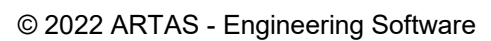


The ultimate mechanism designer



Contents

1 Preface	8
1.1 What is SAM	8
1.2 What is new in SAM 8.2.....	8
1.3 About this User's Guide.....	12
1.4 Purchase Agreement.....	13
2 Overview	16
2.1 The Design Process.....	16
2.2 Capabilities	16
Design	16
Modelling	17
Input Motion	17
External Forces/Torques	17
CAD Interface	21
Analysis Results	21
Post-Processing	22
Optimization	22
2.3 History, Present and Future.....	23
2.4 European Academic Software Award.....	24
3 System Requirements & Installation	26
3.1 System Requirements.....	26
3.2 Installing SAM.....	26
3.3 Use of SAM 3.0, 4.x, 5.x, 6.x, 7.0, 8.0 and 8.1 project files.....	27
4 A Guided Tour	30
4.1 Running a sample project.....	30
A quick example	30
4.2 Using the Design Wizard to create a mechanism.....	41
4.3 Building a new mechanism (without Design Wizard).....	45
Example of a 4-bar mechanism (motion analysis)	46
5 Design Wizards	58
5.1 4-Bar Mechanism.....	58
General (nodes)	58
General (nodes, angle, dimension)	59
Angle Function Synthesis	60
3-Position Synthesis (I)	62
3-Position Synthesis (II)	64
5.2 Exact Straight Line Mechanism.....	64

Symmetric Slider-Crank Mechanism	65
Geared Double-Crank Mechanism	66
Hypo-Cyclic Planetary Gear Mechanism	67
Planetary Gear-Coupled Dyad	68
Belt-Coupled Dyad	69
5.3 Approximated Straight Line Mechanism.....	69
Watt	70
Chebyshev	71
Roberts	72
Evans	73
Hoecken	74
6 Graphics	76
7 Modelling Considerations	78
7.1 Units	78
7.2 Number Representation.....	78
7.3 Exact Nodal Coordinates.....	78
7.4 Motion File	79
Format	80
Importing a motion file	80
Exporting a motion file	80
7.5 Multiple Input Motions.....	81
7.6 Linear Actuator.....	81
7.7 Inclined Linear Guiding.....	81
7.8 Building Multi-Node Linkages.....	82
7.9 SAM/CAD design process.....	83
7.10 User Defined Results	84
8 Kinetostatics	88
8.1 Reaction/Bearing Forces.....	88
8.2 External Force/Torque.....	90
9 Optimization	96
9.1 Introduction.....	96
9.2 Optimization Objective.....	96
Objective Performance Number	98
Reference or Target Definition	99
9.3 Parameter Space.....	100
9.4 Constraints.....	101
9.5 Optimization Options.....	102
Settings Evolutionary Algorithm	103
Settings Simplex Algorithm	104
10 Examples	106

10.1	Motion Analysis.....	106
	Crank Rocker Mechanism	106
	Quick-Return Mechanism	106
	Slider Driven Gear Transmission	108
	Double Crank Mechanism	109
	Planetary Mechanism (Hypo-Cyclic)	109
	Curved Slider	111
	CAM mechanism	112
10.2	Force Analysis.....	113
	Transfer Mechanism (incl. gravity effects)	113
	Belt drive with external force	115
	Height dependent forces on sports car spoiler	117
10.3	Optimization.....	118
	Driving Torque Minimization	118
	Optimization of trajectory	120
	Constrained Optimization	122
11	Elements	132
11.1	Beam	132
11.2	Slider	135
11.3	Curved Slider.....	138
11.4	Gear	141
11.5	Rack Pinion.....	145
11.6	Belt	150
11.7	Sensor	153
11.8	Spring, Damper, Friction.....	154
11.9	Rotational Spring, Damper, Friction.....	160
11.10	Non-linear spring.....	165
11.11	Gas spring.....	167
12	Integrated Analysis Environment	172
12.1	Keyboard and Mouse.....	172
12.2	Toolbar	173
12.3	Menu Commands.....	175
	File Menu	175
	Build Menu	176
	Graphics Menu	177
	Input Motion Menu	178
	Input Dialog Box.....	179
	Loads Menu	180
	Force/Torque Dialog Box.....	181
	Analysis Menu	183
	Optimization Menu	183
	Display Menu	184
	Results Menu	184
	Window Menu	185
	Help Menu	185

13 Literature	188
14 Appendix	190
14.1 Location of Files.....	190
14.2 Status/Warning Messages.....	190
14.3 DXF Template File.....	191
14.4 What was new in SAM 8.1.....	191
14.5 What was new in SAM 8.0.....	192
14.6 What was new in SAM 7.0.....	203
14.7 What was new in SAM 6.2.....	204
14.8 What was new in SAM 6.1.....	204
14.9 What was new in SAM 6.0.....	204
14.10 What was new in SAM 5.1.....	206
14.11 What was new in SAM 5.0.....	206
14.12 What was new in SAM 4.2.....	207
14.13 What was new in SAM 4.1.....	207
14.14 What was new in SAM 4.0.....	208
15 Glossary of Terms	212
Index	213

Chapter

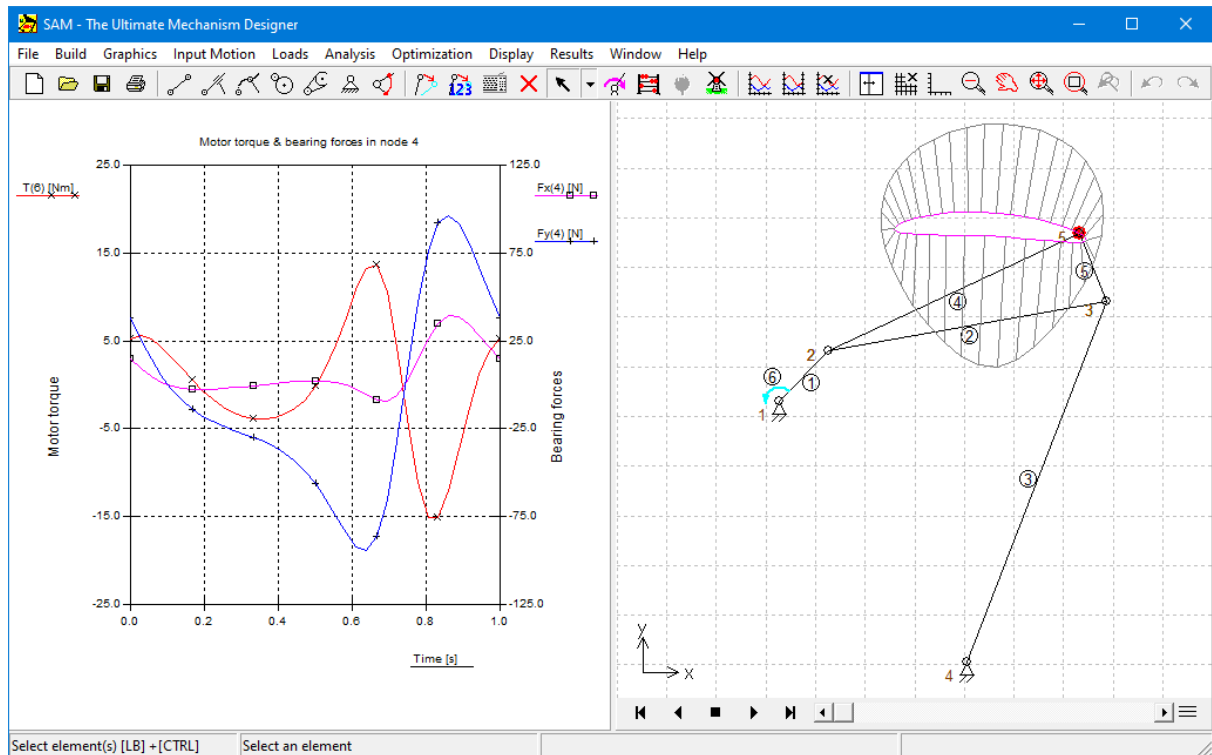


1

1 Preface

1.1 What is SAM

SAM (Synthesis and Analysis of Mechanisms) is an interactive PC-software package for the design, analysis (motion and force) and optimization of arbitrary planar mechanisms. Mechanisms can either be generated via the design wizards or they can be assembled from basic components including beams, sliders, gears, belts, rack-pinion, springs, dampers and friction elements. SAM integrates pre-processing, numerical analysis, post processing, such as animation and xy-plots, and automated constrained optimization in one single easy-to-use program.



Typical screen shot of a mechanism design session

The mathematical foundation of the analysis kernel, which is inspired by the well-known finite element approach, offers a large number of features and overcomes many of the problems of traditional mechanism programs. Open loop, closed loop, multiple loop and even complex planetary mechanisms can equally well be analyzed due to the finite element formulation. Even the most complex mechanisms, including planetary gear trains, can be modeled within minutes.

1.2 What is new in SAM 8.2

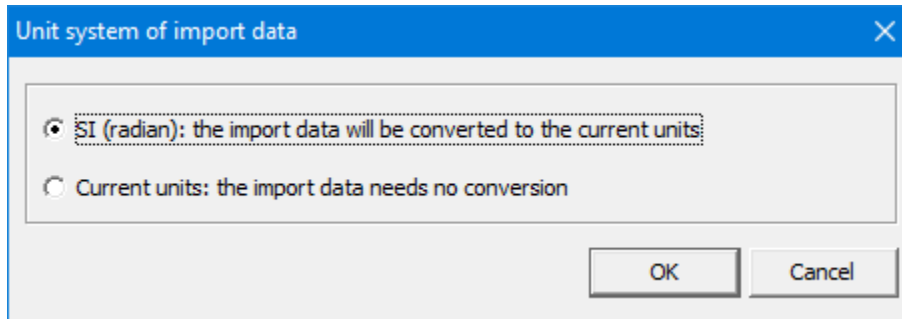
Unified selector for nodes, elements and graphics/shapes

In previous versions of SAM the selector icon had two operating modes, one for selecting mechanism related objects, and one for graphics related objects/shapes. As of SAM 8.2, there is just one selector icon.

Unified import of data (no more linking to files + option to interpret data as SI or current units)

- In previous versions of SAM external data (motion files, optimization reference files, forces definitions ...) was sometimes accessed via a link to the file, and sometimes the data was imported and stored within the SAM project file. As of SAM 8.2, all external data is always imported and part of the SAM project file.

- During the import process the user can state the unit system of the external data. If the data in the external file is based on SI(rad) units, an automatic conversion into the current unit system being used in SAM will be executed. If the data in the external file is already in line with the current units, the user can choose the second option and import the data without conversion.



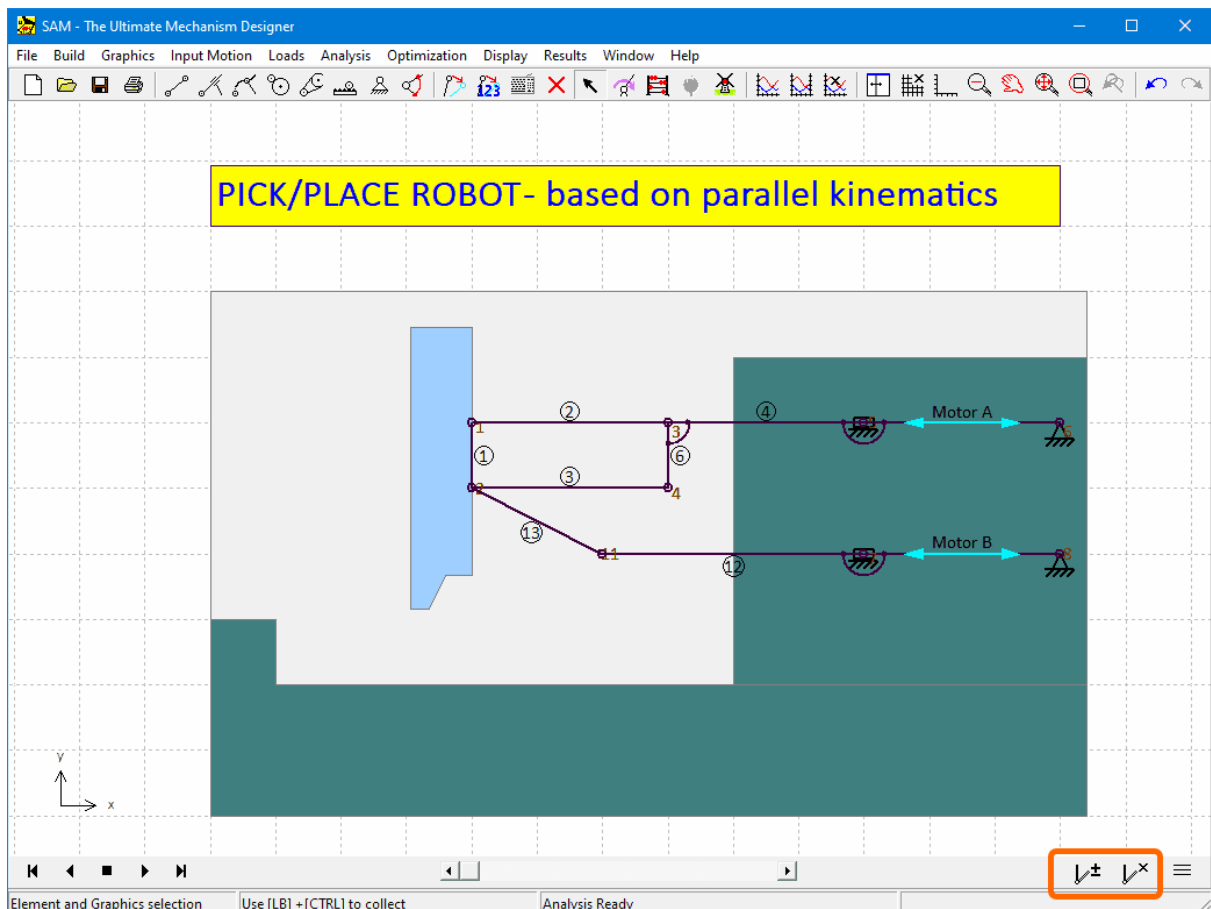
Dialogue in which the user can choose which unit system corresponds to the data in the import file.

Text box including call-out

To enhance annotation options, a text box including call-out functionality has been added to the list of available graphics/shapes.

Fill color + display order

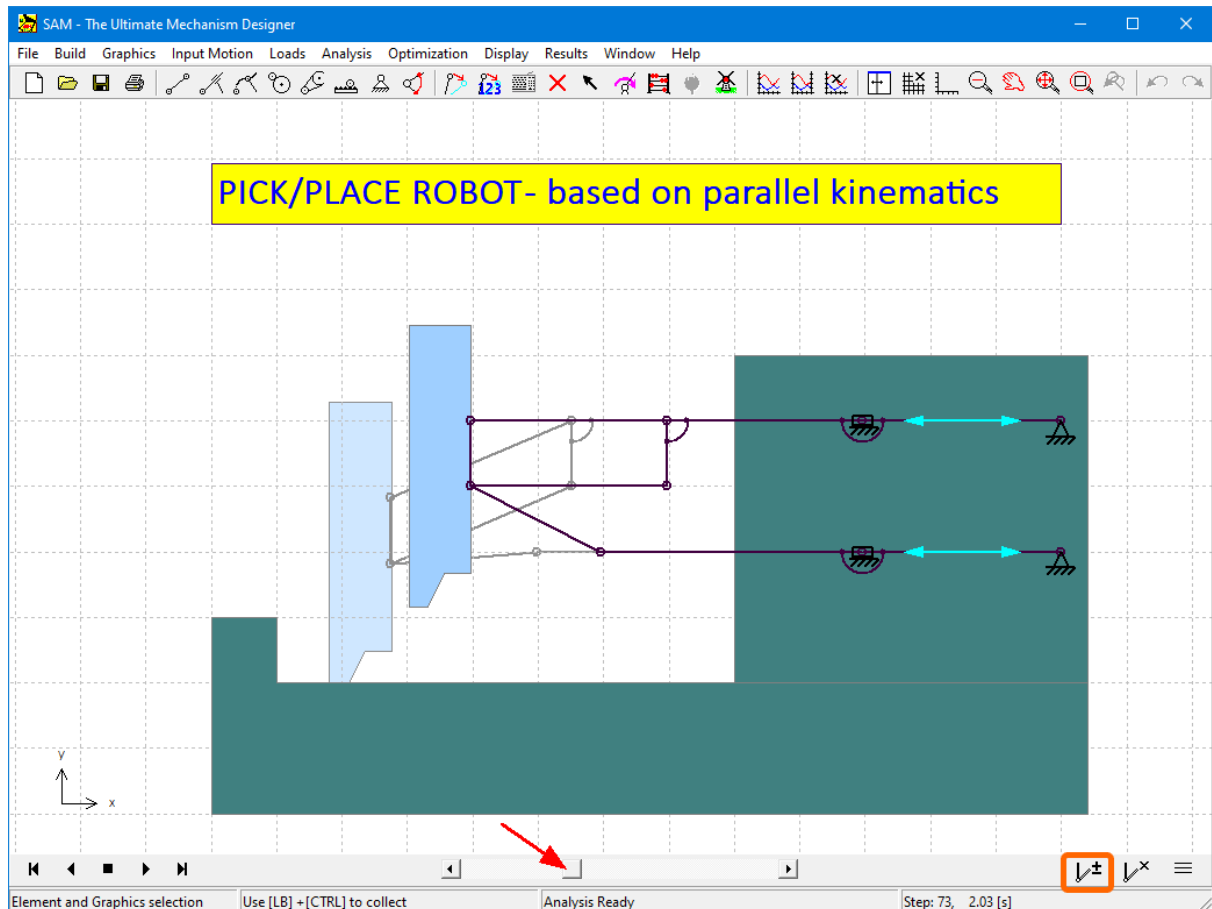
The user can define a fill color for all shapes as well as the order in which stacked shapes are displayed.



Example showing fill color and display order

Freeze/Unfreeze

Sometimes it is desirable to display one or more additional positions of the mechanism during stand-still or animation. For that purpose freeze/unfreeze buttons have been added to the animation player. The first one adds/removes one single position, and the second button removes all additional positions. Moving the mechanism towards the corresponding position and pressing the freeze/unfreeze button does the trick.



