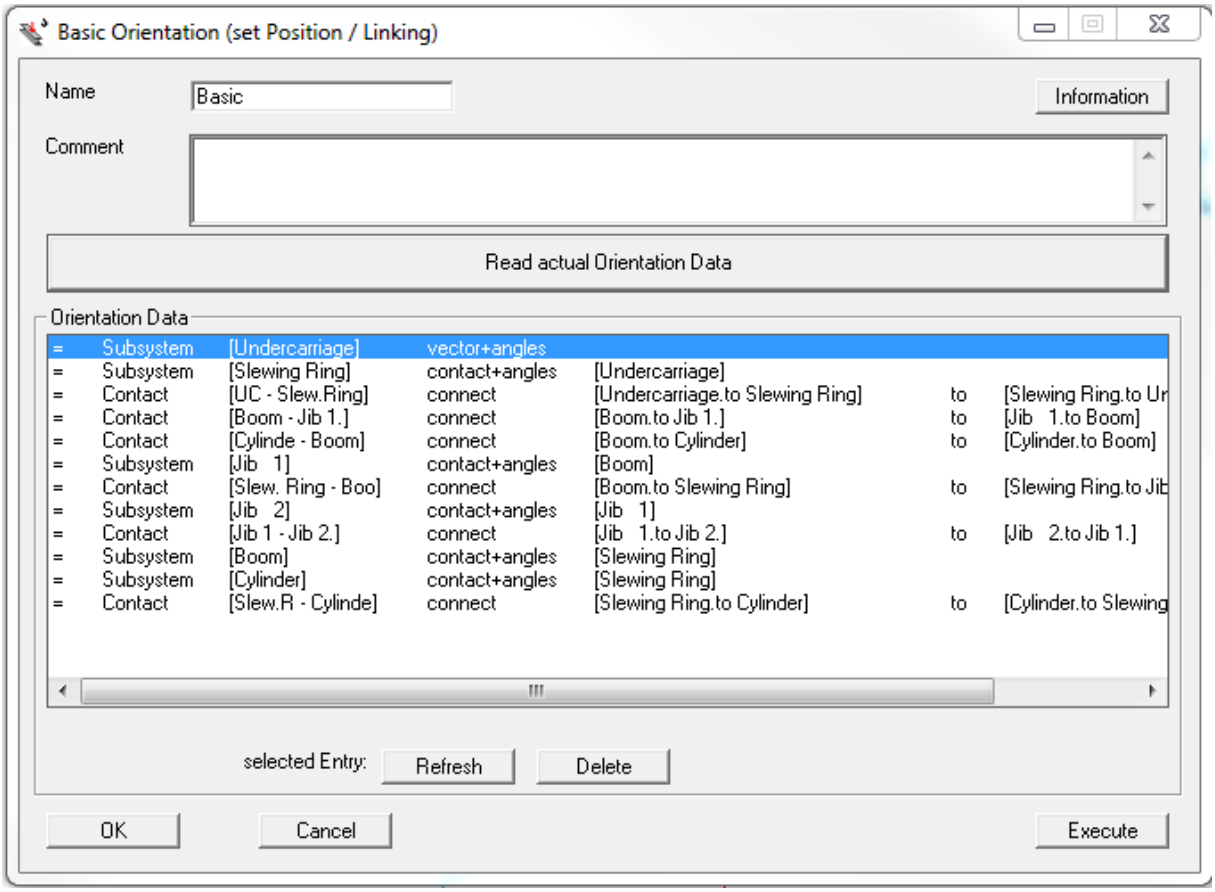
“Orientations” are the geometric definition of the working position. There are three types of orientations: “Basic Orientation”, “Relative Orientation” and “Orientation Modification”.

In the “Basic Orientation” a set of orientation information is stored. When you create a basic orientation the current state of the model is recorded as a kind of snapshot. The following information is stored:

1) Definition and values of individual subsystem orientations
2) As a result of 1) the subsystem hierarchy in the object tree
3) Connections used in contacts

Eventually this information needs to be updated if the model is modified.

In the basic orientation dialog an equal sign at the beginning of each entry indicates that the entry definition is valid for the current model state.



A “Relative Orientation” is an ordered set of orientation changes that are applied to a defined start orientation.

“Orientation Modifications” are only modification changes without a defined start orientation.

Working positions can be represented in KRASTA by basic orientations only. In this case modifications are applied manually and required basic orientations are created manually if achieved. While this may be an easy approach in the first place, revision of the model may be quite time consuming because all working positions may need individual manual updates.

In this example only one basic orientation exists. Individual movements and targets are defined as orientation modifications as a kind of building blocks for relative orientations. In this case every target value is only entered once in the model, which leads to a model that is easily maintained if target changes are required.

3.6.2 Telescoping

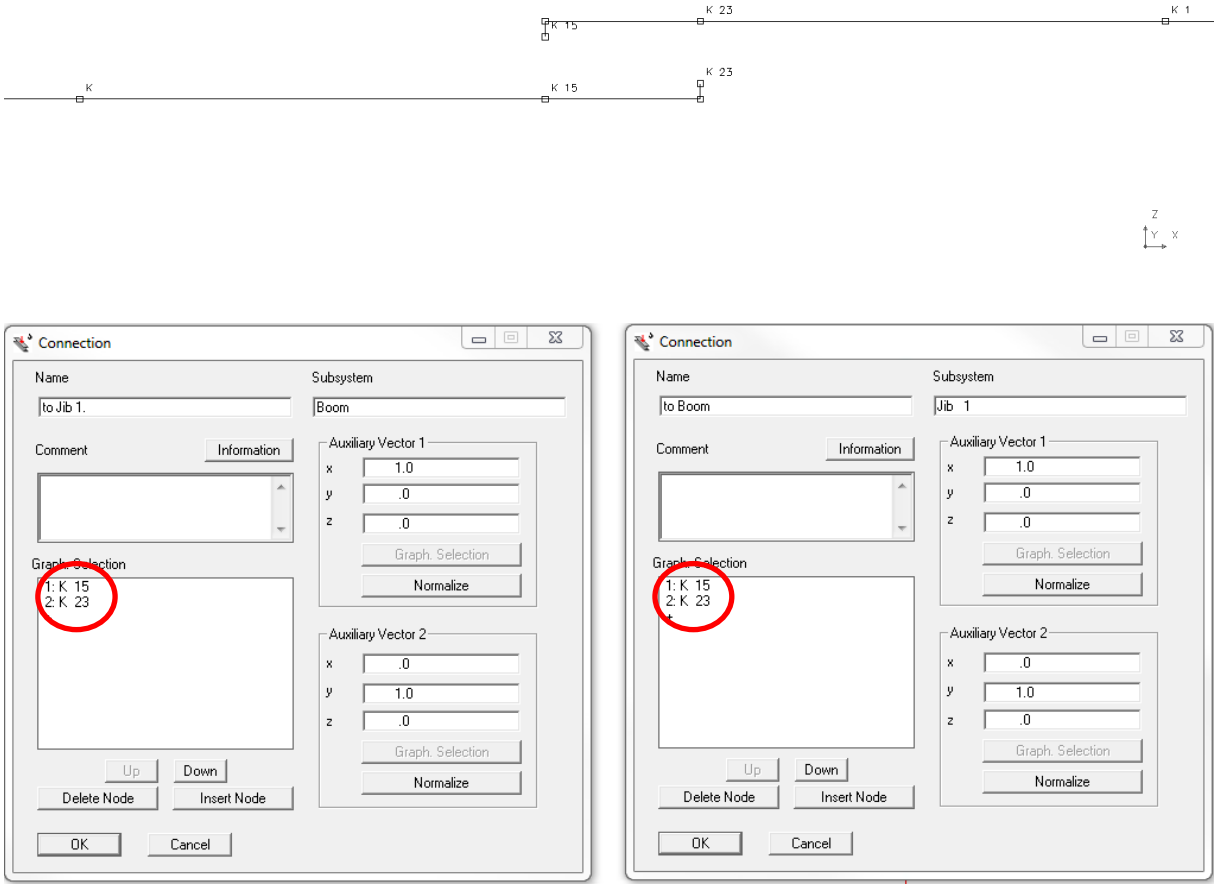
Telescoping requires different sets of compatible connections for the telescoping jibs. In the example each connection consists of two nodes. Compatible connections require an equal distance between the connection node pairs.