Adding/Removing a freeze position by moving the mechanism towards the corresponding position and pressing the add/remove button

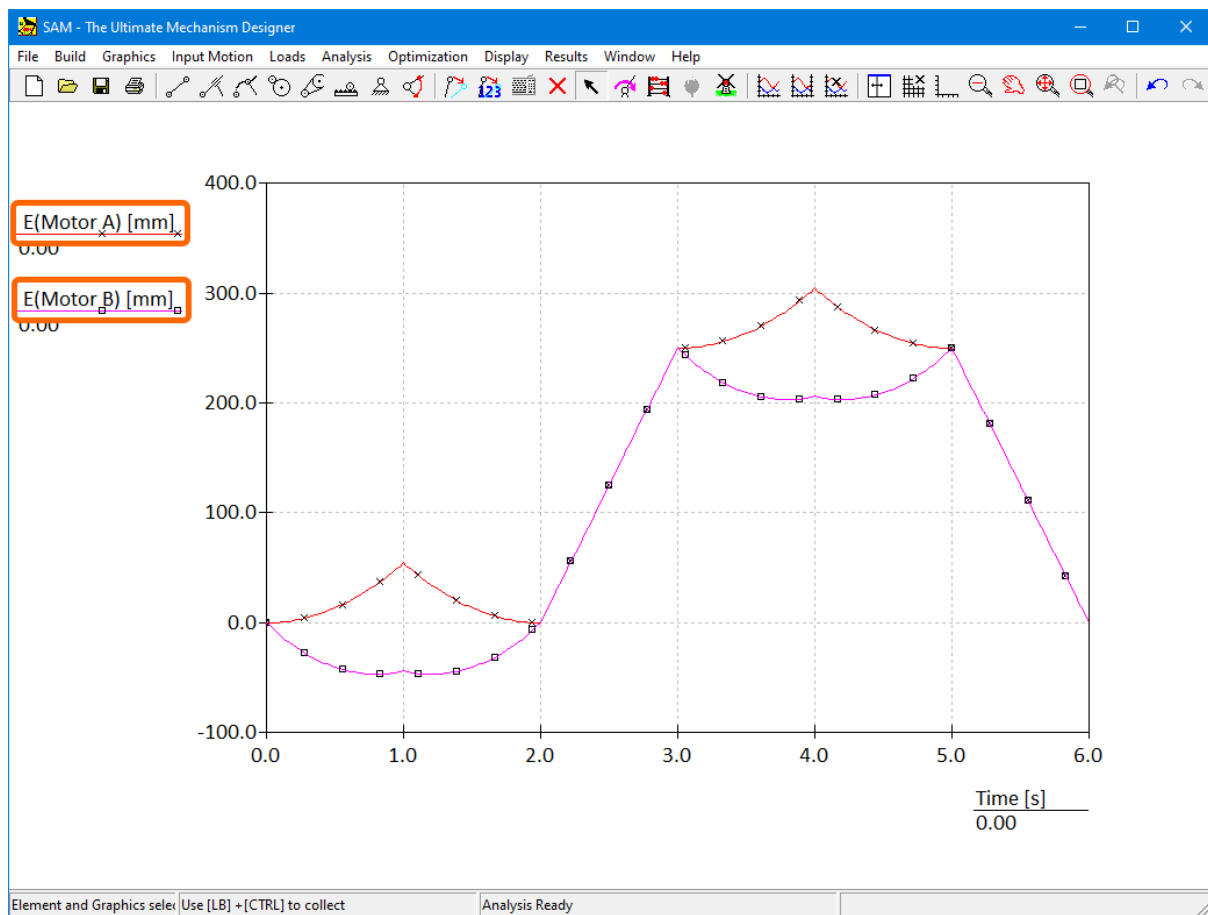
Cascading styles (Preferences - Project - Object)

Various styles, like font size, (fill) color, ... can be defined at various levels. The settings at preferences level are the initial settings for each new project, but can be adapted at project level. Within each project, the settings at project level serve as initial values for each new object, but can be adapted at the level of each individual object. Most settings are only found at two out of these three levels.

User-defined element/node labels

The default numbers to identify a element or node have been replaced by labels. Initially, the label is identical to the internal element/node number. In the previous example of the pick&place robot, the two linear motor elements correspond to elements 5 and 7, but the labels have been replaced by 'motor A' and 'motor B'.

Please note that in the xy-plots these user-defined labels are displayed (see example below), whereas during post-processing of results (Results>User Defined...), the use of the internal identification numbers is still required.



Motion (=elongation E) of the driving elements with labels 'Motor A' and 'Motor B' are shown as function of time

Movable element/node labels

Sometimes one feels the need to overrule the default location of the element/node labels as defined under File>Preferences>Display. After selecting an element or node, the right-mouse drop-down menu now also offers the command 'Move Label', which allows to adjust an individual label with the mouse.

New optimization option: rejection of experiment based on percentage completed motion steps

In most cases, it is useful to reject any optimization experiment during which the full motion could not be completed. Generally, this is caused by convergence issue since the indicated input motion step cannot be executed by the mechanism. However, in some optimization cases, for example in case one wants to maximize the output motion range of a mechanism without knowing upfront what the corresponding input motion will be, it can be useful to accept experiments that did not complete all input motion steps. To facilitate this, an extra optimization option has been introduced, which defines the rejection of an experiment based on the percentage of completed motion steps. The default value is 100%, meaning that the entire motion cycle must be completed, but the value can also be set to 75%, 50%, 25% or 10%.

Optimization Options

Modus

☒ Automatic (global exploration & local optimization) ☒ Optimize all solutions from global exploration

☐ User-Controlled

Local Optimization

☐ Evolutionary Algorithm

☒ Simplex Method

Evolutionary Settings

Nexperiments

Population Size

Simplex Settings

Max Iterations

Search within boundaries ☒

Input Motion

Reject individual experiments if the successful number of motion steps, expressed in percentage, is less than:

OK Cancel

New optimization option

1.3 About this User's Guide

Chapter 1: Preface gives an overview of the manual and contains the purchase agreement.

Chapter 2: Overview gives you some background information about SAM, its capabilities and plans for the future.

Chapter 3: System Requirements & Installation contains information about system requirements and the installation procedure.

Chapter 4: A Guided Tour shows how to run an existing mechanism project and explains how to create your own mechanism and how to perform a complete analysis cycle, starting from scratch and showing all steps from mechanism definition up to customizing of graphs.

Chapter 5: Design Wizard gives an overview of the automatic generation of mechanisms using the easy-to-use Design Wizards.

Chapter 6: Modeling Considerations discusses a number of the more delicate modeling issues.

Chapter 7: Kinetostatics is about quasi static force analysis, definition of external forces and bearing forces.

Chapter 8: Optimization explains the various optimization possibilities.

Chapter 9: Examples presents picture stories of various mechanism projects.

Chapter 10: Elements tells you all about the various elements that can be used.

Chapter 11: Integrated Environment discusses in more detail the use of mouse and keyboard, the toolbar and the various menu items.

Chapter 12: Appendix

Chapter 13: Glossary of Terms

Chapter 14: Index

1.4 Purchase Agreement

This software and accompanying documentation are protected by Copyright law and also by International Treaty provisions. Any use of this software in violation of Copyright law of the terms of this agreement will be prosecuted to the best of our ability.

Under no circumstances may you copy this software or documentation for the purpose of distribution to others. Under no condition may you remove the copyright notices made part of the software or documentation.

ARTAS does not assume any liability for the use of SAM beyond the original purchase price of the software. In no event will ARTAS be liable to you for additional damages, including any lost profits, lost savings or other incidental or consequential damages arising out of the use of or inability to use the program, even if ARTAS has been advised of the possibility of such damages.

By using this software, you agree to the terms of this section. If you do not agree, you should immediately return the entire software for a refund.

Chapter



2

2 Overview

2.1 The Design Process

The process of designing mechanisms, basically can be divided into three distinct phases:

- Synthesis
- Analysis
- Optimization

After a proper specification of the demands, the first step in the design cycle consists of the synthesis phase, in which the designer attempts to find the type of mechanism and its dimensions, such that the requirements are met (as good as possible). Experience, previous designs, mechanism handbooks and the design wizards implemented in SAM can guide this creative process.

Once a mechanism has been chosen, its motion and force behavior can be analyzed. Typical questions, such as "how long is the usable stroke of an approximate linear guiding, given a certain acceptable deviation from the straight line" or "what are the bearing forces", can be answered with little effort using computer simulation. Looking at the class of mechanism independent programs, one can observe various approaches, such as:

- Modular kinematics
- Vector analysis
- Kinematic constraints
- Finite element method

SAM is based on the latter of these approaches, which has many advantages and overcomes many of the problems of traditional methods.

Finally, the optimization module that is part of the Professional edition of SAM offers constrained single-function multi-parameter optimization based on a mix of evolutionary algorithms and Simplex techniques. Constraints are dealt with by treating each violation of a constraint as a penalty that is added to the original cost function. The software offers the option to define own results from the standard set of results via an advanced formula parser. This option is also used to define and add penalties.

2.2 Capabilities

2.2.1 Design

SAM offers a set of design wizards which will help you to synthesize mechanisms for specific tasks, such as:

- Angle function generation (a minimum of 3 pairs of input/output angles are to be satisfied).
- 3-Position/angle synthesis of the coupler plane
- Approximated straight line motion
- Exact straight line motion

In case these design wizards do not provide the solution to the specific design problem, the user has to rely on his/her experience, previous design, handbooks or trial & error to invent the mechanism, which can then be modeled and analyzed in SAM.

2.2.2 Modelling

SAM is equipped with a large library of basic elements, including:

- beam, slider, curved slider
- belt, gear, rack-pinion
- sensor (both translational and rotational)
- spring, damper and friction element (both translational and rotational)
- non-linear spring
- gas-spring

This allows the analysis of a huge variety of mechanisms. The unique mathematical foundation of the program offers a large number of features and overcomes many of the problems of traditional mechanism programs. Open loop, closed loop and even multiple loop mechanisms are treated in the same way and even the most complex mechanisms, including planetary gear trains, can be modeled within minutes.

2.2.3 Input Motion

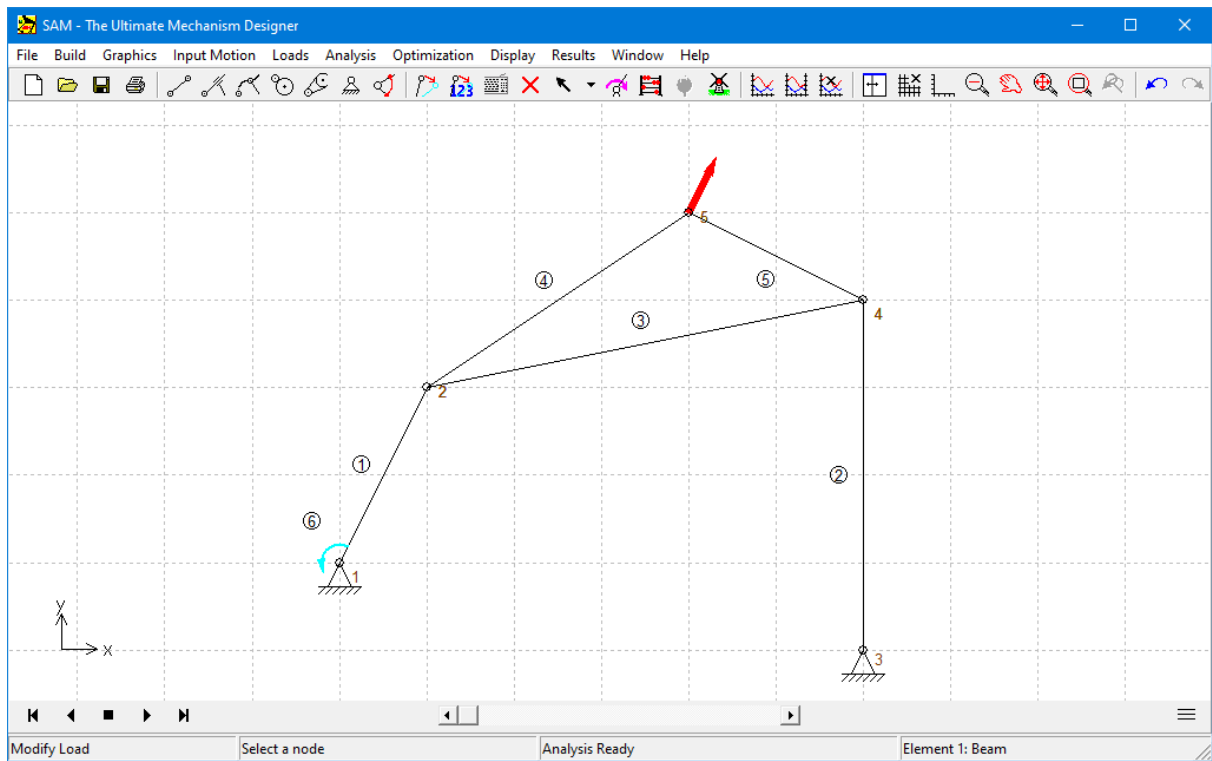
SAM allows the definition of up to 10 independent inputs, which can either be defined in terms of absolute displacements or in terms of elemental change of shape to model relative inputs (e.g. elongation of a hydraulic cylinder or relative rotation of a robot elbow). Each of the inputs can be defined independently. Various frequently used input motion laws are available:

- constant velocity
- polynomial
- cyclical motion
- 2.order velocity profile
- cubic splines

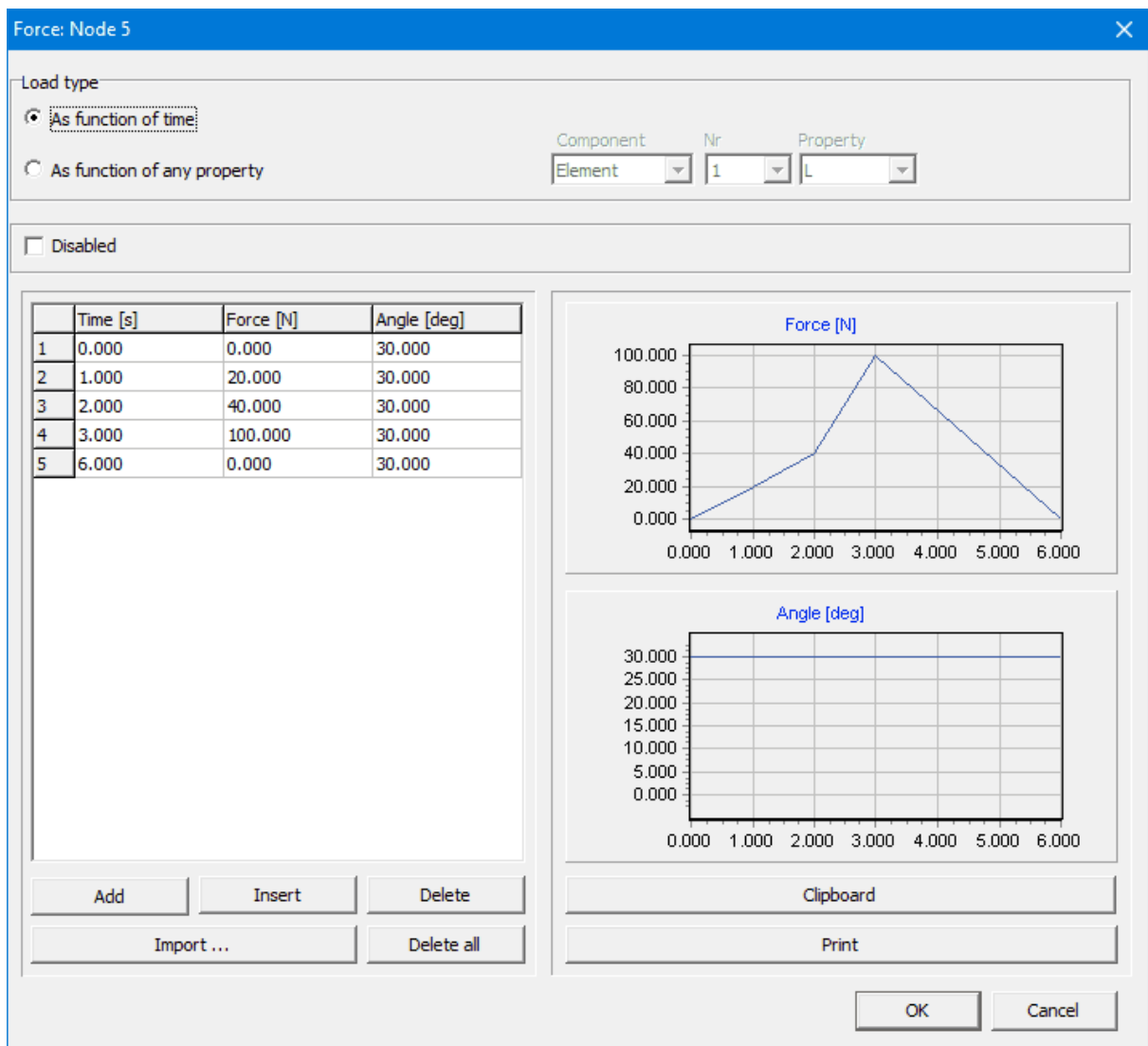
The various input motion laws can be combined to form any desired input diagram. Inputs can also be imported from an external ASCII file or defined via a table to enable the definition of arbitrary motions.

2.2.4 External Forces/Torques

External forces or torques can be applied as function of time or as function of any motion results that is calculated by SAM. To illustrate this feature the following example mechanism is used.



Example mechanism showing the external force/torque definition capabilities (currently a force at a -90 degrees angle relative to the angle of element 5 (node 4 to node 5) is shown).



Force profile as function of time

Force: Node 5

Load type

☐ As function of time

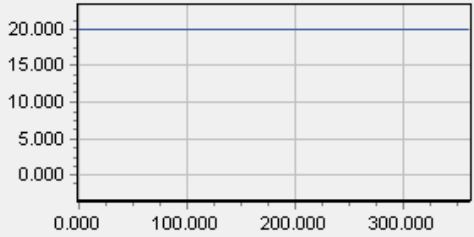
☒ As function of any property

Component: Element Nr: 5 Property: A

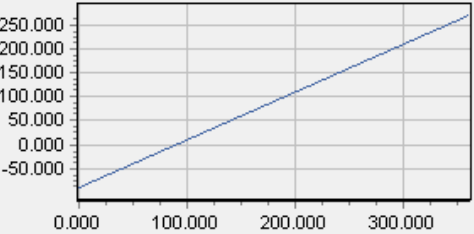
☐ Disabled

	A [deg]	Force [N]	Angle [deg]
1	0.000	20.000	-90.000
2	360.000	20.000	270.000

Force [N]

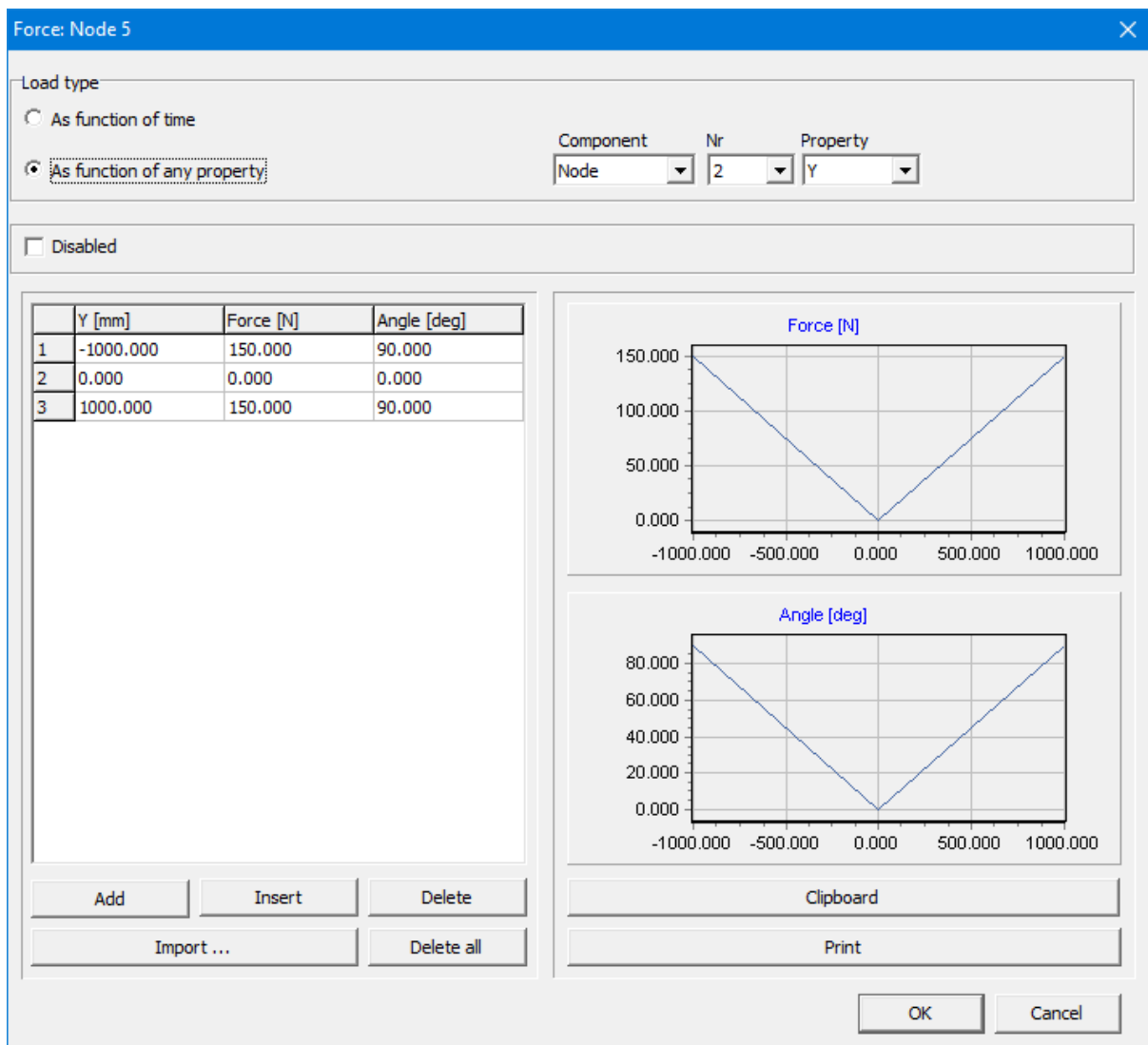


Angle [deg]



Buttons: Add, Insert, Delete, Import ..., Delete all, Clipboard, Print, OK, Cancel

Force profile as function the angle A of element 5, such that the force is always under a -90 degrees angle relative to the element angle (see example mechanism)



Force profile as function the vertical position Y of node 2

2.2.5 CAD Interface

The DXF import/export facility lets you export your conceptual mechanism design to any CAD program to work out the details and it lets you import CAD data to easily set-up the mechanism in SAM or to perform animation of the final mechanism.

2.2.6 Analysis Results

Once the mechanism has been constructed and the inputs have been defined any of the following kinematic quantities can be calculated (all relative or absolute):

- nodal position, displacement, velocity, acceleration
- angles, angular velocity and acceleration

Furthermore, SAM can perform force-analysis, thus enabling the calculation of:

- driving torque (force)
- reaction forces in bearings

- internal forces in elements
- required or transmitted power
- A sophisticated formula parser offers the possibility to combine simulation results in multiple ways to derive user defined results.

2.2.7 Post-Processing

The analysis results can be displayed either in tabular or graphical form. The tabular listing can be viewed on the screen, send to a printer or stored in a readable formatted list file. The x/y plot option allows to plot any variable against time or any other variable. An unlimited number of functions can be combined into one x/y plot with optionally two different scalings to allow proper multiple display of variables with different amplitude ranges. It is possible to output selected data to an external file (ASCII format) for customized post-processing.

A sophisticated formula parser offers the possibility to combine simulation results in multiple ways to derive user defined results.

SAM can also animate the mechanism motion.. As a further aid for the designer the path and velocity hodograph of any number of moving points can be plotted. Also, a complete project documentation (ASCII-format) can be automatically generated.

2.2.8 Optimization

Included in PROFESSIONAL version of SAM only !

The optimization module of SAM offers constrained single-function multi-parameter optimization based on a mix of evolutionary algorithms and Simplex techniques. Constraints are dealt with by treating each violation of a constraint as a penalty that is added to the original cost function. The software offers the option to define own results from the standard set of results via an advanced formula parser. This option is also used to define and add penalties.

Taking the initial design/topology as a starting point one can for example further improve the quality in which the trajectory of a coupler point equals the target trajectory by changing the geometry of the mechanism within pre-defined ranges. Or one can minimize the peak or RMS value of the driving torque of a mechanism by adding a compensating mass and let SAM determine the optimal value of the mass and its position within the allowable range. Just as in the case of the trajectory optimization one can also specify a reference function and minimize the difference between the actual and the reference function. When designing for example fitness equipment one is generally seeking a predefined force as function of displacement.

The goal for optimization can be the minimization or maximization of a variety of properties (peak, RMS, average, ...) of the difference between the actual and the target behaviour of a mechanism, such as :

- trajectory of a node (with or without prescribed timing)
- any motion or force quantity (as function of time or another quantity)
- any user-defined quantity

SAM seeks the optimum by modifying the following properties within user-defined ranges :

- geometry of mechanism

- element properties, such as mass, spring constant, transmission ratio, ...

The optimization process in SAM is based on a two step approach, consisting of :

- Exploration of the design space
- Optimization of a specific solution

First, the entire parameter space is explored globally using a combination of a pure Monte-Carlo technique and a so-called Evolutionary Algorithm, which is a optimization technique derived from Genetic Optimization. The top list of such a global exploration are shown in the Explore list box, which displays the value of the optimization function and the corresponding parameters. The individual with the best property is listed at the top. Next, the designer can select one of the results from the Explore window and start a local optimization. This local search can be either based on a Simplex technique or on a Evolutionary Algorithm with a smaller parameter range centered around the selected solution.

The combination of a global exploration strategy and a local optimization strategy (with the designer in the loop for selecting the mechanism that is further optimized) is believed to give the best trade-off between speed and coverage of the design space. **Next to this user-controlled approach a fully automated approach can also be selected.**

2.3 History, Present and Future

The development of SAM already started in 1985 (on a Kaypro-X computer and CP/M operating system), however it took until 1991 before the first DOS-version of the program was released and since then many enhancements have been added. A brief overview :

1985: SAM 0.0 Start of development (Kaypro-X, CP/M as O.S.)

1989: SAM 0.1 First DOS version

1991: SAM 1.0 First official release, kinematics only

1991: SAM 1.1 Printer/plotter drivers

1992: SAM 2.0 Kinetostatics of linkage mechanisms

1993: SAM 2.1 CAD import added

1994: SAM 2.2 Polar and relative node definition, formula parser

1995: SAM 3.0 First Windows version

1998: SAM 4.0 CAD-interface, various display options, ...

1999: SAM 4.1 Engineering units, project documentation, ...

2001: SAM 4.2 Kinetostatics of belt and gear, CAD-tool, ...

2003: SAM 5.0 Design Wizards, optimized analysis kernel, ...

2005: SAM 5.1 Non-linear spring, more and new input motions, ...

2007: SAM 6.0 Optimization, Context sensitive right mouse button

2010: SAM 6.1 Clamped cubic spline, extended automated optimization, new display features, automated updates, videos in GIF format, ...

2014: SAM 7.0 User-defined results, constrained optimization, easy calculation of bearing forces in moving nodes, new transformation & grouping commands, new licensing system

2019: SAM 8.0 Rotation motor element, extended torque/force definition, curved slider / CAM profile, (semi-)dynamic construction aid, gas spring element, SVD analysis for improved validity diagnosis of mechanism, extended animation player, ...

2021: SAM 8.1 Rack pinion element, automatic synchronization of multiple input motions, increased analysis/optimization speed, ...

2022: SAM 8.2 Unified import of external data with optional unit conversion, new shapes, fill colors and order of shapes, freeze/unfreeze functionality, cascading styles, ...

Future plans for SAM include :

- Collision detection
- Friction models based on friction coefficients and normal forces
- Synthesis of cam mechanisms

2.4 European Academic Software Award

SAM was submitted twice for the "European Academic Software Award" (EASA). Both times, after being evaluated very thoroughly by various international experts, SAM reached the final of this prestigious competition, which has been created to stimulate the development of high quality software for education and research. EASA is a joint initiative of various European institutes from Austria, Germany, United Kingdom, The Netherlands, Scandinavia, Spain and Switzerland. The event is even supported by the European commission (directorate XIII).

Chapter



3

3 System Requirements & Installation

3.1 System Requirements

SAM requires the following hardware and system software :

- Microsoft Windows 7, Windows 8.x, 10 or higher
- At least 50Mb hard disk space (incl. manual).

3.2 Installing SAM

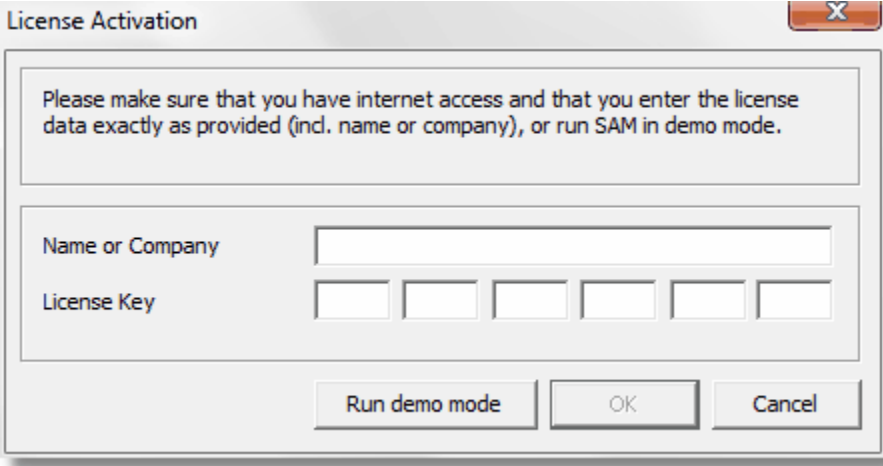
SAM comes in a compressed format. The installation program will expand the files and install each at the proper location on the hard disk.

- Run "SAM82_SETUP.EXE" to start the installation program.
- Select the directory where you want SAM to be installed.

You can either add the SAM icon to an existing program group of your choice or you can create a new program group.

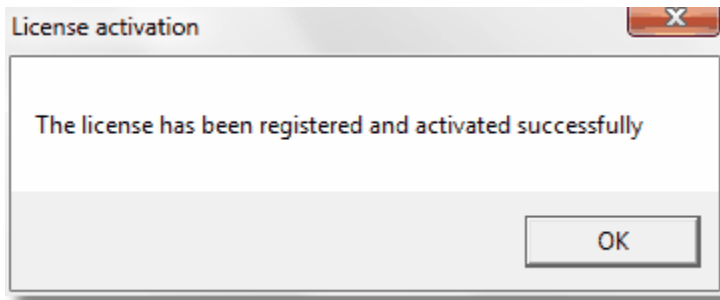
Node-Locked License

The first time that you start-up SAM after the installation you will be requested to fill in your specific license data that has been supplied with the original software. Please make sure that you have internet access during the activation process and that you type in the name/company and license data in exactly the same way as it was provided by us. If you have a trial license you must also have internet access every time you start SAM.

A screenshot of a 'License Activation' dialog box. The title bar says 'License Activation' with a close button (X). The main text area contains the instruction: 'Please make sure that you have internet access and that you enter the license data exactly as provided (incl. name or company), or run SAM in demo mode.' Below this, there are two input fields: 'Name or Company' (a single text box) and 'License Key' (a series of six small text boxes). At the bottom, there are three buttons: 'Run demo mode', 'OK', and 'Cancel'.

License Information Dialog

Once you have supplied the correct license data, you will get the following confirmation and SAM is fully operational.



Floating License

Please contact your IT-manager.

3.3 Use of SAM 3.0, 4.x, 5.x, 6.x, 7.0, 8.0 and 8.1 project files

The new SAM 8.2 project file format/syntax differs from older versions of the software. Old SAM 5.x, SAM 6.x, SAM 7.0, SAM 8.0 and SAM 8.1 projects can be opened in the new version. When saving such a project in the new version for the first time, the user is asked to confirm that the project will be saved in the new format. This process is irreversible and files can only be edited with the new version or higher after saving.

Note : Older SAM versions do not support the new SAM 8.2 project format and **can not** open 8.2 project files.

SAM 3.x and 4.x projects were saved as a binary file, while from version 5.0 onwards an ASCII file is used. The last version of SAM that can still open an old binary project file is SAM 6.1. In order to work with old 3.x or 4.x binary project files in the current SAM version, these project files must first be opened with a (temporary) license SAM 6.1 and saved as a SAM 6.1 ASCII file, which can then be opened in the current SAM version.

Chapter



4

4 A Guided Tour

4.1 Running a sample project

In order to open and run a sample project, simply perform the following steps :

1. Open an existing mechanism project



Choose Open from the File menu or click the corresponding tool in the toolbar. Select one of the example files using the Open dialog box, that will appear. Then click on the OK button.

2. Animate mechanism

Choose any of the animation buttons in the lower section of the mechanism window.

3. View analysis results



Choose Select from the Results menu to select analysis results for further post-processing followed by Export or Graph.

The Export option will call the editor of your choice and display the selected data in tabular format, whereas the Graph option will result in an xy-plot of the selected data as a function of time. Select more than one item for a graph and watch the changes in the graph when you click one of the y-labels in the graph.

Furthermore, simultaneous display of a graph and the animated mechanism can be performed by selecting Tile from the Windows menu.

4.1.1 A quick example

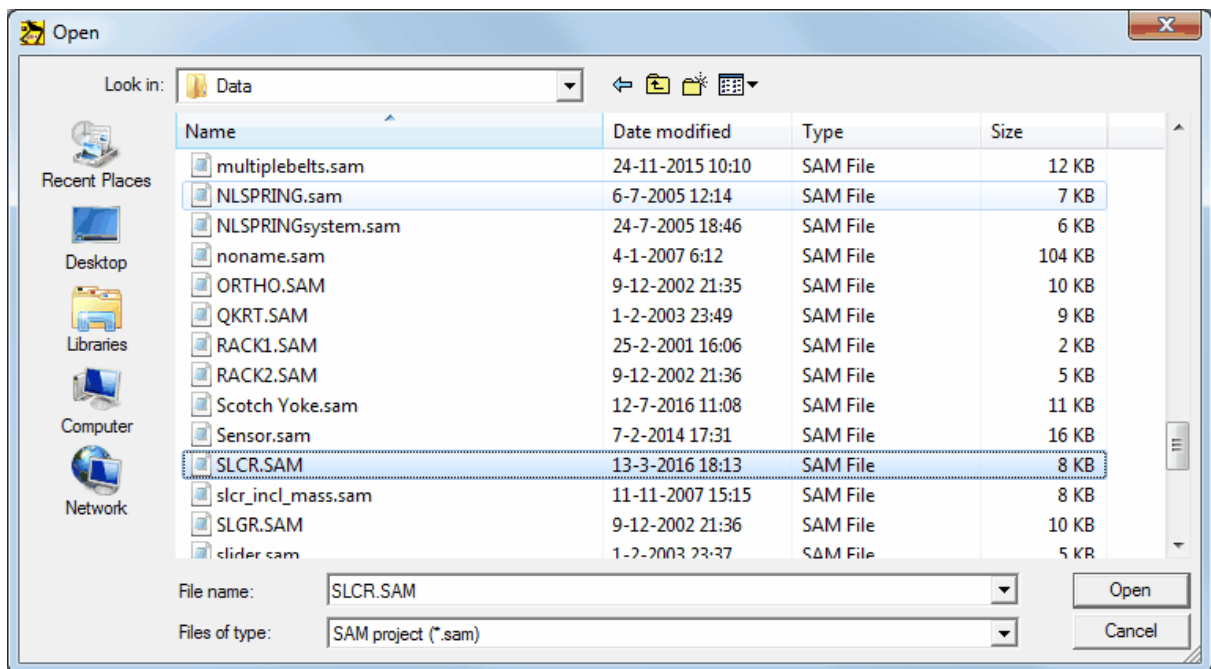
In order to illustrate the various steps we will look at the example of a slider-crank mechanism.

Motion Analysis

Choose Open from the file menu or click



The Open dialog will appear.

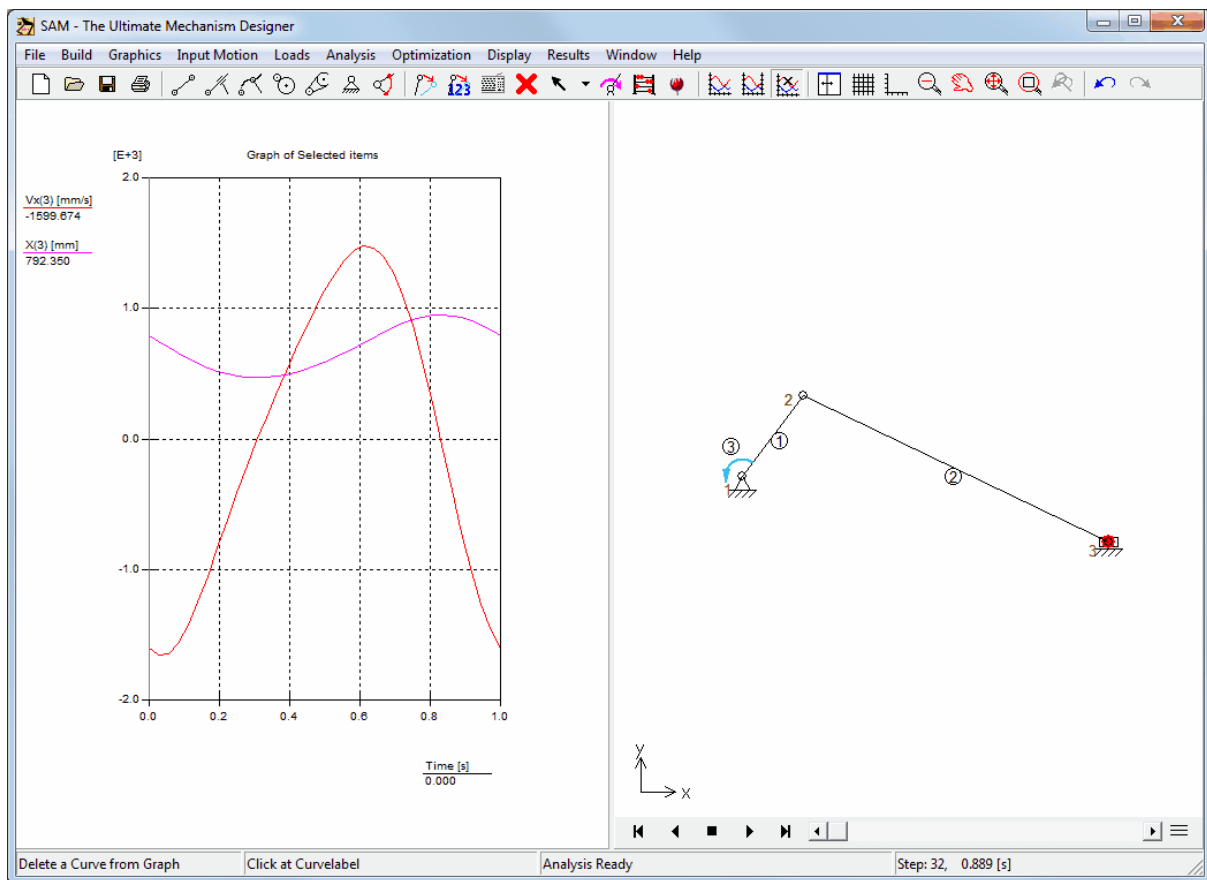


Open dialog box

Select the file "SLCR.DAT" by clicking on it, then click on the OK button.