There is only one contact that connects the two parts. The connections used in the contact are changed as required. Multiple contacts between two parts with contradicting connection pairs may not be created as contradiction connection information cannot be achieved at once.

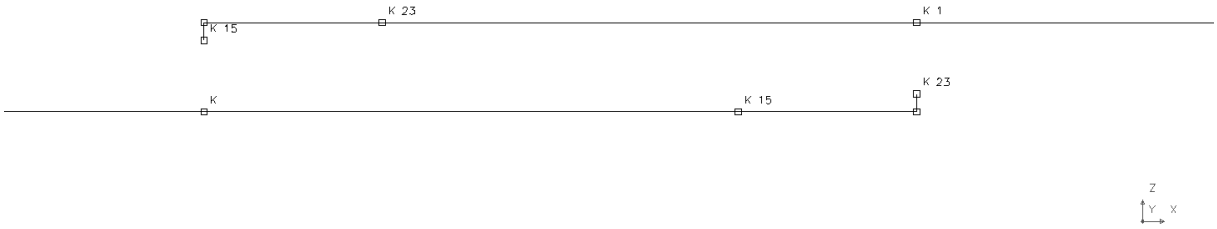
It is recommended to name connections and contacts according to their purpose.

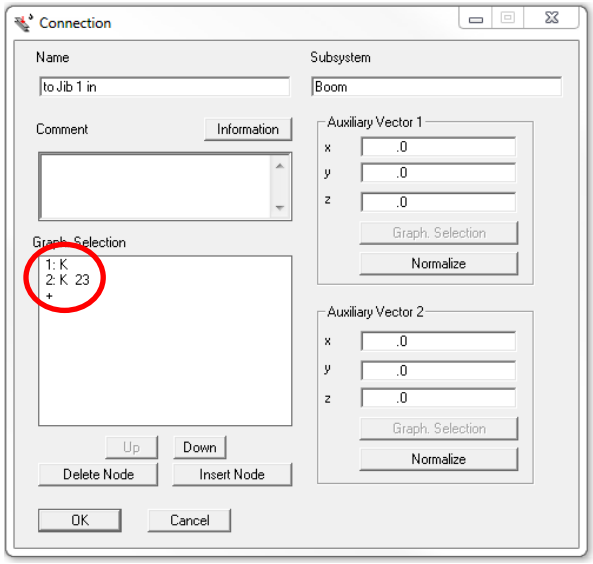
The order of nodes within the connection definition must be in the right order. If the nodes are in the wrong order Jib 1 will be reversed pointing backwards.

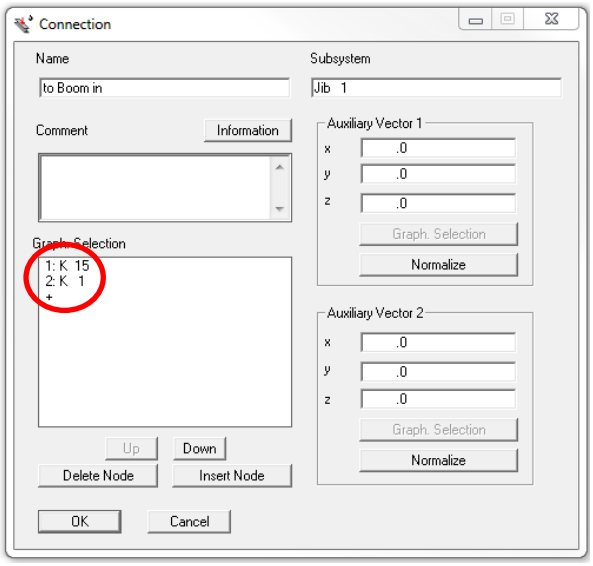
In the extended state the boom and jib 1 are connected by the nodes K15 and K23.