Alternatively you can double-click the file only instead of a single click followed by the OK button

The mechanism file will be loaded and your screen will look like this :



Screen shot after loading of mechanism project file "SLCR.SAM"

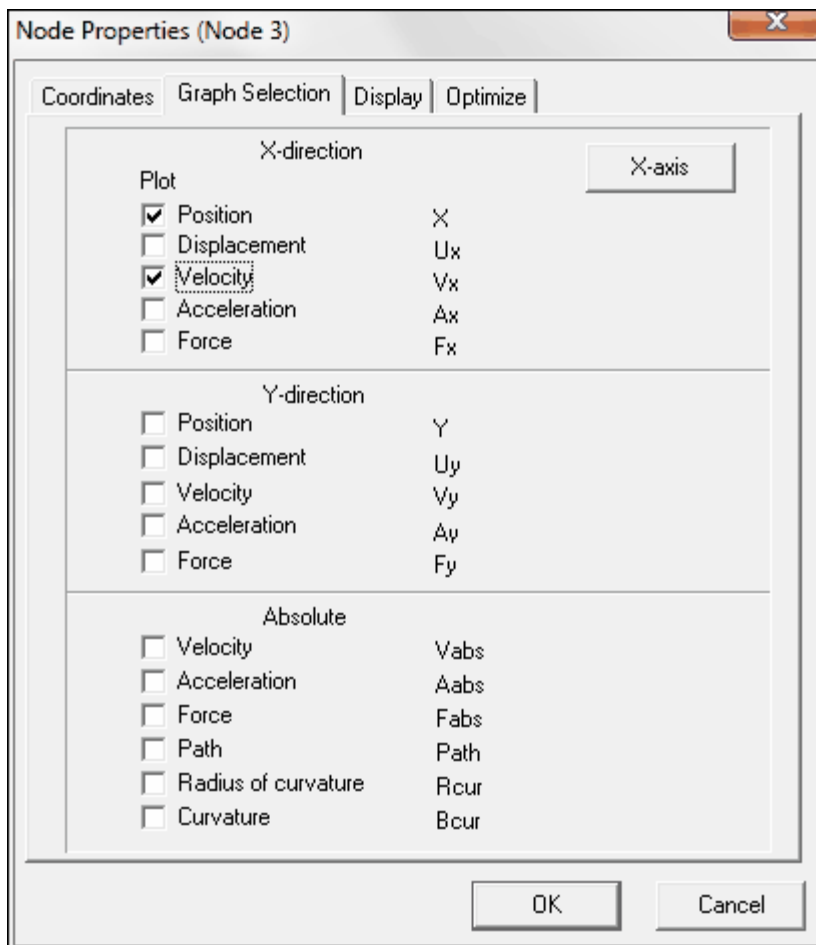
In the right window the mechanism is shown (including a mass of 25kg in node 3), whereas the left window shows the position and velocity of node 3 in x-direction.

Press any of the Animation keys underneath the mechanism

You will now see an animated display of the mechanism.

Choose Select in the Result menu or click . Next, click node 3.

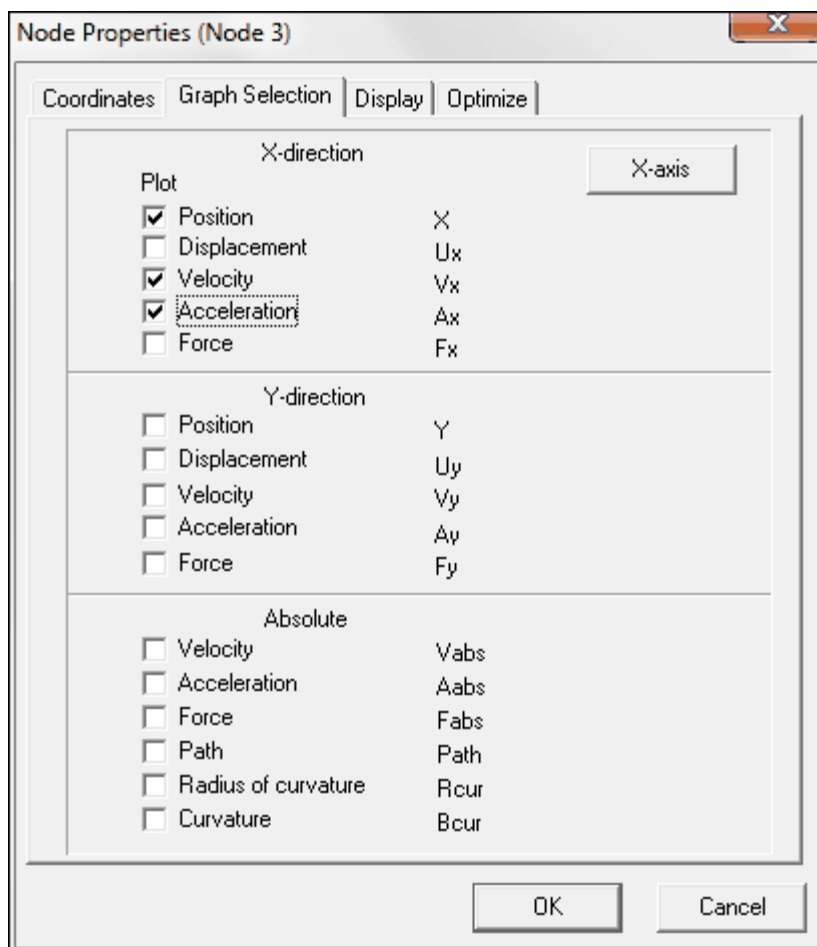
A dialog box will appear that shows all properties of that node. The checked items are currently plotted in the graph.



Node properties dialog box

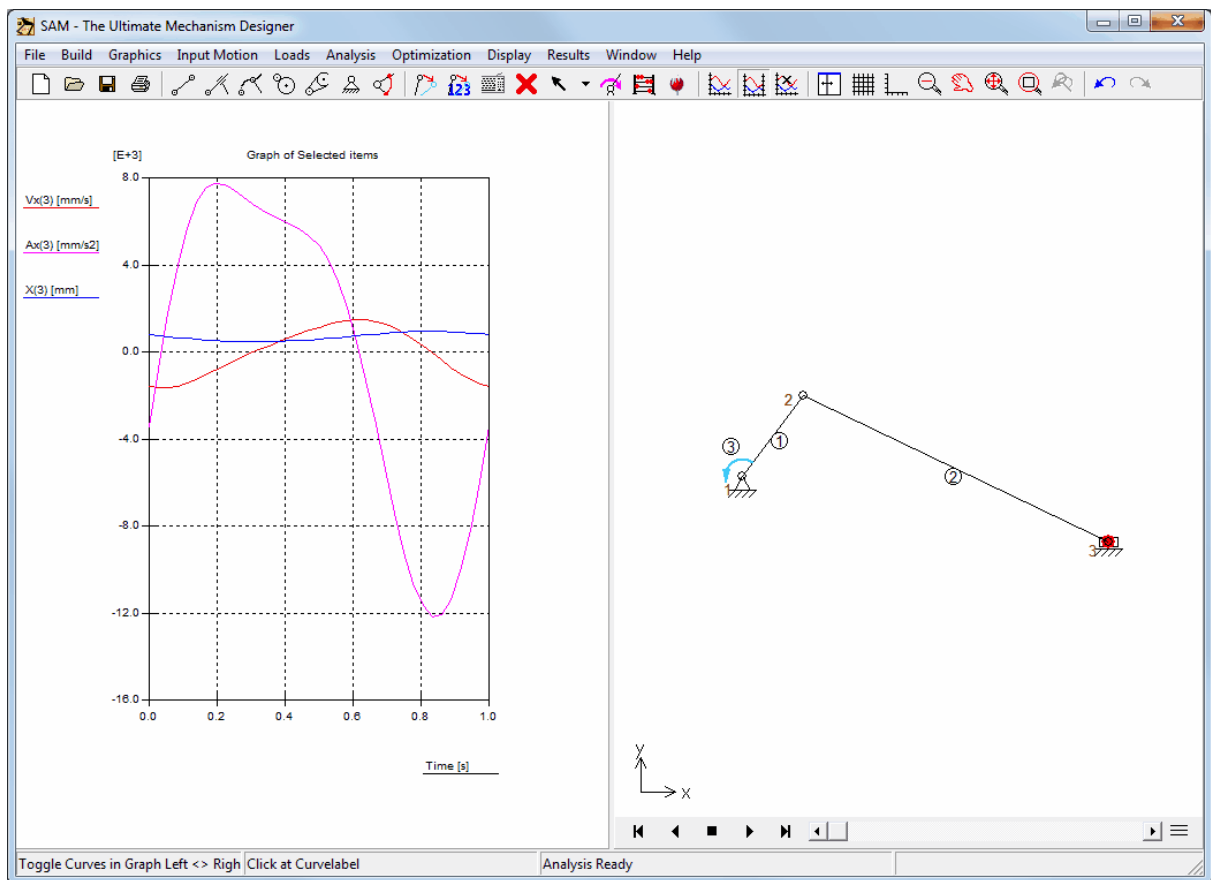
As you can see most items are blanked, meaning that these are not plotted (and also not calculated). If you want more results, you have to select that item as well and press OK.

Check the Acceleration A_x and press OK.



Node properties dialog box

Automatically the graph is updated and the requested result is displayed.

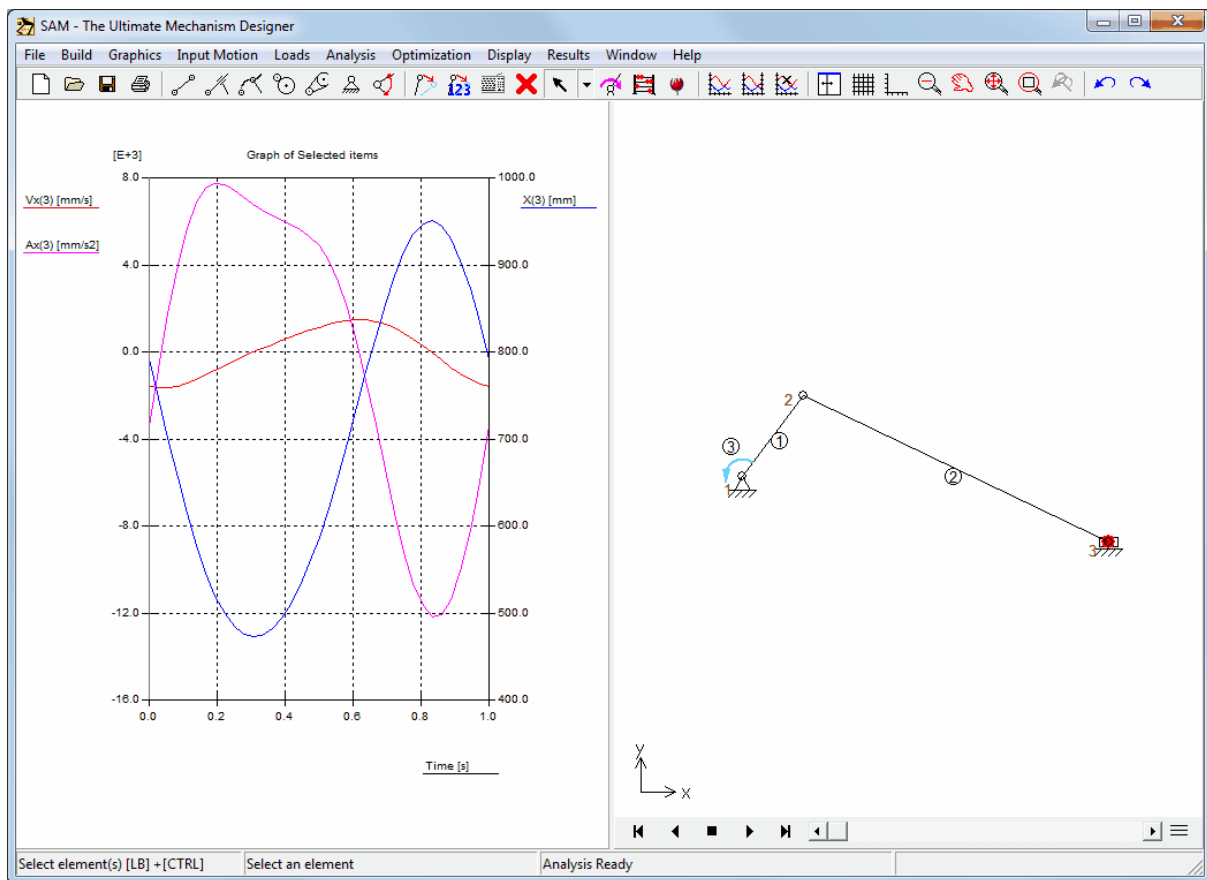


a. X-position, X-velocity and X-acceleration of slider as function of timeb. Mechanism

Choose "Toggle Curve" from Results menu or click 

Next, click on X(3)-label in graph-window

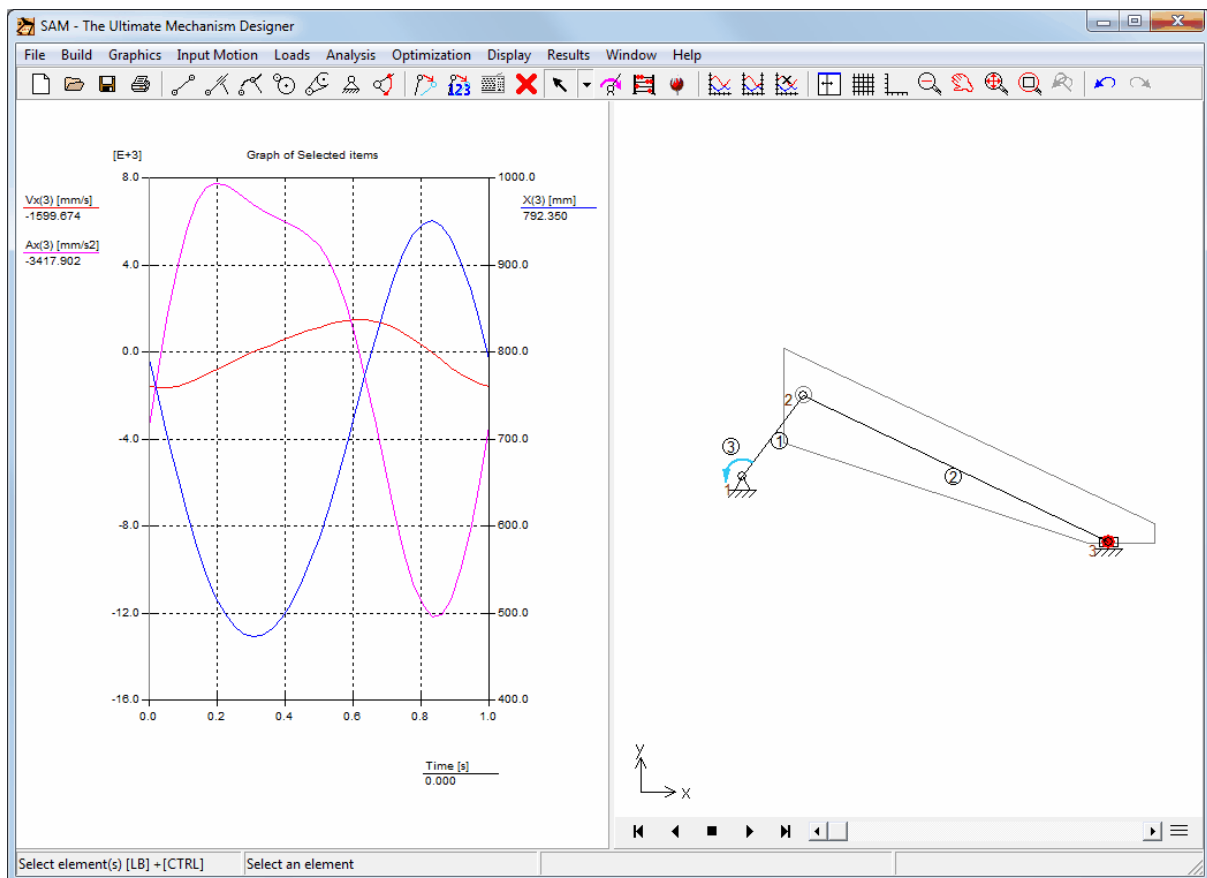
The X(3)-label has moved to the right side and the X-displacement X(3) of node 3 is scaled according to the right y-axis scaling (this double y-scaling mechanism is especially useful in case the range of the displayed variables differs significantly).



a. X-position, X-velocity and X-acceleration of slider as function of time (2 different y-scalings) b. Mechanism

Choose Import DXF in the File menu and select the file "SL.DXF".

The DXF-file will be imported and all graphical items of that file will be automatically treated as one group. Your screen will look like this:



Screen shot after importing the DXF-file "SL.DXF"

Choose Attach Group in the Graphics menu and follow the instructions.

First you have to select the group of graphical items that you want to attach (click the DXF-data that you just imported), then you have to click on the element to which you want to attach that group (click element 2).

Select Animation in the Display menu or click 

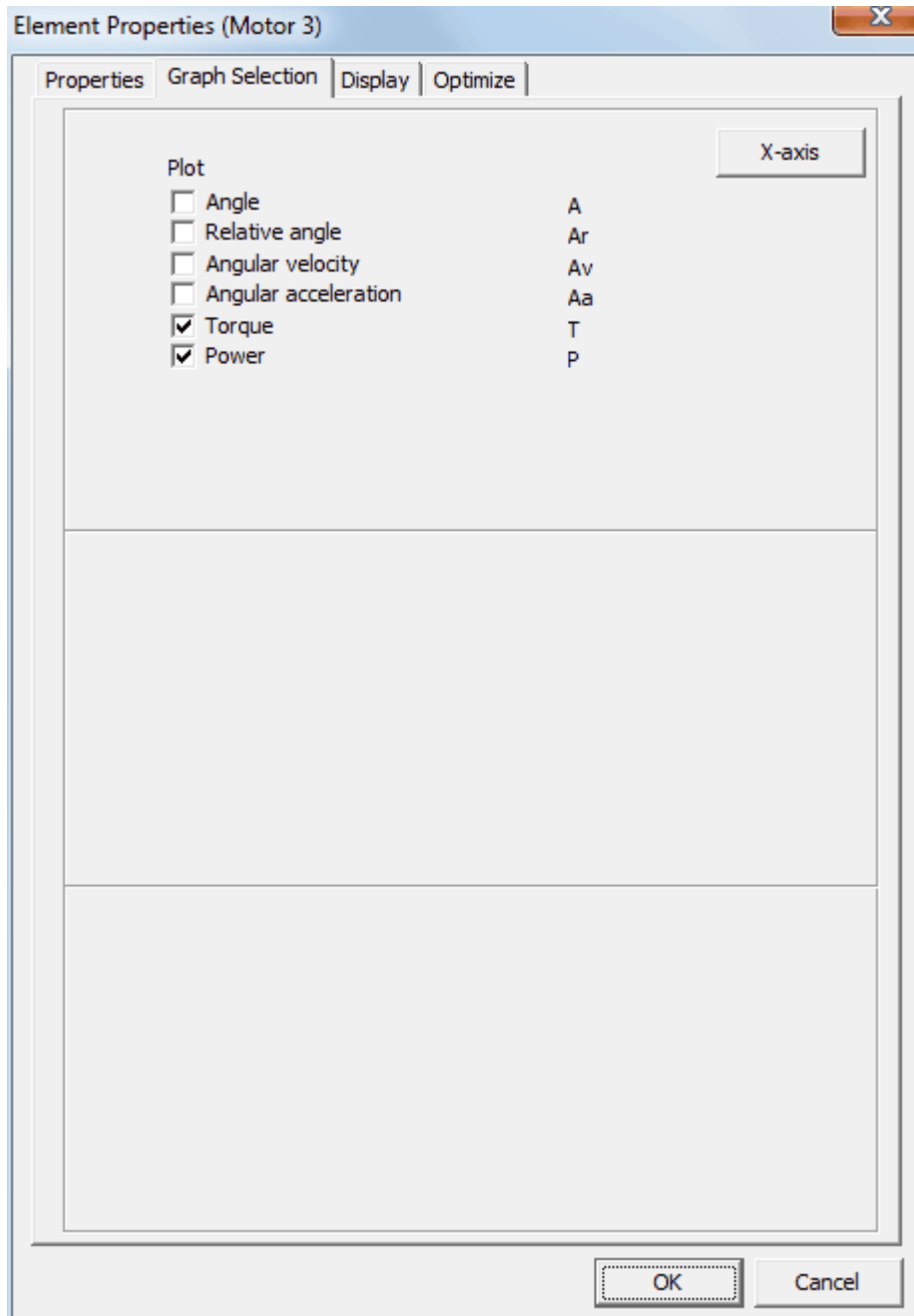
You will now see an animation in which the graphical items move together with element 2.

Force Analysis

If you want you can remove the DXF graphics since it is not essential for the force analysis. First we will clear all results and then look at the required driving torque and the required driving power of the motor element. Then, we will look at the bearing forces of

element 1 in node 1 and node 2.

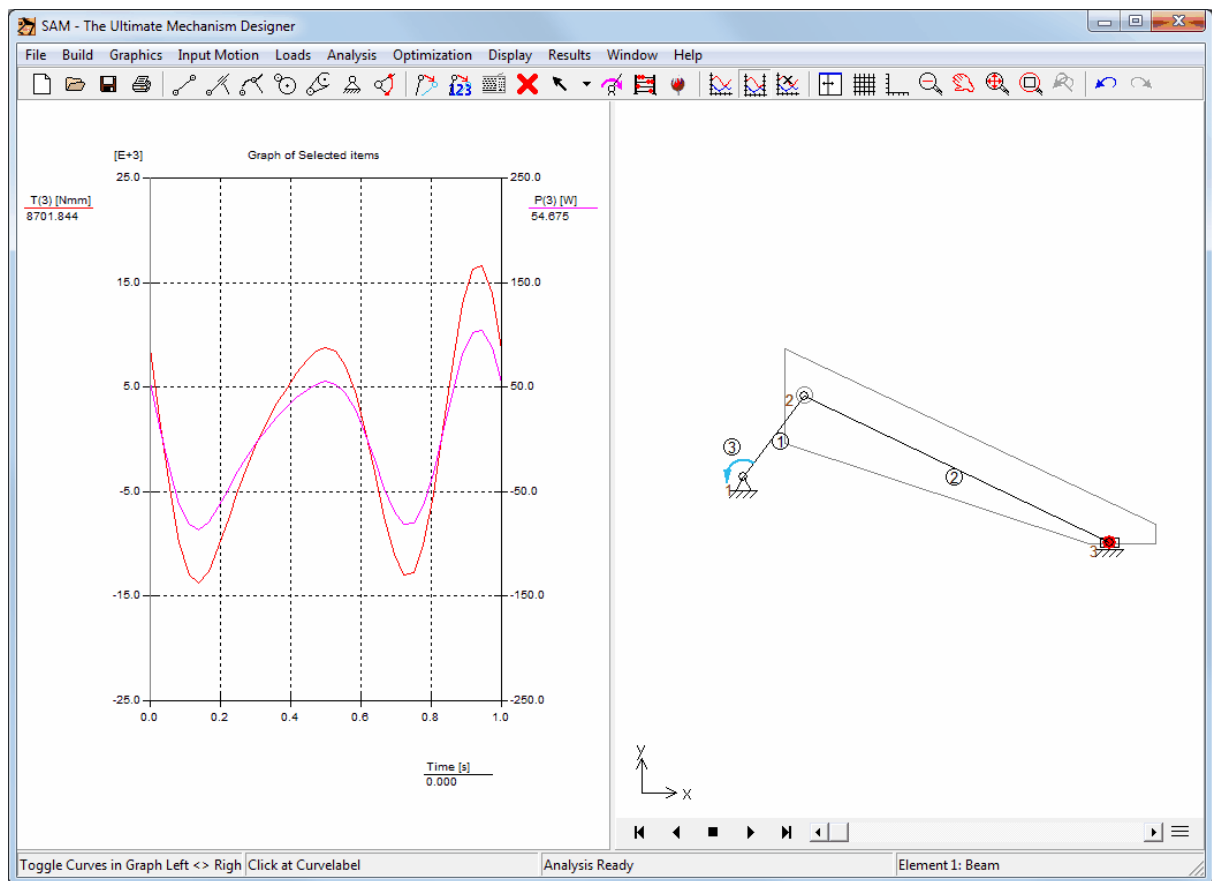
Results>Clear Selection, then select element 3 and activate the items shown below



Element properties dialog box

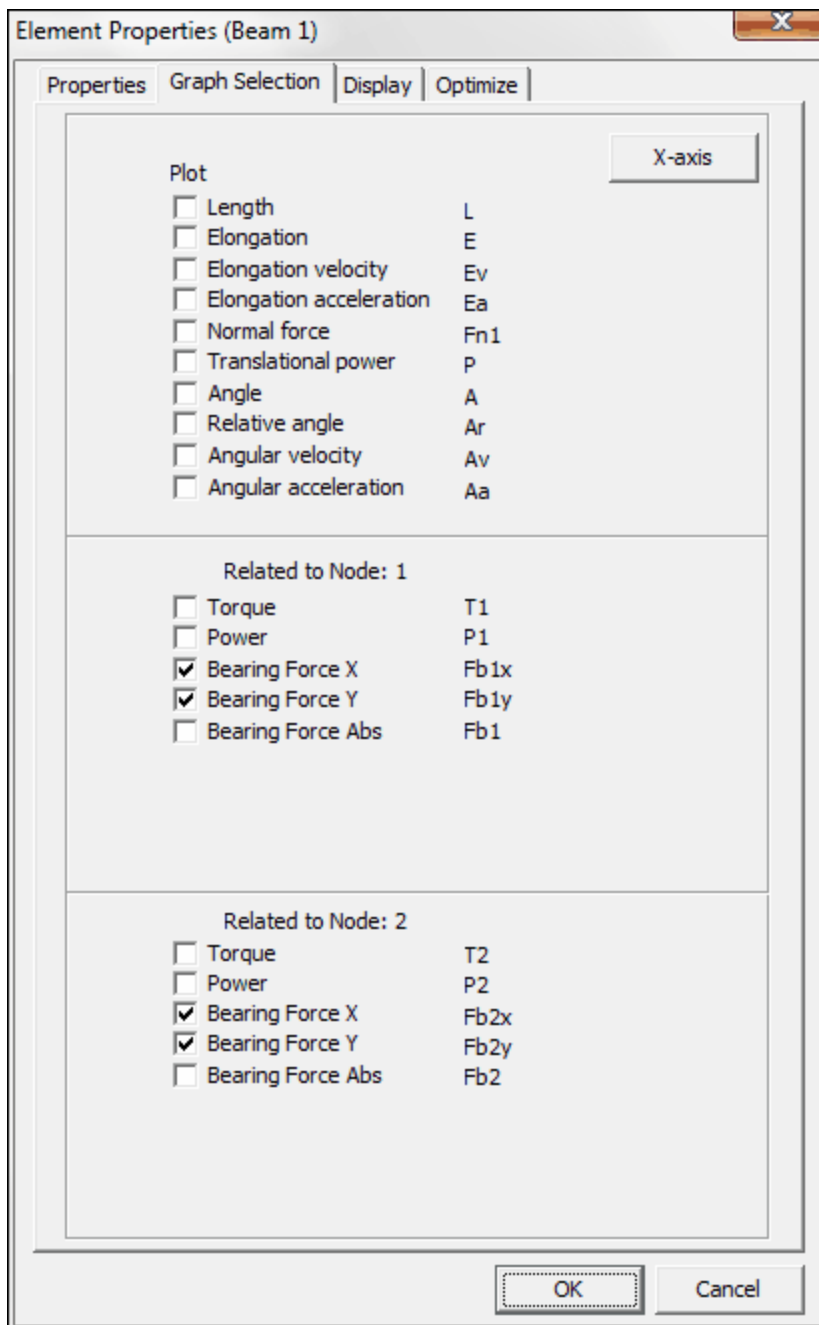
You will now see a plot of the driving torque and driving power as function of time. Again, due to difference in range of the corresponding results, it is best to have separate

scaling.

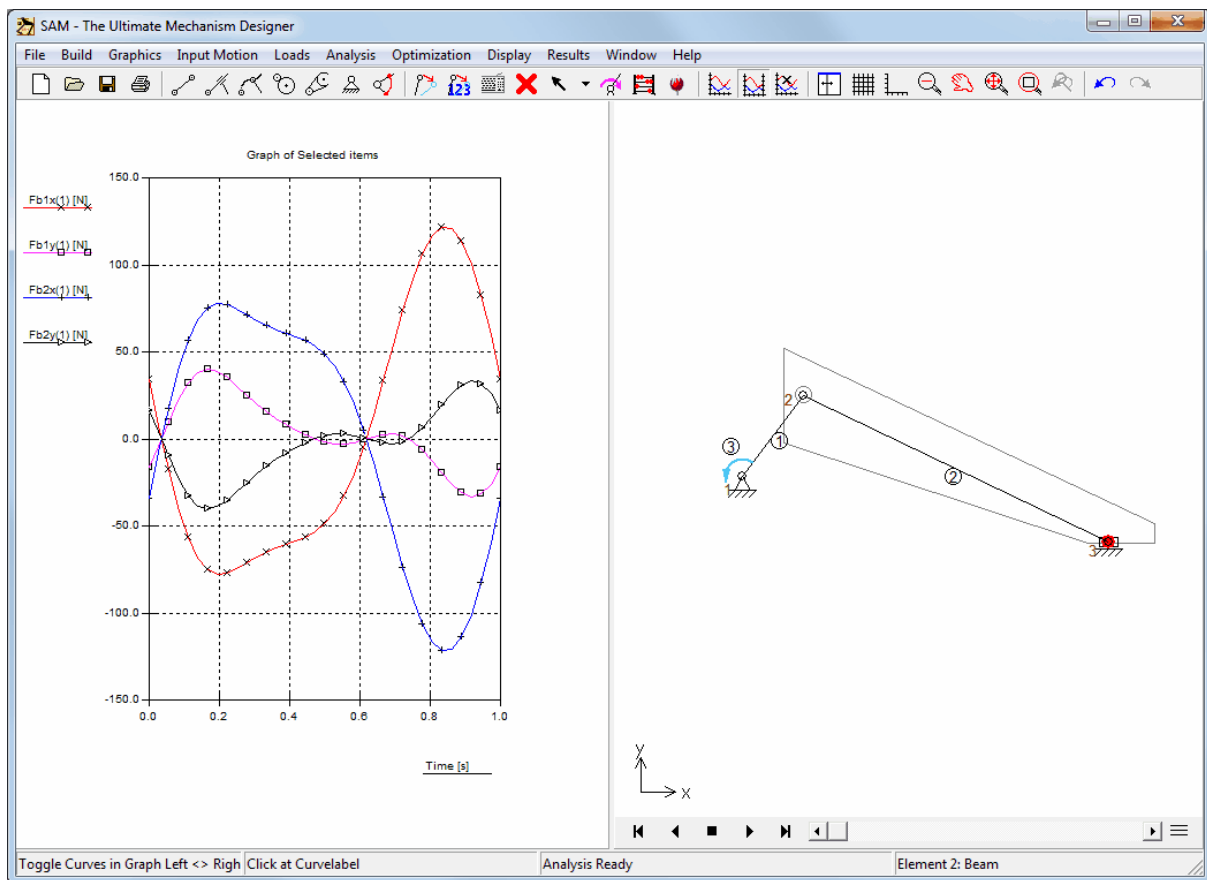


Screen shot showing the mechanism plus the driving torque and driving power

Next we will look at the x/y bearing forces of element 1 at the nodes 1 and two. In order to do so we need to make the following selection of results:



Element properties dialog box



Screen shot showing the mechanism plus the x/y bearing forces of element 1 in nodes 1 and 2 (for better visibility symbols have been added to each of the curves)

This is the end of our first quick tour based on an existing mechanism project file. In the next chapters you will learn how to design a mechanism based on the Design Wizard and how to "manually" build your own mechanisms.

4.2 Using the Design Wizard to create a mechanism

SAM offers a set of design wizards which will help you to synthesize mechanisms for specific tasks. We will shortly introduce you to the concept of the design wizard based on the design task to create a 4-bar mechanism that guides the coupler plane in such a way that 3 specific positions + angles are satisfied.

Select File/Wizard/4-Bar Mechanism

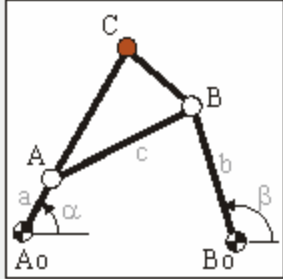
A dialog box will appear with 5 tabs related to specific design tasks that can be fulfilled by a 4-bar mechanism.

4 Bar Mechanism Wizard

Angle Function Synthesis | **3-Position Synthesis (I)** | 3-Position Synthesis (II)

General (nodes) | General (nodes,angle,dimension)

	X	Y
C	600.000 [mm]	900.000 [mm]
A	300.000 [mm]	700.000 [mm]
B	800.000 [mm]	800.000 [mm]
Ao	200.000 [mm]	500.000 [mm]
Bo	800.000 [mm]	400.000 [mm]



This dialog allows you to generate any 4-bar mechanism by specifying all node coordinates.

After OK you can return to this wizard via F8

OK Cancel

Design Wizard "4-Bar Mechanism"

Choose "3-Position Synthesis (I)"

This brings the "3-Position Synthesis (I)" tab to the front position. In this dialog you can specify 3 positions/angles of a the coupler plane plus the location of the fixed nodes Ao and Bo. As a first introduction we suggest to accept the default values.

4 Bar Mechanism Wizard

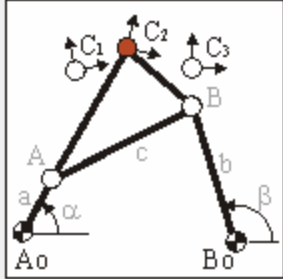
General (nodes) | General (nodes,angle,dimension)

Angle Function Synthesis | **3-Position Synthesis (I)** | 3-Position Synthesis (II)

List of goal positions and angles for Point C

	x [mm]	y [mm]	angle [deg]
C (1)	250.000	400.000	20.000
C (2)	350.000	450.000	0.000
C (3)	500.000	400.000	-5.000

Ao [mm] [mm]
 Bo [mm] [mm]



Based on the three specified positions/angles of the coupler point C and the base pivots Ao and Bo, a bar mechanism is synthesized (Burmester approach). Some combinations of specified inputs are such that the resulting mechanism can not reach all positions without being dis-assembled. Also it might be necessary to

After OK you can return to this wizard via F8

OK Cancel

Design Wizard "4-Bar Mechanism - 3-Position Synthesis (I)"

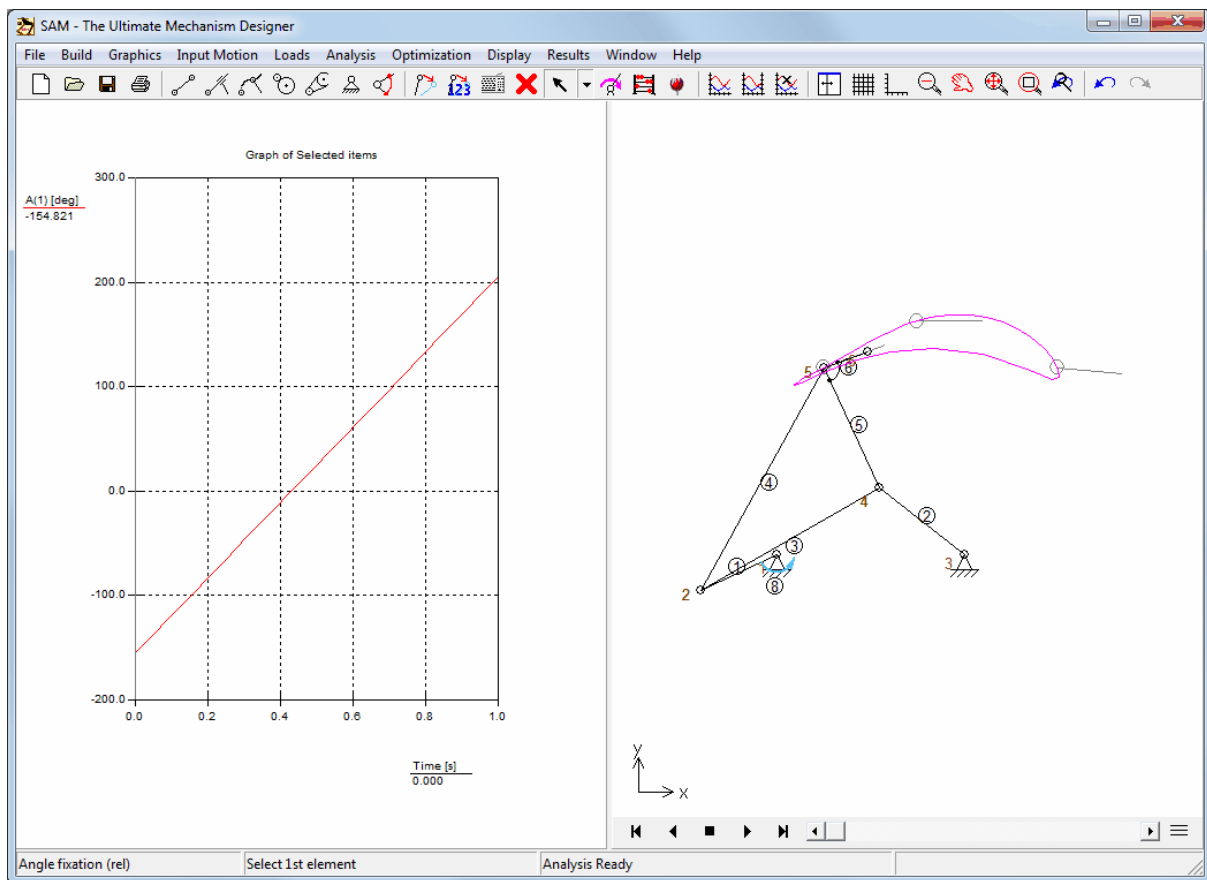
Press OK to accept the current values

Based on the three specified positions/angles of the coupler point C and the base pivots Ao and Bo, a 4-bar mechanism is synthesized, analyzed (based on a certain input motion) and the path of the coupler point (+ the 3 specified goal positions/angles) is displayed in the mechanism window. Initially, the graph window shows the crank angle as function of time.

Result of Design Wizard "4-Bar Mechanism - 3-Position Synthesis (I)"

Press  (Zoom max) for a better view of the mechanism followed by one of the Animation icons

Your screen will now look like this and the mechanism is animated.