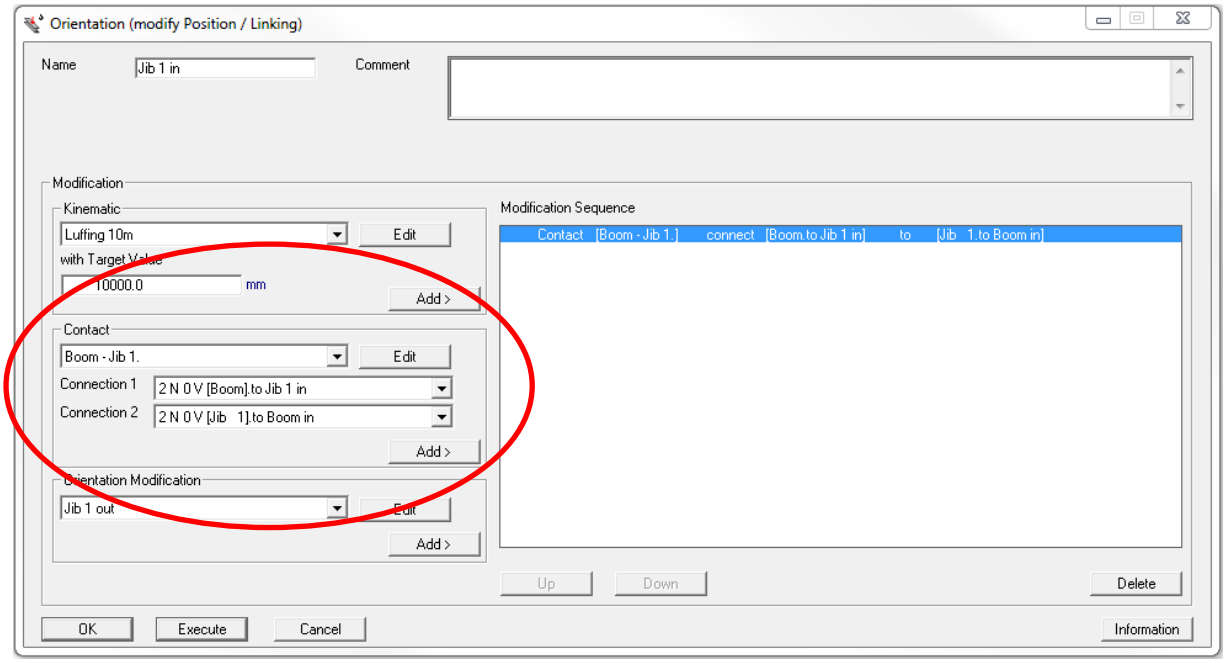
In the retracted state the nodes K and K23 of the boom are connected to the nodes K15 and K1 of jib 1.







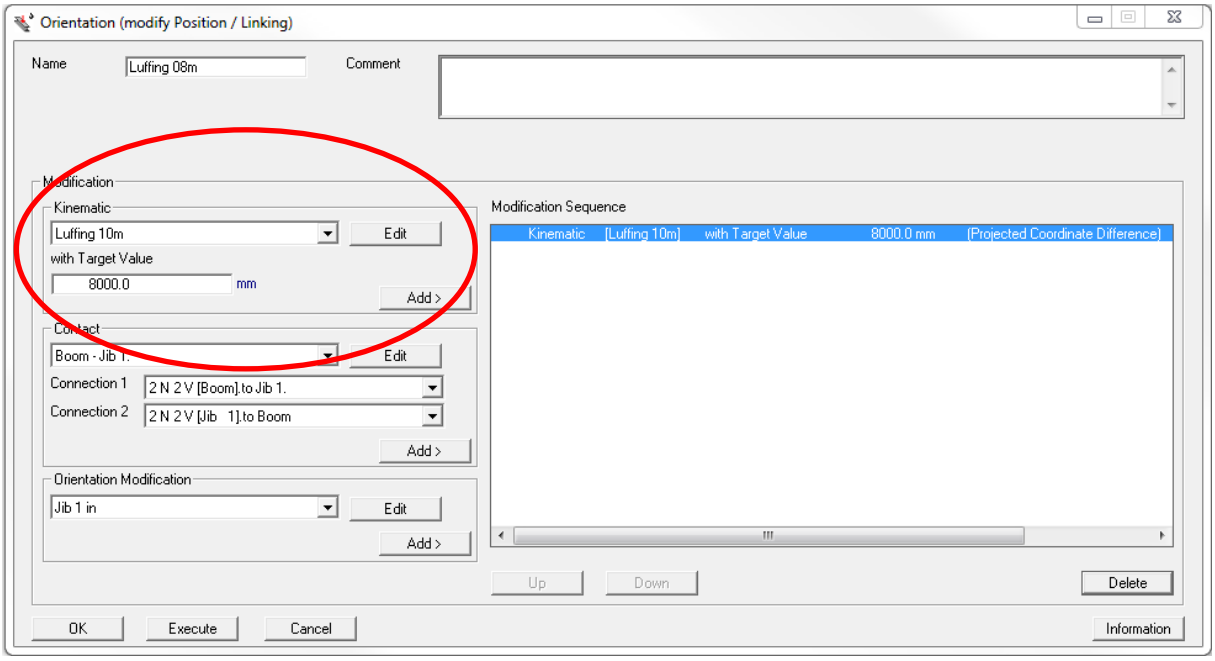
After the required connections are created the contact can be changed manually to test the connections. If the manual test is passed, orientation modifications are created for each jib and each telescopic position.



For retracting jib 1 into the boom the contact “Boom – Jib 1” and the according connections are set. The settings have to be added to the modification sequence.

For extending jib 1 the “Orientation Modification” is created accordingly.

3.6.3 Luffing



For the luffing a kinematic with target value 10m is already available. Within orientation modifications and relative orientations kinematics can be “recycled” with new target values. In the example we use the kinematic “Luffing 10m” with a target value 8m.

For each luffing target value one orientation modification is created.

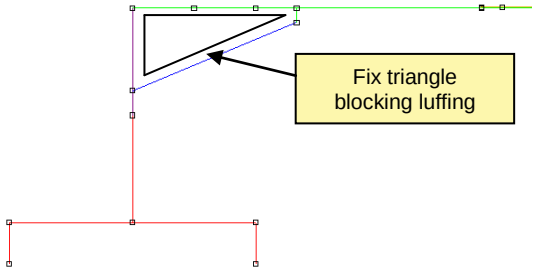
3.7 Kinematics

With “Kinematics” subsystems can be moved relative to each other, e.g. luffing of the Boom. In step 3.6 a basic orientation is created. The defined kinematics are used in “orientations” to achieve a certain working position. “Orientations” are the geometric definition of the working position.

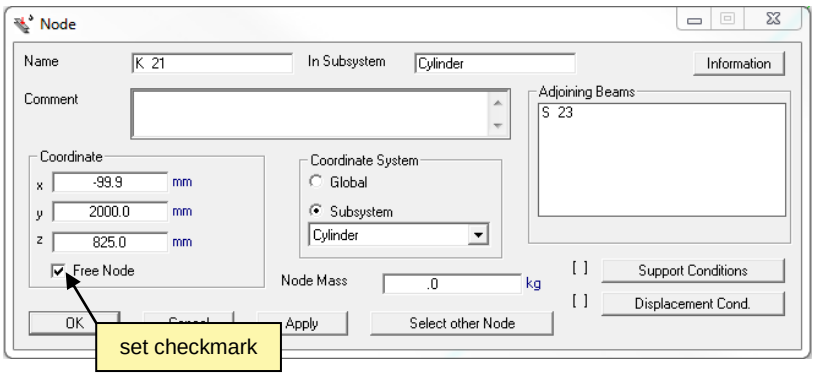
This example shows how to create kinematics for luffing and for slewing.