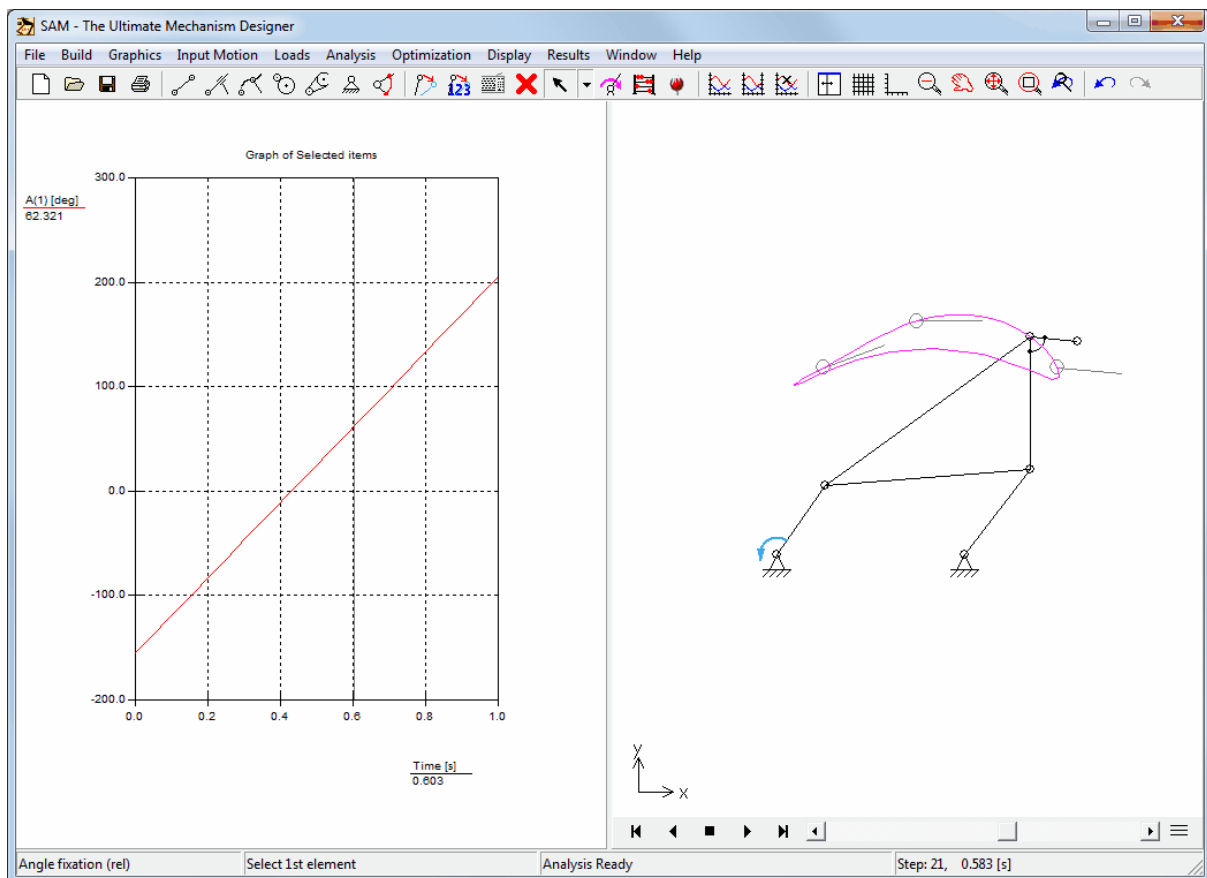


Result of Design Wizard "4-Bar Mechanism - 3-Position Synthesis (I)"

Stop the animation and move mouse over graph windows

Moving the mouse over the graph window will result in a displacement of the vertical cursor line and the mechanism will move accordingly. In this way, you can do a detailed study of mechanism's motion.

If the mechanism is not moving together with the vertical line in the graph you should press F4 (File>Preferences) and switch the Graph-Cursor-Link ON.



Result of Design Wizard "4-Bar Mechanism - 3-Position Synthesis (I)"


Press <F8> to return to the Design Wizard.

If the outcome of the synthesis is not to your satisfaction, you can return to the design wizard and try another sets of input data. In this specific 3-Positions design task the starting and end position is generally fixed but you could "play" with the middle position/angle. Also you might choose different locations for your based pivots A_o or B_o .



4.3 Building a new mechanism (without Design Wizard)

These quick steps provide an overview of how to use SAM to create and analyze a new mechanism. Although the steps you take may differ depending on the type of mechanism and the type of analysis, the basic steps are as follows :

1. Create new project

Choose New from the File menu to open a new project or use the equivalent speedbutton . You will be prompted to specify the initial dimension of the working area (may be changed in a later stage).

2. Assemble mechanism from basic elements ...

Using the various menu items from the Build menu or the tools from the toolbar you can assemble a mechanism from basis elements. Double-click an element to change its properties or select Delete Element from the Build menu to delete an element. In order to move a node select the Move Node or Node Coordinate option in the Build menu. Alternatively you can use  or .

3. Define support conditions

Support conditions can be defined either from the Build menu or by clicking the Support tool in the toolbar. You will be asked to select a node by clicking on it. Now move the cursor around the node and watch how the support condition changes from x-support to xy-support to y-support and so on. When your desired support type is displayed, simply press the left mouse button in order to activate it.


4. Define inertia's, external forces, gravity

Define inertia's, external forces and gravity from the Force menu.

5. Specify the input motion(s)

Input motion can be defined either from the Input Motion menu or by clicking the corresponding icon in the toolbar. You can define various kinds of input type, such as x-displacement, y-displacement, angle, relative angle and elongation. After selection of the appropriate node or element, an input dialog box is opened, which lets you define the input motion by combining basic motion laws, such as constant, linear, sinusoidal and polynomial motion (please refer to the corresponding section in the chapter "Input Menu").

6. Analysis

Depending on the settings of AUTO-RUN (see File/Preferences/Analysis) an analysis is performed automatically as soon as all relevant data of the mechanism is available or you need to start the analysis manually (by pressing Analysis or clicking the Abacus ).

7. Animate Mechanism

Choose Animation from the Display menu or click the Windmill tool to start the animated display. You can also animate the mechanism "manually" or move it to a new initial position via the menu Build/Change_Initial_Position.

8. View Analysis Results

Choose Select from the Results menu to select analysis results for further post-processing.. Next, choose Export to display the selected data in tabular format or Graph to view an xy-plot of the selected data as a function of time. Try to click the y-label in the graph and see what happens.

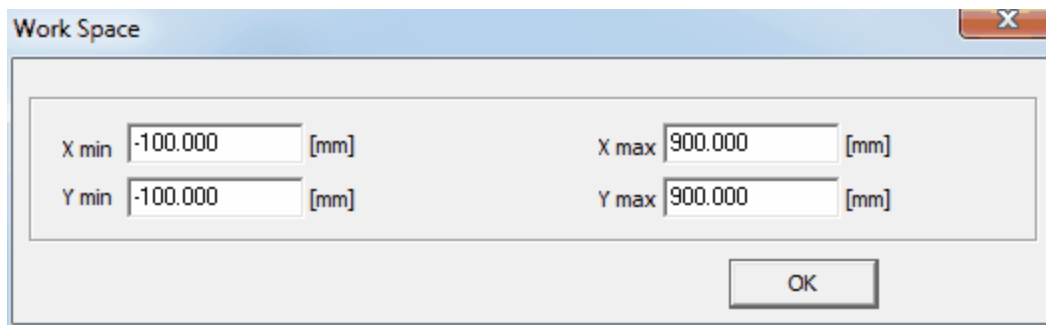
9. Save Project

Choose Save from the File menu to save the project.

4.3.1 Example of a 4-bar mechanism (motion analysis)

Select New from the File menu

A Work Space dialog box will appear in which you can enter the dimension of your workspace. The numbers that you see in the dialog box are the default values. These can differ from the numbers used during the preparation of this manual.



Work Space dialog box

Click OK to accept the default values of the workspace.

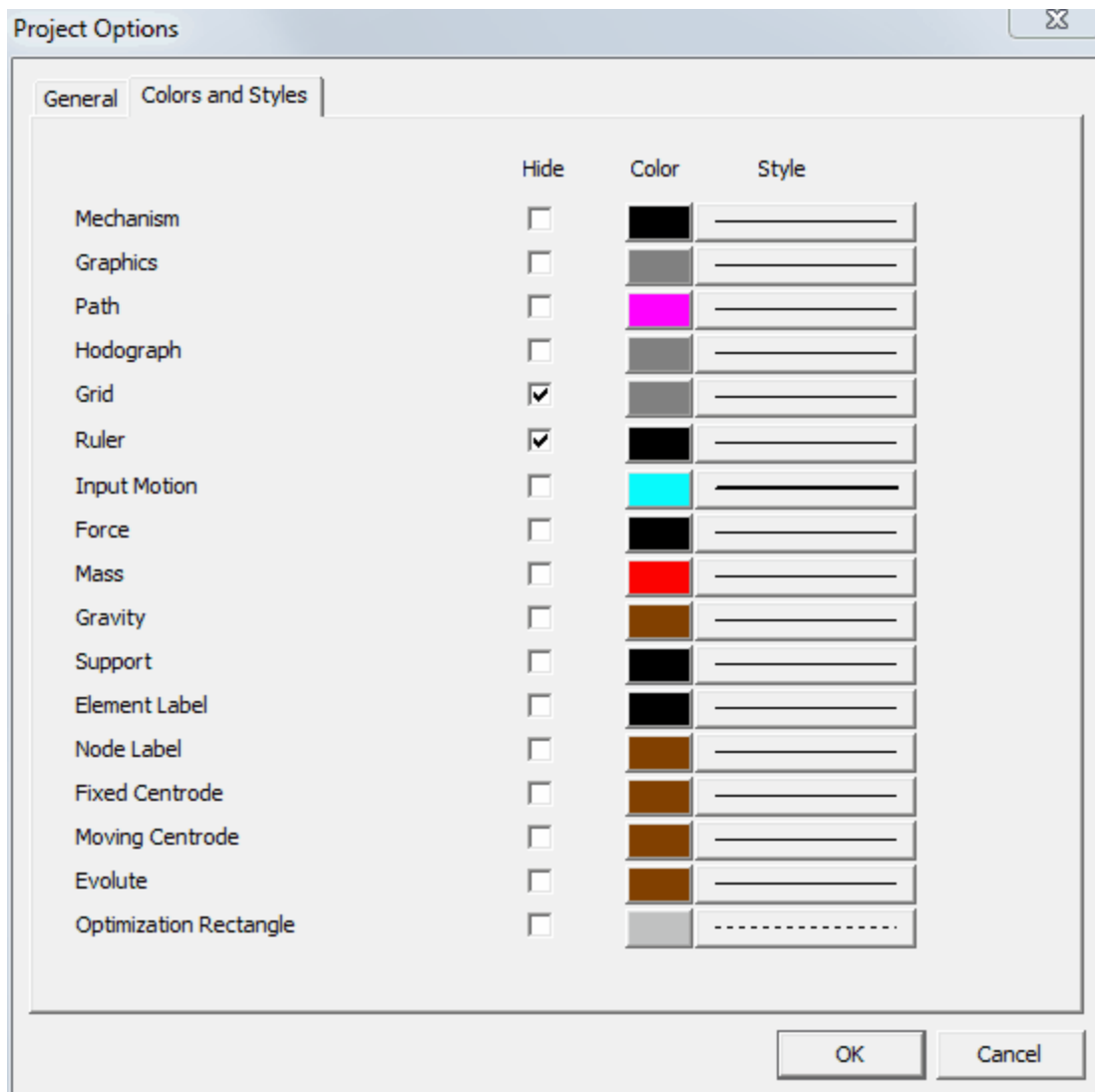
You will now see an empty workspace with a graph-window and a mechanism window.

Maximize the mechanism-window by pressing the corresponding button.

Using a maximized mechanism-window is very practical while you are building your mechanism.

Choose Options in the Display menu and select the tab Colors and Styles.

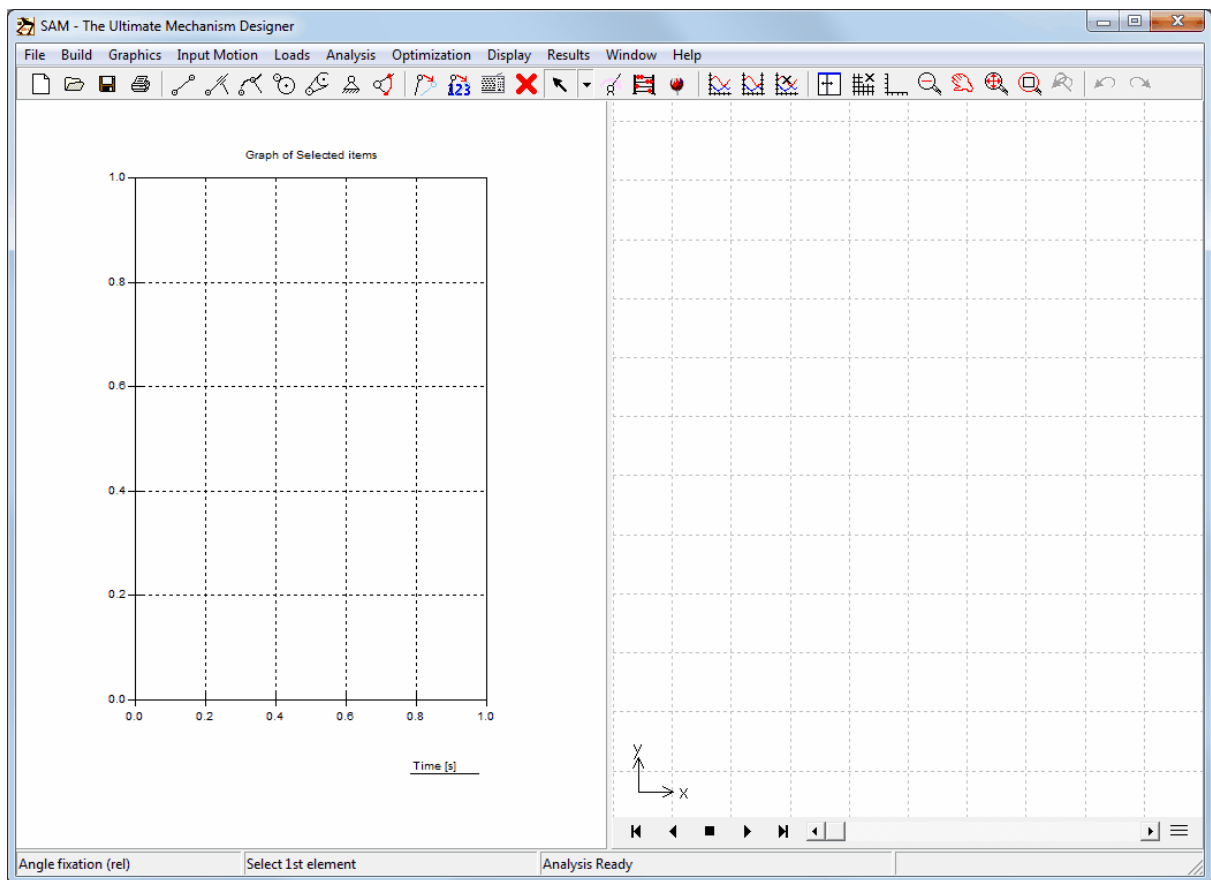
In the dialog box that will appear you can customize the display of various items. As you can see the Grid and the Ruler are switched off (Hide=on).



Display Options dialog box

Unhide the display of the Grid, choose any color and style that you want and click the OK button

This will turn the grid on, using the default grid size which is related to the dimension of the workspace. Your screen will now look like this :



Workspace with default grid

Select Beam from the Build menu or click 

You can now start creating a beam element.

Move the mouse to the starting point of the element and press the left button to create the first node.

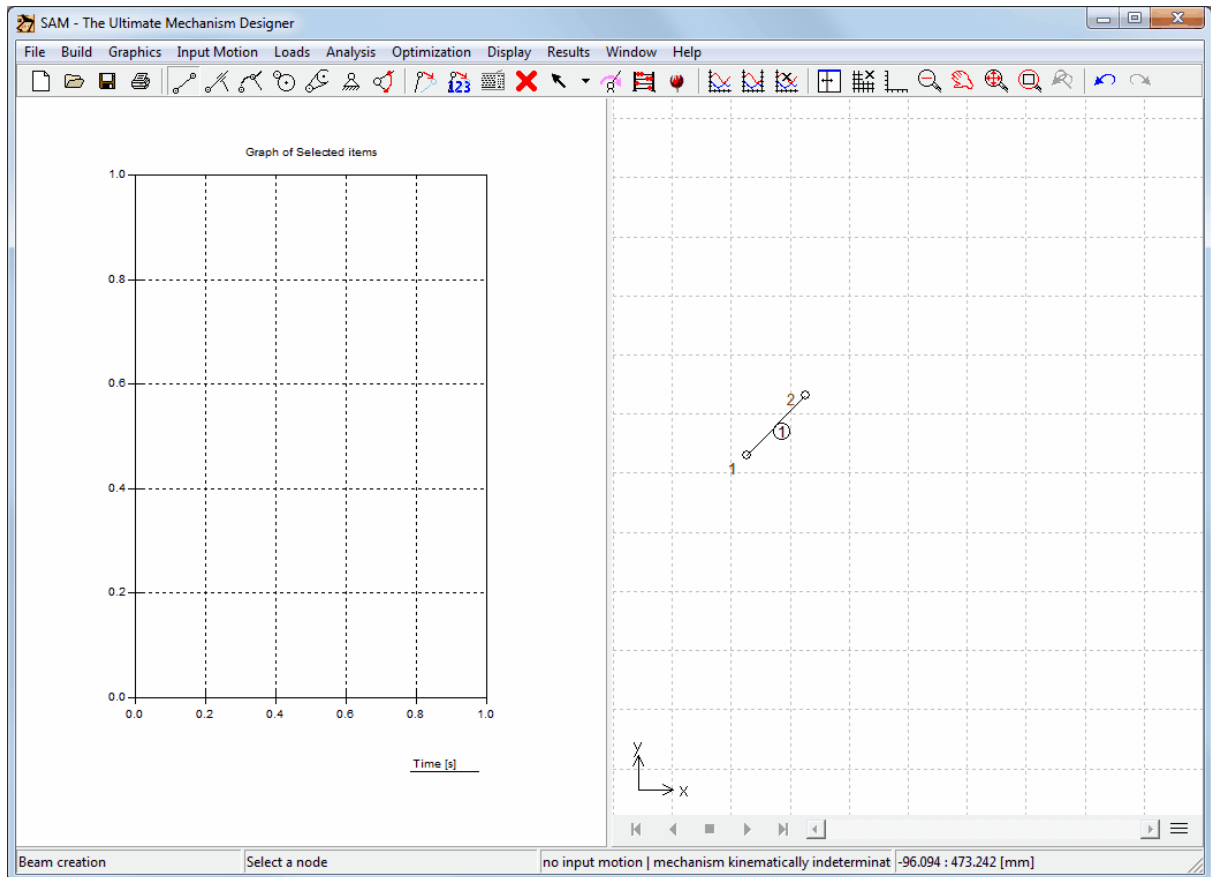
While you move the mouse you can see the current mouse coordinates in the status line at the bottom of the windows.

Move the mouse to the end point of the element and press the left button to finish creating a beam element.

While moving the mouse the current shape of the element will be displayed. If you want to cancel the creation process halfway (first node is already chosen but not the second node) press the right mouse button. Once an element has been created and you want to

delete it again, you can select *Delete* from the *Build* menu and click the element that you want to delete.

Your screen should now look similar to this :

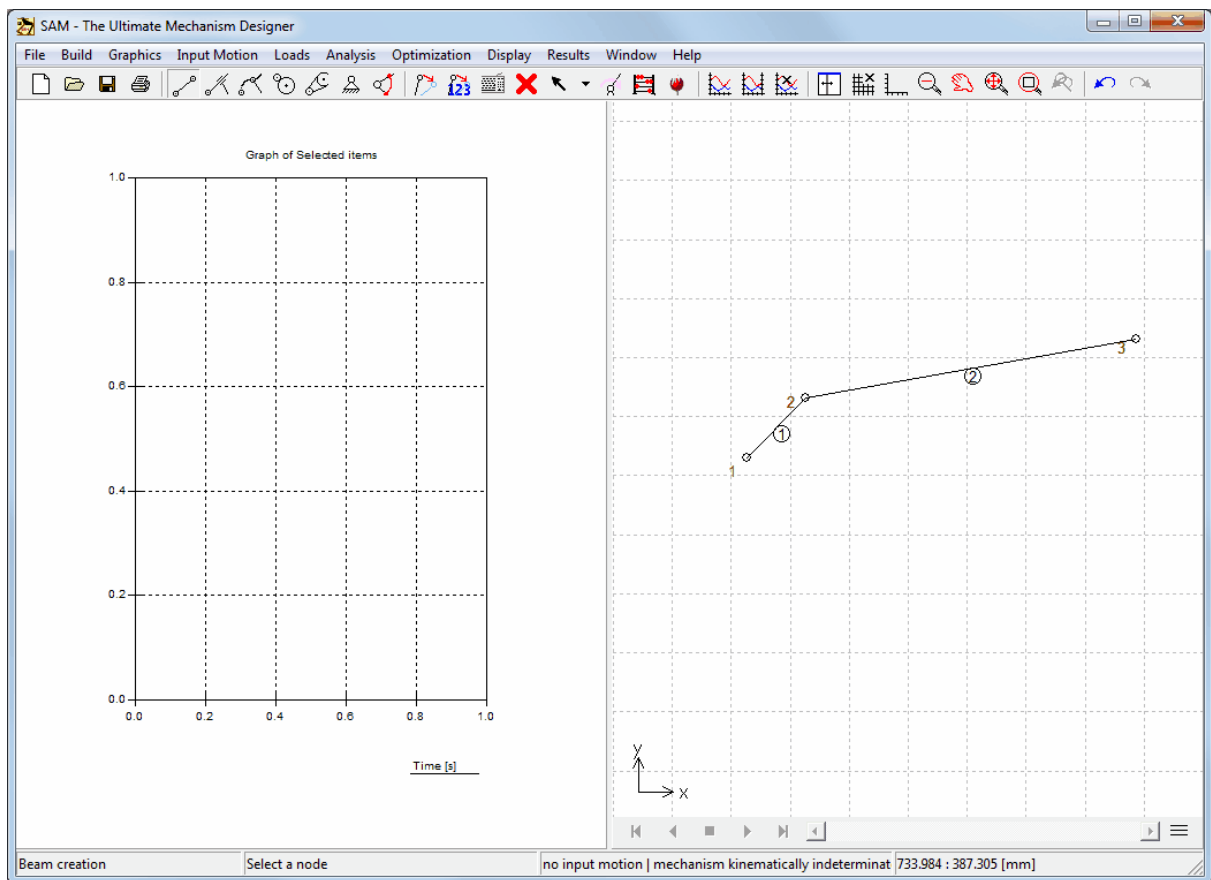


Workspace after creation of first element.

Create second linkage. Move mouse to node 2 of first element and click once, then move mouse to a new position and click again.

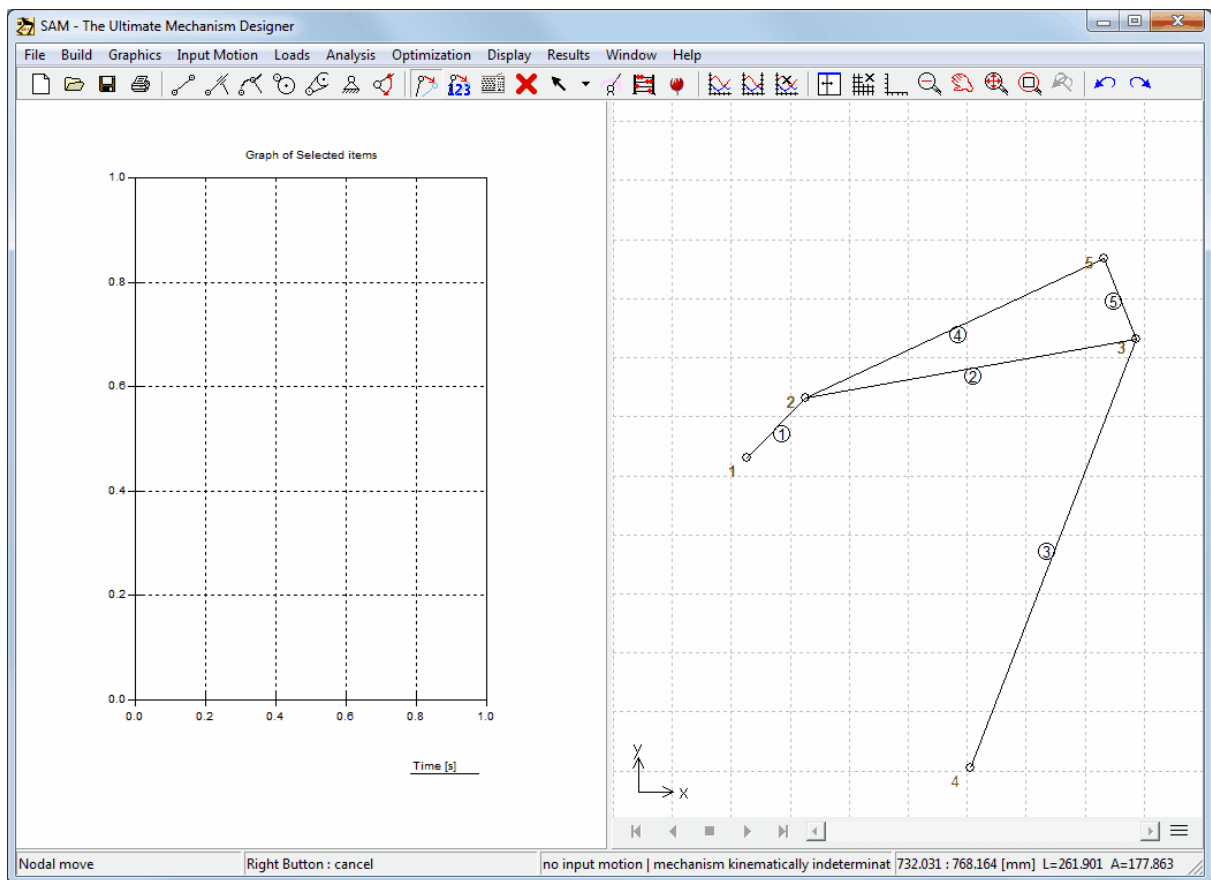
The second linkage is created in the same way, except that one of the nodes of the new element has to coincide with an already existing node of the first element in order to create a connection.

As the mouse comes near an existing node the shape of the cursor changes to indicate that it has snapped to that node. When the mouse is clicked the existing node will be used instead of creating a new one.




Workspace after creation of second linkage

Create all other linkages of 4-bar mechanism (including a coupler point) in the same way.



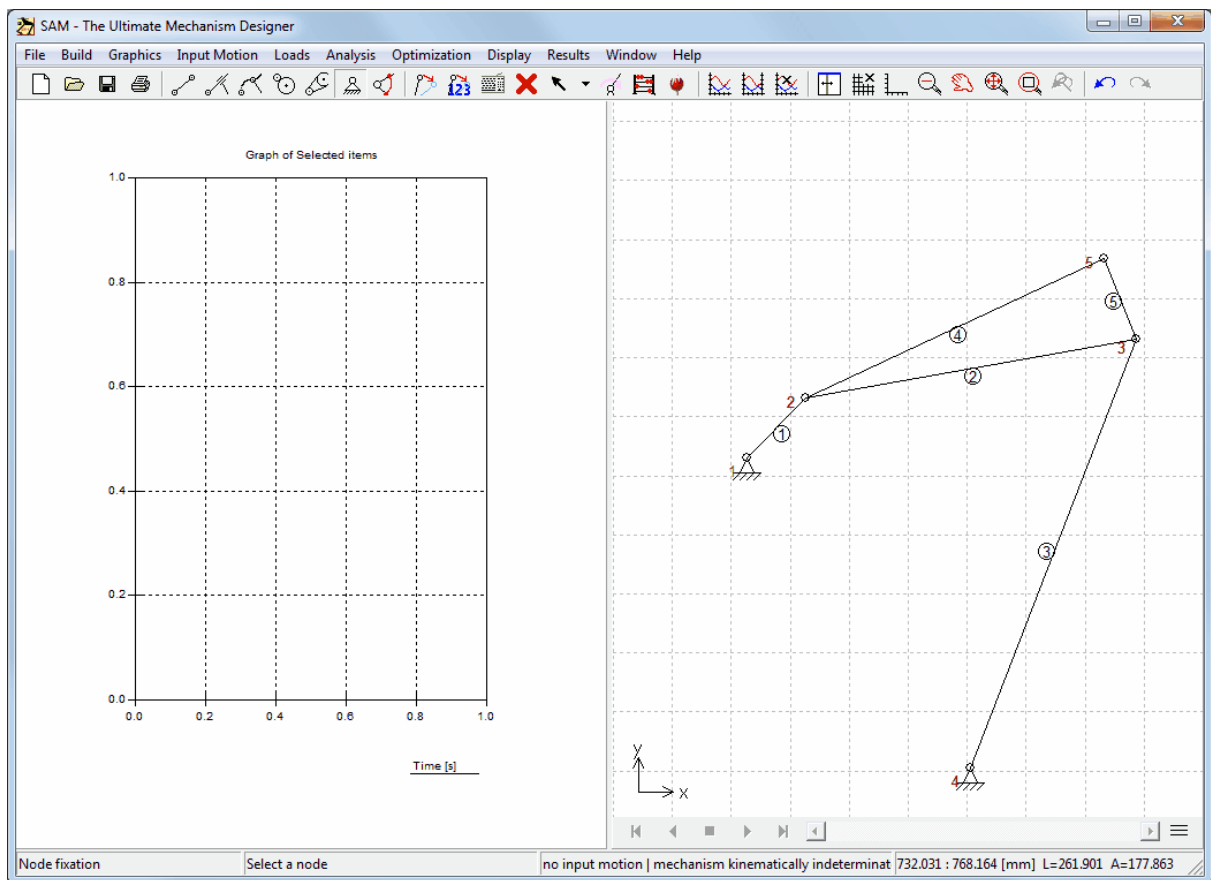
All elements of the 4-bar mechanism have been assembled

Select "Fix Node (x and/or y)" in the Build menu or click . Next, click on node 1 and move around the node until the xy-support condition is displayed. Then press the left mouse button again.

As you will see the support condition changes from x-support to xy-support to y-support and so on depending on the position of the cursor with respect to the node that you have selected. This is a very practical way of applying support conditions.

Apply the same support conditions to node 4.

This will result in the following situation on your work space.



4-bar mechanism including support conditions

Select Angle from the Input Motion menu or click  and click on node 1.

The Input Motion dialog box will appear. The heart of the input dialog box is formed by a list box, which contains a sequential list of selected basic motion blocks (the graph shows the equivalent of these data). Initially, this list is empty.

Input Motion

Polynomial 5th order | File | Spline | Trapezoidal-1 | Trapezoidal-2
Linear | Quadratic | Cycloidal | Polynomial 345

Parameters	Value	Units
Motion	360.000	[deg]
Time	1.000	[s]
Intervals	36	[-]

Add Insert Modify

List of actual curve-parts:

Nr	Type	Time	Intervals

Delete Clear All Clipboard Print OK Cancel

Angle [deg]

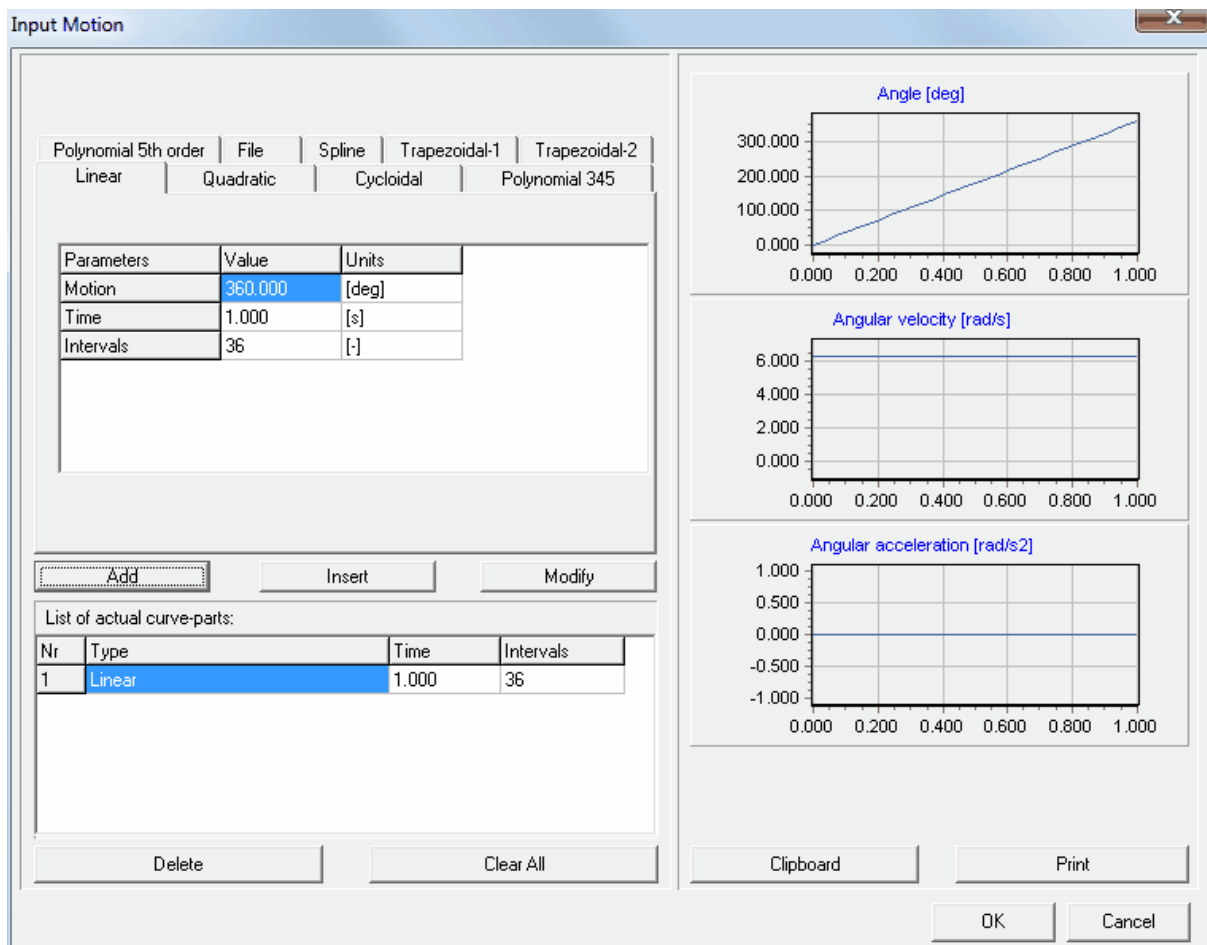
Angular velocity [rad/s]

Angular acceleration [rad/s²]

Input Motion dialog box

Click the "Add" button

This will add the currently active motion law and the current motion data (motion, duration and number of steps) to the motion list. Furthermore, the graphical representation will be updated. By repeating this process, it is very easy to define all sort of input motions by combining basic motion laws. If you want to delete an item from the list, simply activate it by clicking it once and press the "Delete" button. In order to modify an entry you have to activate it and to press the "Modify" button.



Input Motion dialog after a standard motion law has been added to the motion list

Press OK to accept the current input motion.

Press any of the Animation keys

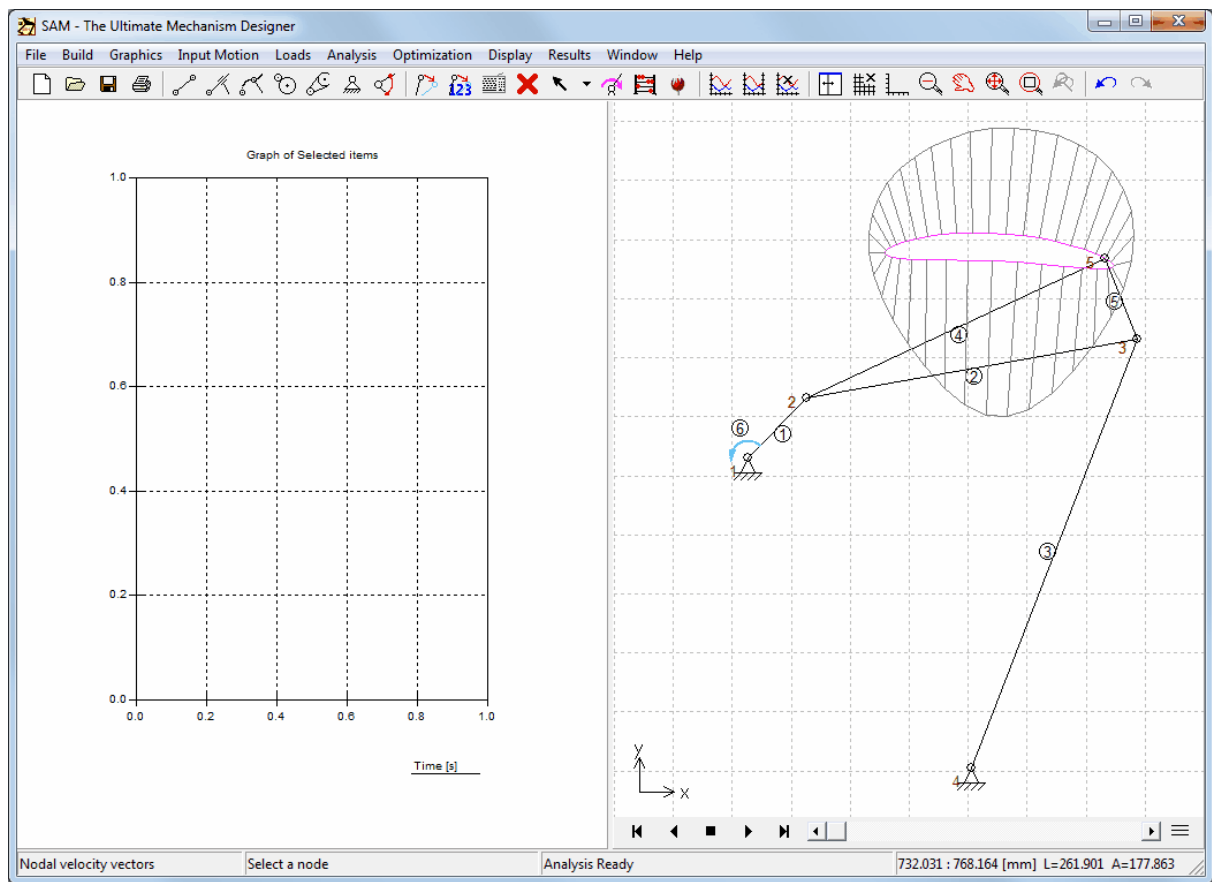
You will now see an animated display of the mechanism.

Select Path in the Display menu and click the coupler point (node 5)

You will now see the path of that point.

Select Hodograph in the Display menu and click again the coupler point.

Apart from the path of that node, you will also see the velocity hodograph.



Path and velocity hodograph of coupler point

Chapter



5

5 Design Wizards

5.1 4-Bar Mechanism

The menu File>Wizard>4-Bar Mechanism invokes the dedicated 4-bar mechanism wizard, which allows you to generate a general 4-bar mechanism in a very easy way and to perform angle function synthesis and 3-position synthesis.