3.7.1 Luffing

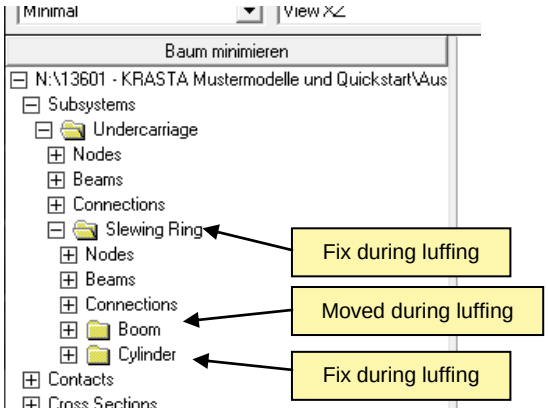
As we have a triangle between slewing ring, boom and cylinder luffing is not possible without further ado. Luffing of the real crane is done by changing the length of the cylinder. In KRASTA we need to change the cylinder beam length as required for the working position. This is possible if cylinder nodes at the intersection between boom and cylinder are marked as “free nodes”. If required KRASTA may change coordinates of free nodes during subsystem movements.



In the example we have two cylinders. Both cylinder nodes (K 21 and K 22) at the intersection to the boom are set free.



The free nodes need to be set at the intersection between moved subsystem and fixed subsystem. If, other than in this example, the cylinder subsystem is child of the boom, the nodes at the intersection between cylinder and slewing ring need to be set “free”.

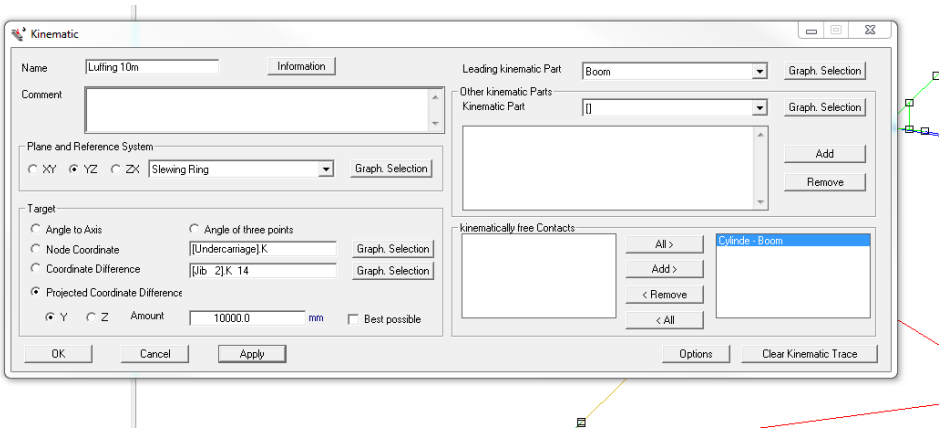


In the kinematics dialog, select the “leading kinematic part”, which should be the topmost subsystem in the subsystem hierarchy displayed in the subsystem branch of the object tree. Child subsystems of the leading kinematic part are moved together with the leading subsystem.

The leading kinematic part is moved in a plane that is defined by two axes of a reference subsystem. It is important to choose a reference subsystem in which the leading subsystem stays in the chosen working plane. Here, the slewing ring is turned against the undercarriage, moving the boom to different planes in the undercarriage coordinate system. The boom remains in the same plane of the slewing ring coordinate system, even if it is turned against the undercarriage. Thus, the slewing ring needs to be chosen as reference subsystem.

Within the reference plane various targets of the kinematic can be chosen. For the luffing of a boom usually an angle or a working radius is chosen. In this case the boom system is moved to a jib working radius of 10m. Two nodes have to be selected and the amount of the projected coordinate difference between them has to be entered. The difference is measured horizontally, which is the y axis.

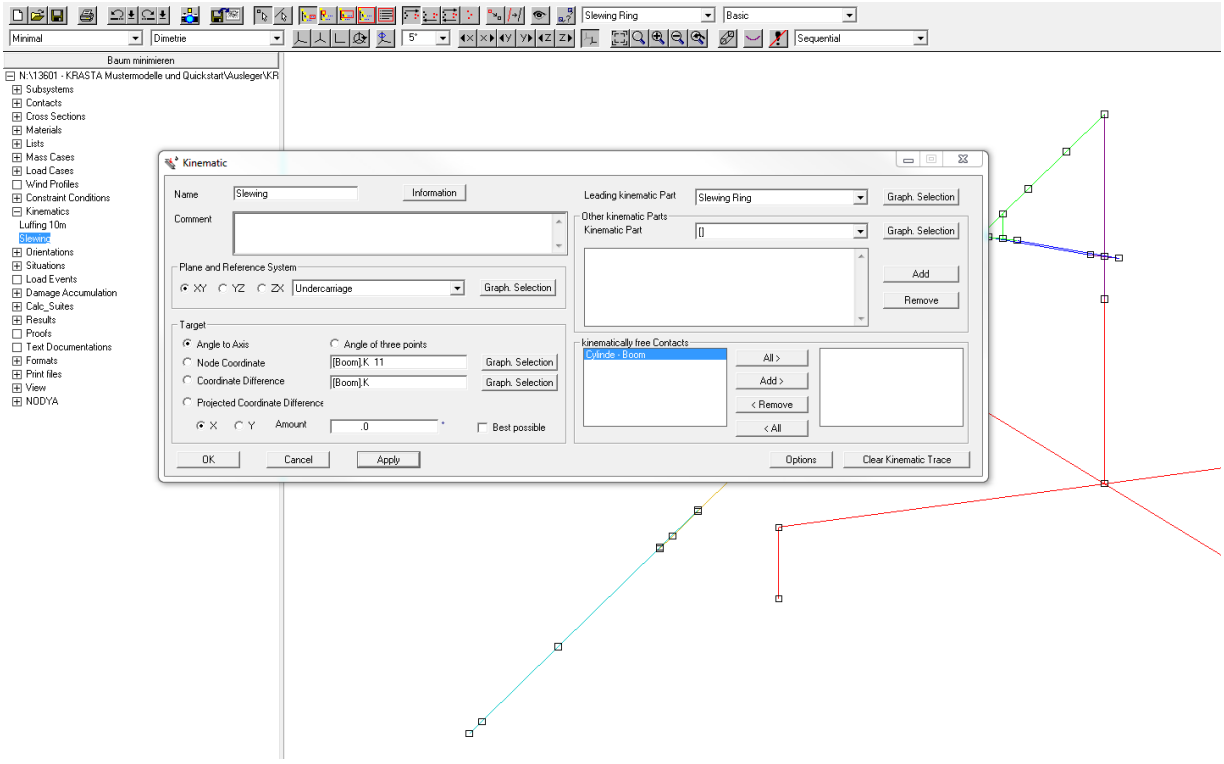
Contacts, in which all contact nodes are set as free nodes can be chosen as “kinematically free contacts”. Select the according contact as free.



To see the effect of the kinematic choose “Apply” within the dialog or drag the object from the object tree and drop it on the graphics window

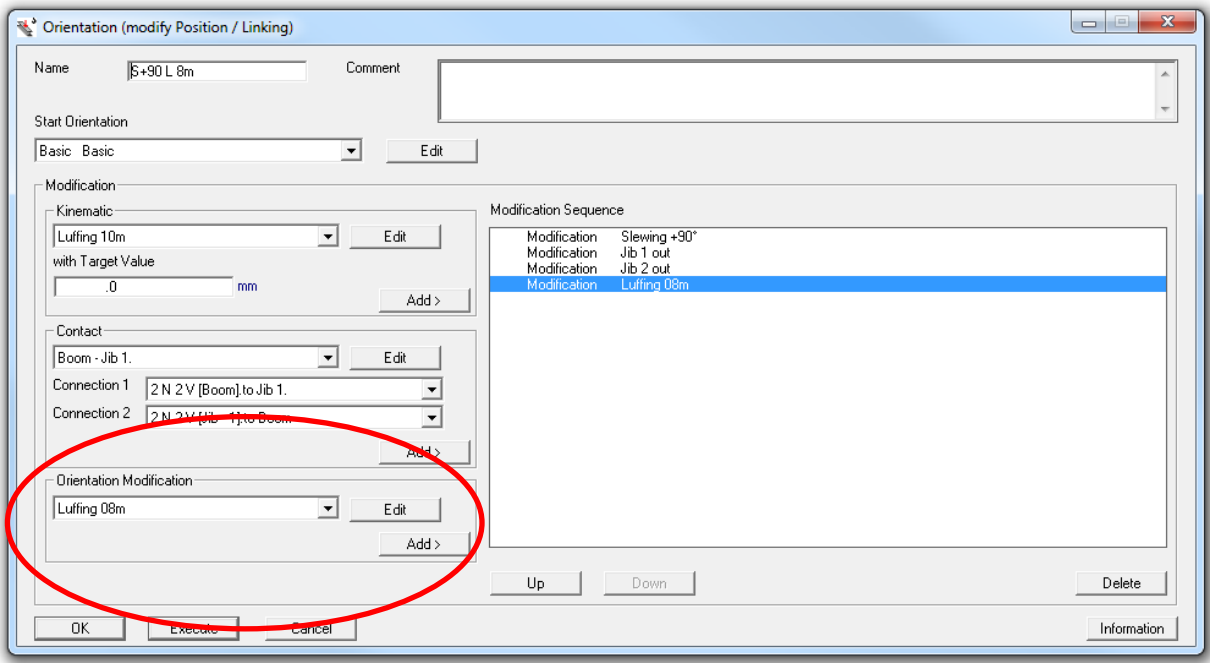
3.7.2 Slewing

For Slewing the XY-plane and the Undercarriage as the reference system are selected. The slewing ring subsystem is selected as the leading kinematic part. As Target the angle to axis is marked and two nodes on the x-axis are selected. The X-axis is selected as the basic orientation and a starting value of 0° is entered. For Slewing no kinematically free contacts are added.



3.8 Working positions

Working positions are assembled as relative orientations with a defined start orientation and orientation modifications containing individual subsystem movements and kinematic targets.



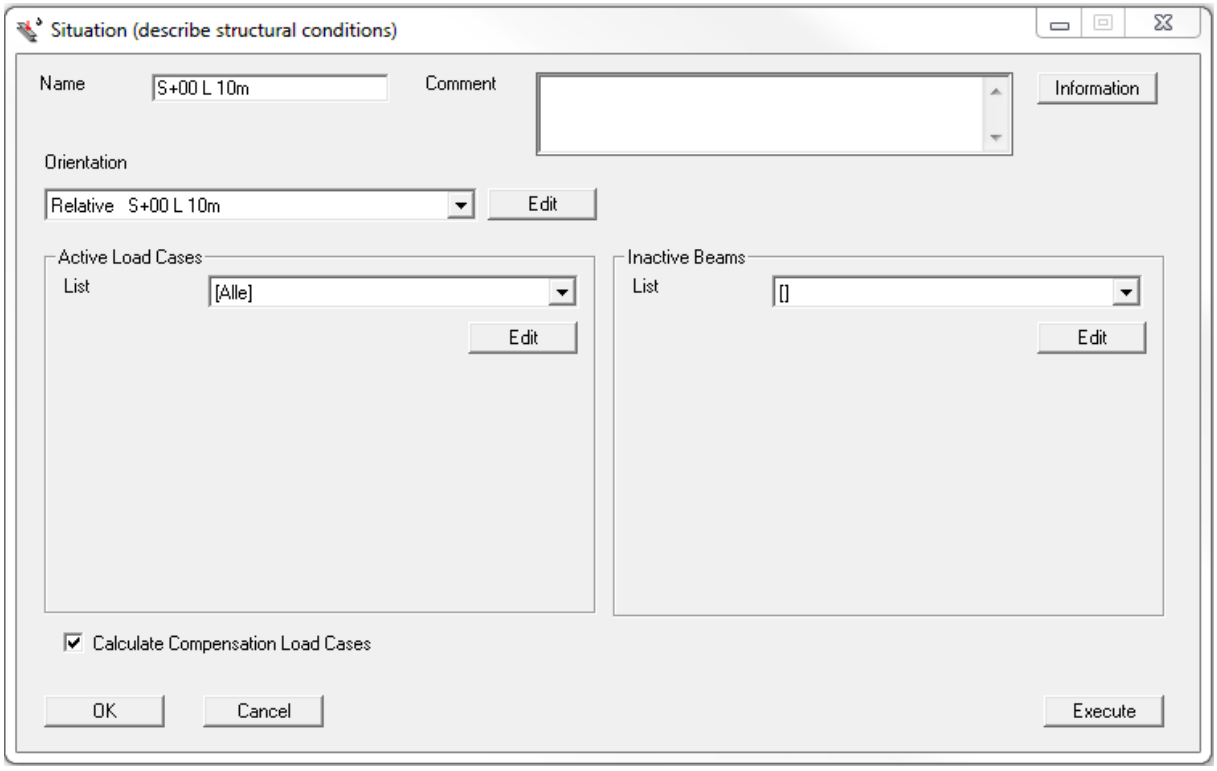
For the working position “Slewing 90°” and “Luffing 8m working radius” the orientation definition is shown above. The “Basic” orientation is set as start orientation. The modification sequence is:

1) Slewing +90°
2) Jib 1 out
3) Jib 2 out
4) Luffing 08m

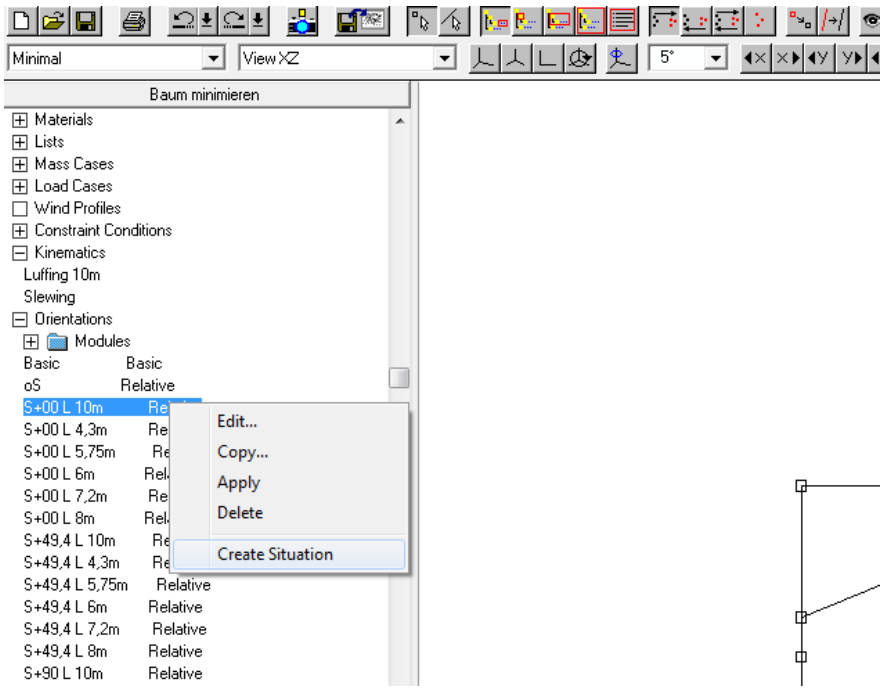
The modification list is applied one entry after the other, the order is usually important. In the example luffing requires the right telescoping situation. Thus, the jib modifications have to be set before the luffing modification.

3.9 Situations

“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.

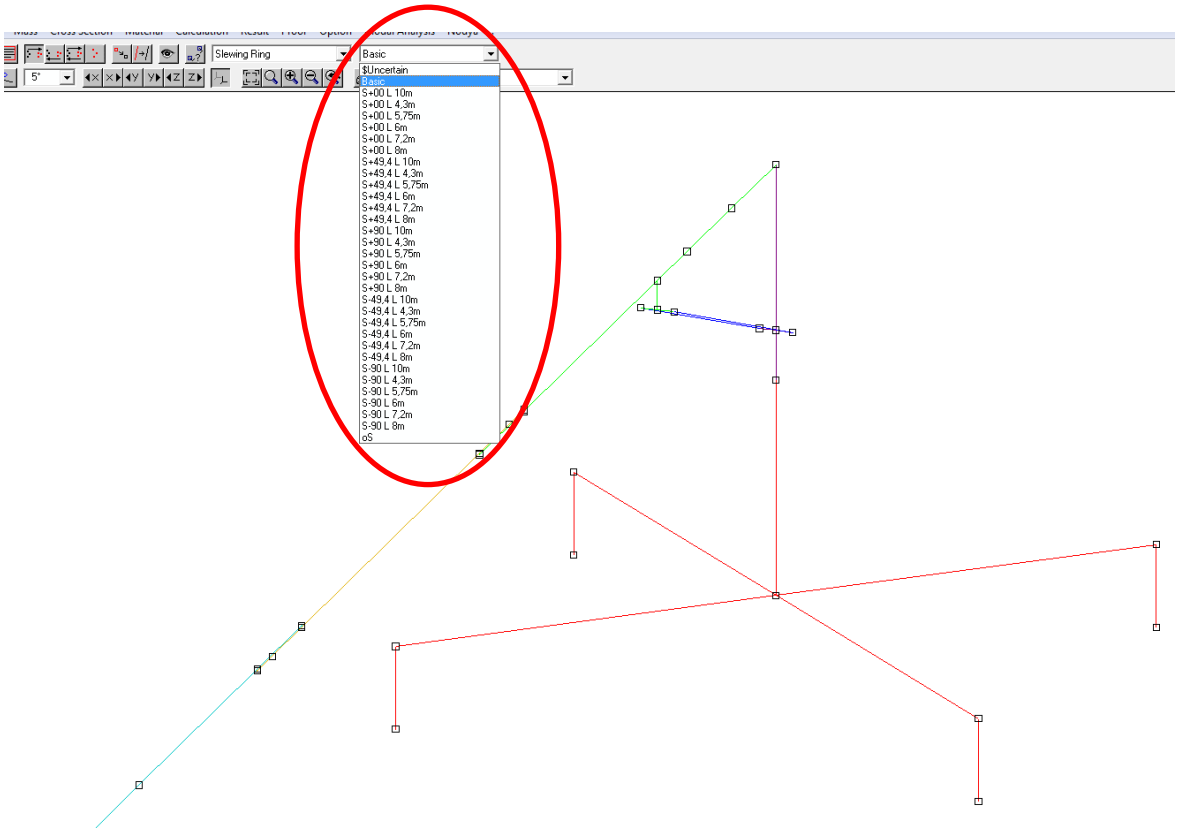


To create a situation you can create a new situation and select a relative orientation or just right click on an Orientation in the object tree and select “Create Situation”.



For mass creation of situations use the following menu item:
Calculation > (Situation) > Create for Orientations…

To visually check the situation you can switch through the situations with the top right selection box. Once this selection box has focused you may use the arrow up and down keys to move within the selection.



3.9.1 Situation lists

Situations can be grouped by situation lists to use them for situation dependent mass cases, calculation suites or result evaluation.

In the example there are situation lists for

- situations with similar payload
- in service situations
- out of service situations
- situations to be calculated by the solver

3.9.2 Situation depended Mass Case

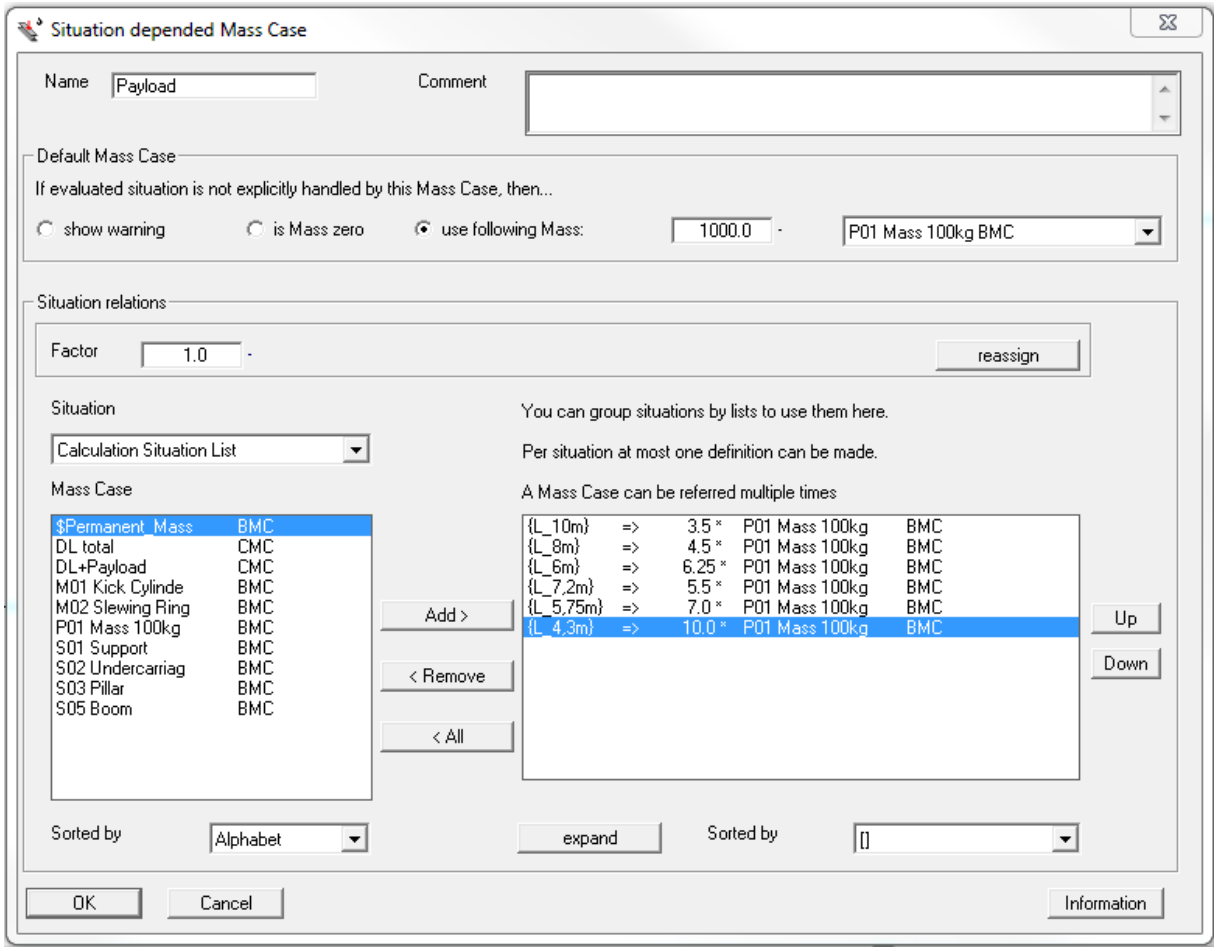
The payload mass should vary depending on the jib working radius. Therefor a situation depended mass case is created. A situation dependent mass case forks to different mass cases and factors depending on the situation that is evaluated.

A default mass case needs to be set. In this example a mass of 100.000 kg is used to notice immediately if the evaluated situation is not explicitly handled by this mass case.

A situation or a situation list is to be selected and added in combination with factor and a mass case. A situation or situation list can only be used once. A situation may be used implicitly multiple times via situation lists. In this case the definition is used that is first in the order of entries. A Mass case can be referred multiple times.

In this example the following payload masses are applied:

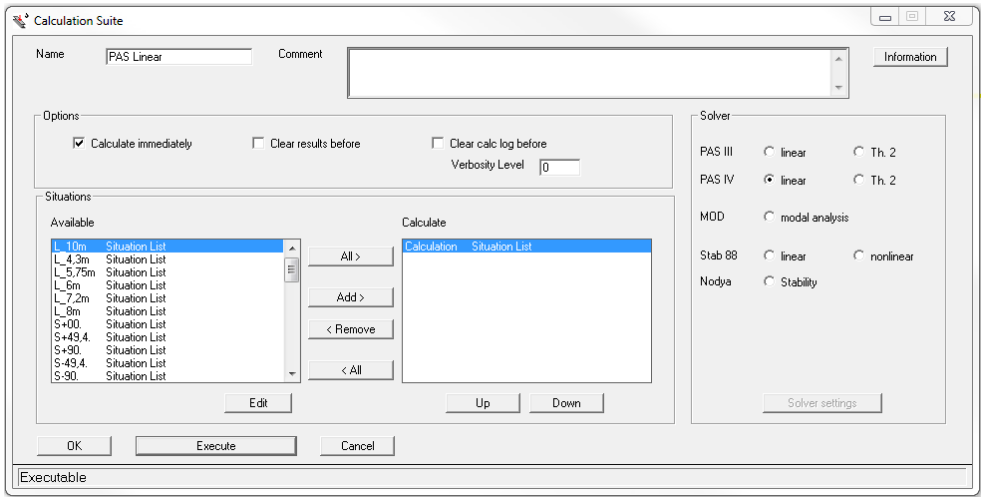
Working Radius [m]	Mass [kg]
10,00	350
8,00	450
6,00	625
7,20	550
5,75	700
4,30	1000



3.10 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…”

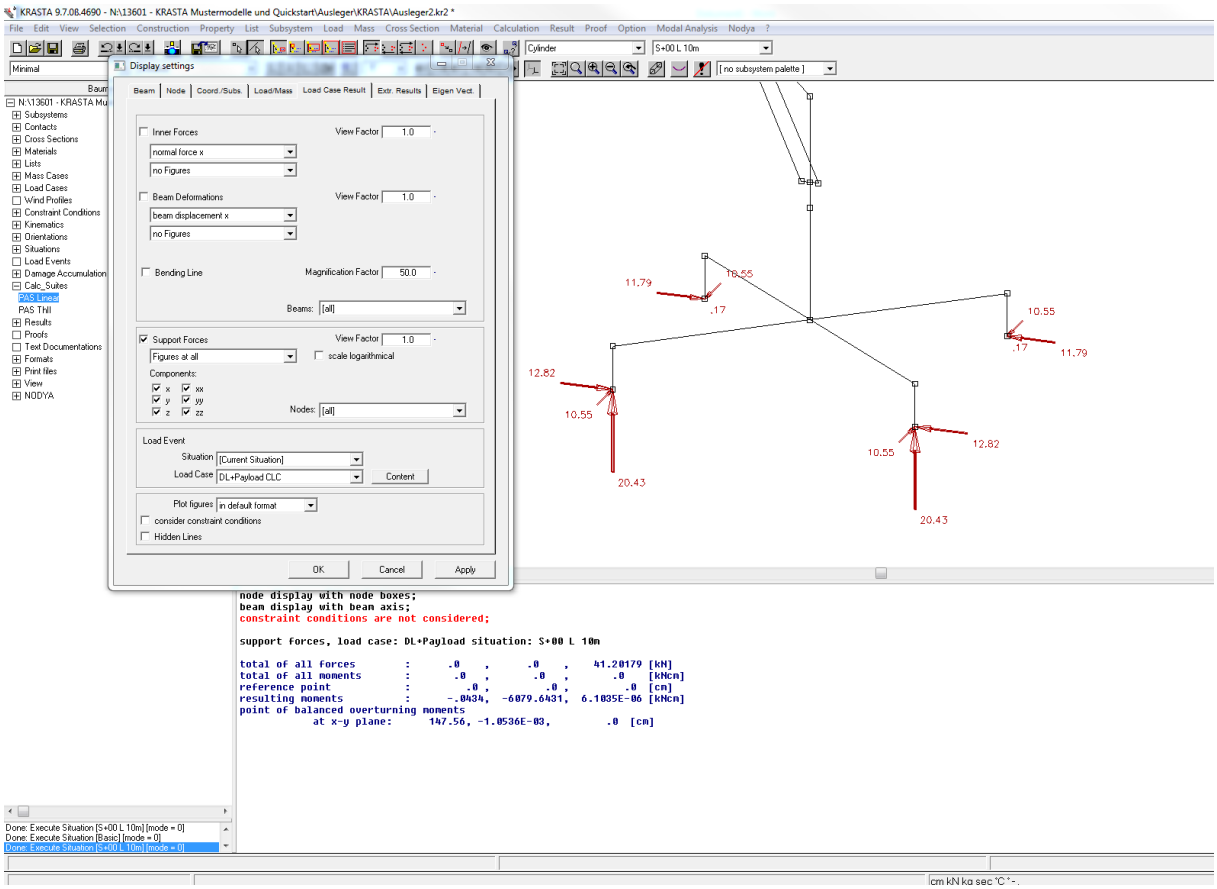
3.11 Result evaluation

3.11.1 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.

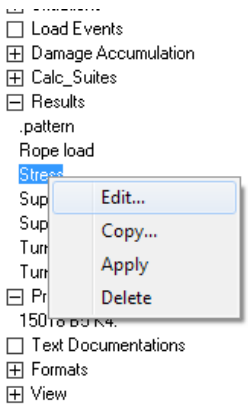
For beams: Inner forces, beam deformations, bending line
For nodes: support forces

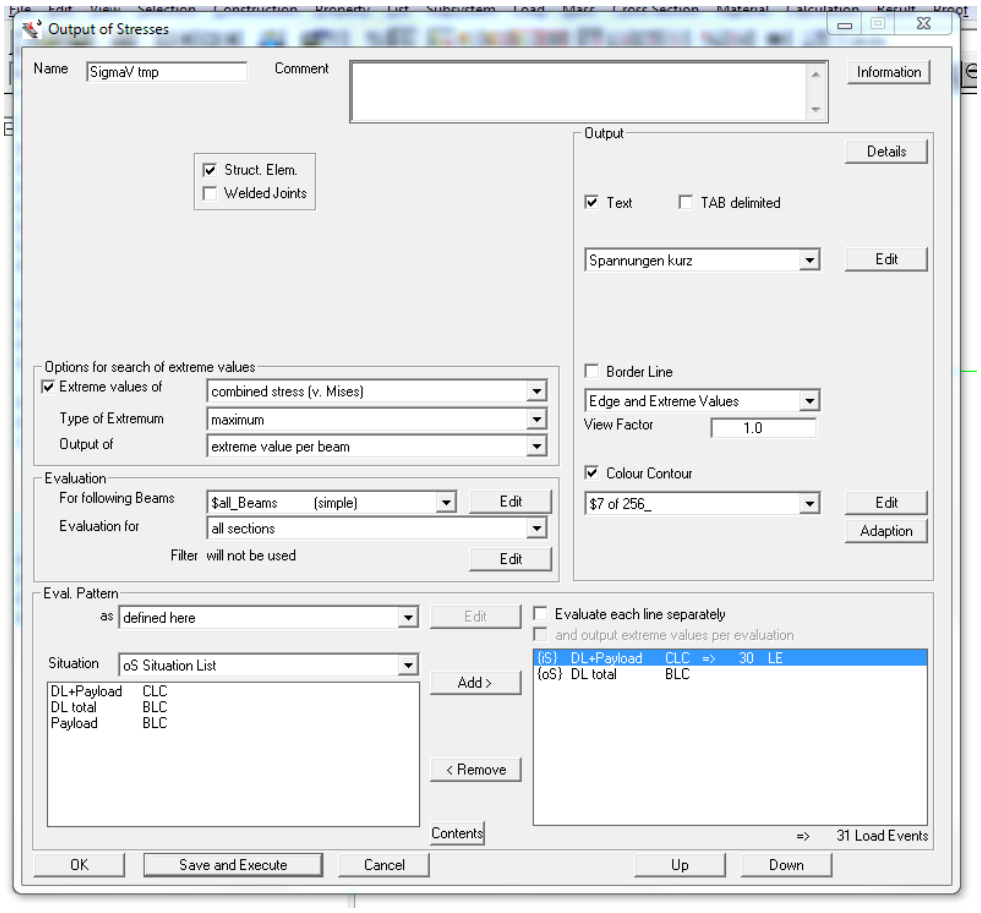
Logic load cases are not available in this dialog, as they usually expand to more than one load permutation.



3.11.2 Result controls

Result controls and proofs are available in the object tree. Some have already been prepared in the example model. 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 results controls can be selected as a prototype.