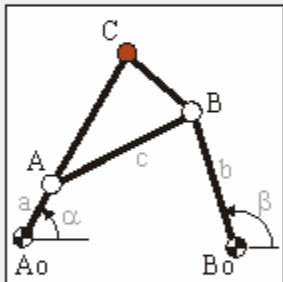
5.1.1 General (nodes)

4 Bar Mechanism Wizard

Angle Function Synthesis | 3-Position Synthesis (I) | 3-Position Synthesis (II)

General (nodes) | General (nodes,angle,dimension)

	X	Y
C	600.000 [mm]	900.000 [mm]
A	300.000 [mm]	700.000 [mm]
B	800.000 [mm]	800.000 [mm]
Ao	200.000 [mm]	500.000 [mm]
Bo	800.000 [mm]	400.000 [mm]



This dialog allows you to generate any 4-bar mechanism by specifying all node coordinates.

After OK you can return to this wizard via F8

OK Cancel

4-Bar Mechanism Wizard: General (nodes)

This dialog allows the user to generate a 4-bar mechanism including a coupler point by specifying the coordinates of all nodes.

5.1.2 General (nodes, angle, dimension)

4 Bar Mechanism Wizard

Angle Function Synthesis | 3-Position Synthesis (I) | 3-Position Synthesis (II)

General (nodes) | General (nodes,angle,dimension)

Alfa [deg]

a [mm]

b [mm]

c [mm]

u [mm]

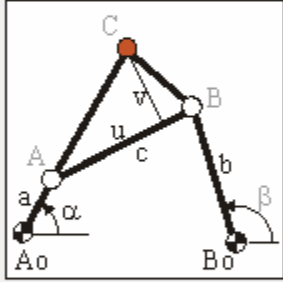
v [mm]

☒ B at leftside of A - B0

X Y

Ao [mm] [mm]

Bo [mm] [mm]



This dialog allows you to create any 4-bar mechanism by specifying the ground pivots Ao ,Bo, the crank angle alfa and the element dimensions a, b, c, u and v.

After OK you can return to this wizard via F8

OK Cancel

4-Bar Mechanism Wizard: General (nodes, angle, dimensions)

This is a variation of the previous generation dialog. The 4-bar mechanism is defined via the ground pivots Ao and Bo, the crank angle α and the linkage lengths a, b, c and the dimensions u and v, which determine the location of the coupler point.

5.1.3 Angle Function Synthesis

4 Bar Mechanism Wizard

General (nodes) | General (nodes,angle,dimension)

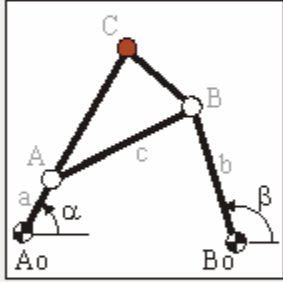
Angle Function Synthesis | 3-Position Synthesis (I) | 3-Position Synthesis (II)

	alfa [deg]	beta [deg]
1	30.000	35.000
2	40.000	37.000
3	70.000	42.000
4		
5		
6		

X Y

Ao 200.000 [mm] 500.000 [mm]

Bo 800.000 [mm] 400.000 [mm]



Based on the specified pairs of input/output angles a 4 bar mechanism is synthesized (Freudenstein). Some combinations of specified input/output angles are such that the resulting mechanism can not reach all positions without being dis-assembled. In such a case, a new equivalent solution is found by shifting all input or output angles.

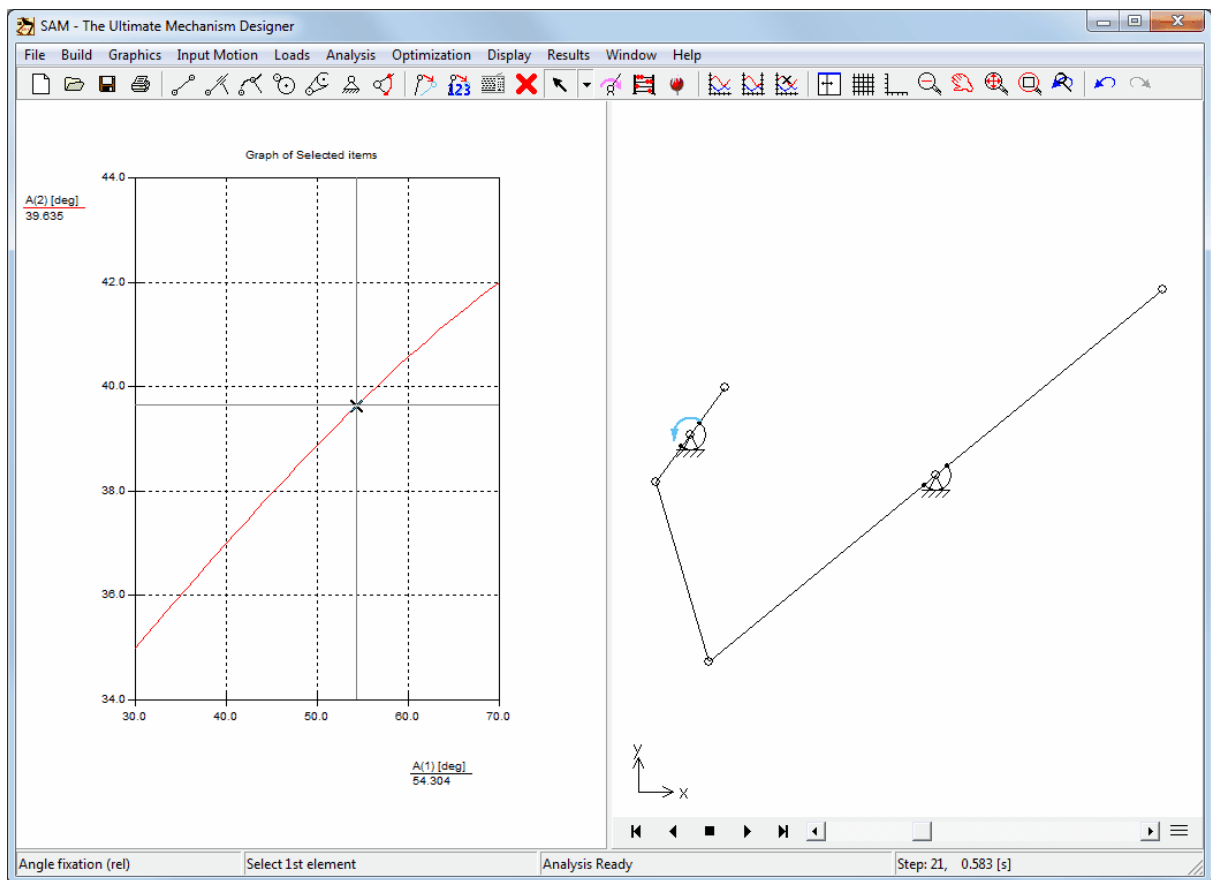
After OK you can return to this wizard via F8

OK Cancel

4-Bar Mechanism Wizard: Angle Function Synthesis

Based on the specified pairs of input/output angles, a 4-bar mechanism is synthesized via the Freudenstein approach. In case of 3 pairs of input/output angles, which is the minimum, a 4-bar mechanism is sought that exactly satisfies the requirements. The user can also specify more than 3 pairs of input/output angles, in which case the requirements will only be met approximately (least squares approximation).

Some combinations of specified input/output angles are such that the resulting mechanism can not reach all positions without being dis-assembled in between positions. In such a situation, the user can attempt to look for an alternative solutions by shifting all input or all output angles and correcting this shift by adding an extra beam to the crank to compensate for the offset-angle.



Result of angle function generation (based on default values)

5.1.4 3-Position Synthesis (I)

4 Bar Mechanism Wizard

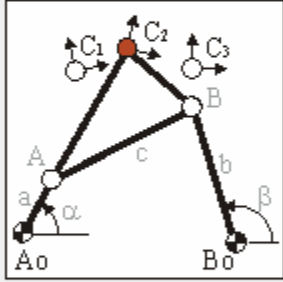
General (nodes) | General (nodes,angle,dimension)

Angle Function Synthesis | 3-Position Synthesis (I) | 3-Position Synthesis (II)

List of goal positions and angles for Point C

	x [mm]	y [mm]	angle [deg]
C (1)	250.000	400.000	20.000
C (2)	350.000	450.000	0.000
C (3)	500.000	400.000	-5.000

X Y
 Ao 200.000 [mm] 200.000 [mm]
 Bo 400.000 [mm] 200.000 [mm]



Based on the three specified positions/angles of the coupler point C and the base pivots Ao and Bo, a 4-bar mechanism is synthesized (Burmester approach). Some combinations of specified inputs are such that the resulting mechanism can not reach all positions without being dis-assembled. Also it might be necessary to

After OK you can return to this wizard via F8

OK Cancel

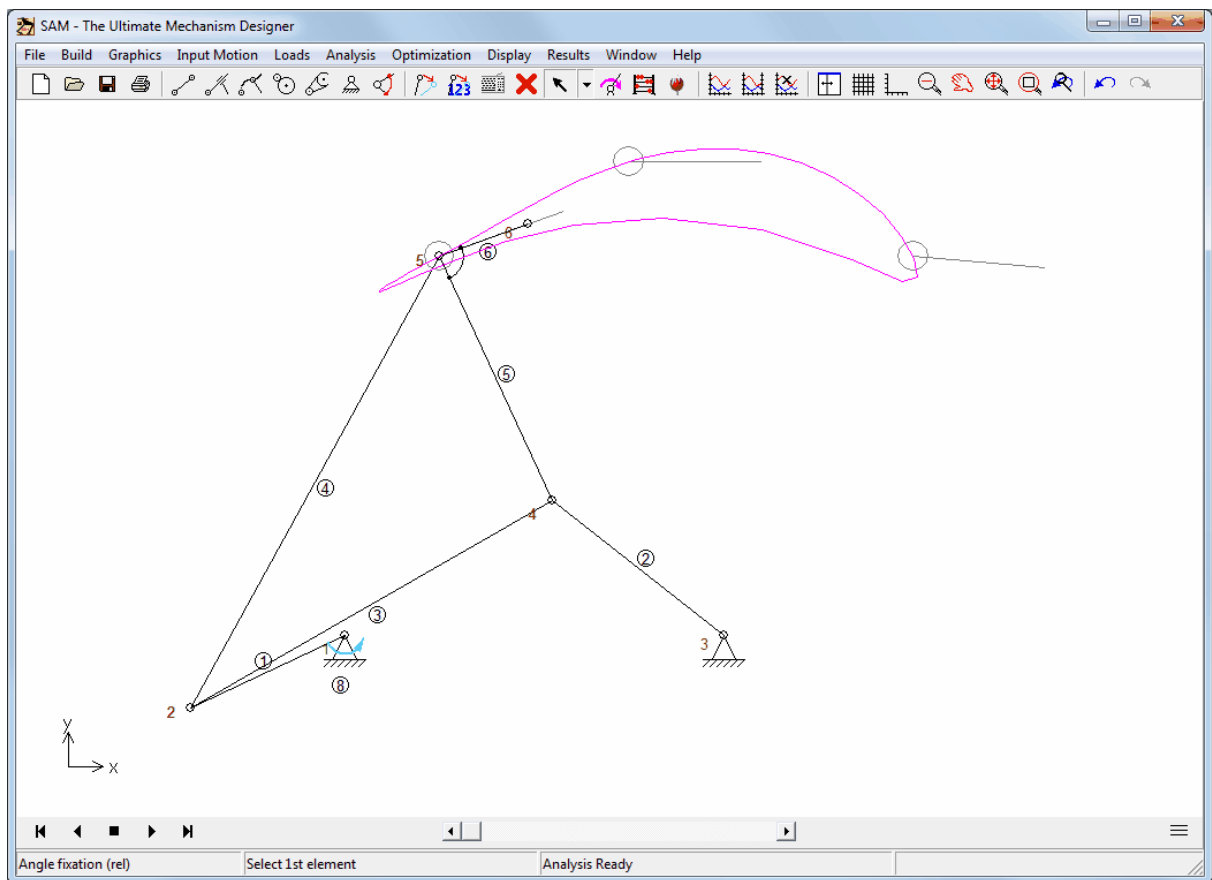
4-Bar Mechanism Wizard: 3-Position Synthesis (I)

Based on three specified positions/angles of the coupler point C and the location of the base pivots Ao and Bo, a 4-bar mechanism is synthesized, which generates a motion of the coupler point that passes exactly through these three prescribed positions with the prescribed orientations of the coupler plane.

Some combinations of requirements are such that the resulting mechanism can not reach all positions without being dis-assembled inbetween positions. In such a situation, the user can attempt to specify different locations of the base pivots Ao and Bo. Furthermore, in many design situations only the starting and end position are really fixed and it is tolerable to vary the middle position.

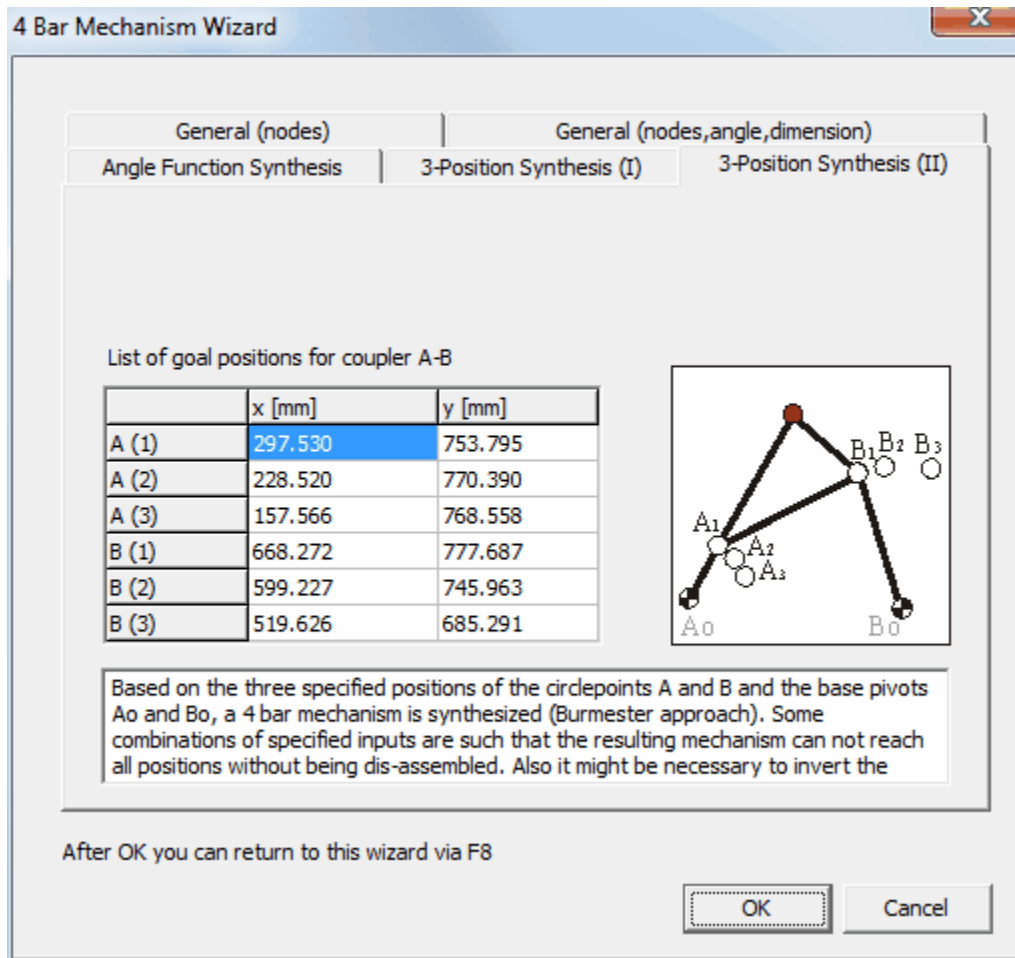
In case the coupler point passes through the prescribed positions in the wrong order, one has to change the direction of the input motion.

After pressing OK, the mechanism is automatically synthesized and displayed including the three prescribed positions/angles of the coupler point.



Synthesized 4-bar mechanism including the three prescribed positions/angles of the coupler point

5.1.5 3-Position Synthesis (II)



4-Bar Mechanism Wizard: 3-Position Synthesis (II)

Based on three specified positions of the circle points A and B and location of the base pivots Ao and Bo, a 4-bar mechanism is synthesized, which generates a motion that satisfies these requirements.

Some combinations of requirements are such that the resulting mechanism can not reach all positions without being dis-assembled in between positions. In such a situation, the user can attempt to specify different locations of the base pivots Ao and Bo. Furthermore, in many design situations only the starting and end position are really fixed and it is tolerable to vary the middle position.

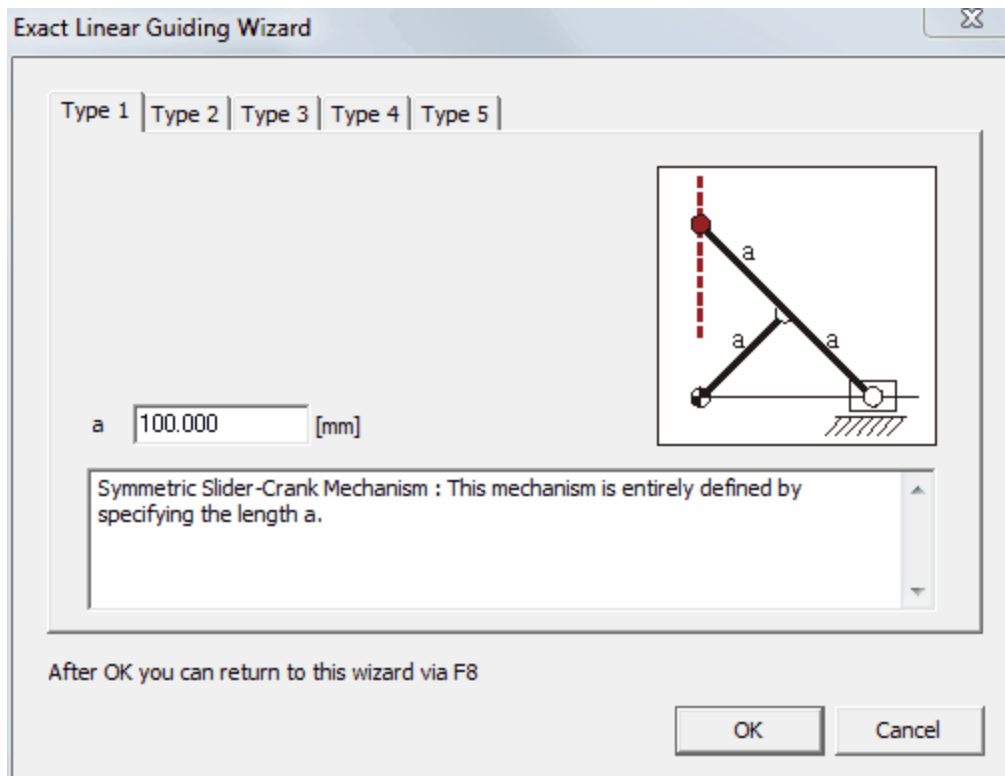
In case the coupler point passes through the prescribed positions in the wrong order, one has to change the direction of the input motion.

5.2 Exact Straight Line Mechanism

The menu File>Wizard>Exact Linear Guiding invokes the dedicated dialog, which allows

you to generate a number of parametrically predefined straight line mechanisms in a very easy way.

5.2.1 Symmetric Slider-Crank Mechanism



Exact Straight Line Motion Wizard: Symmetric Slider-Crank Mechanism

5.2.2 Geared Double-Crank Mechanism

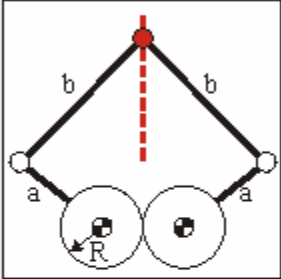
Exact Linear Guiding Wizard

Type 1 | Type 2 | Type 3 | Type 4 | Type 5

a [mm]

b [mm]

R [mm]



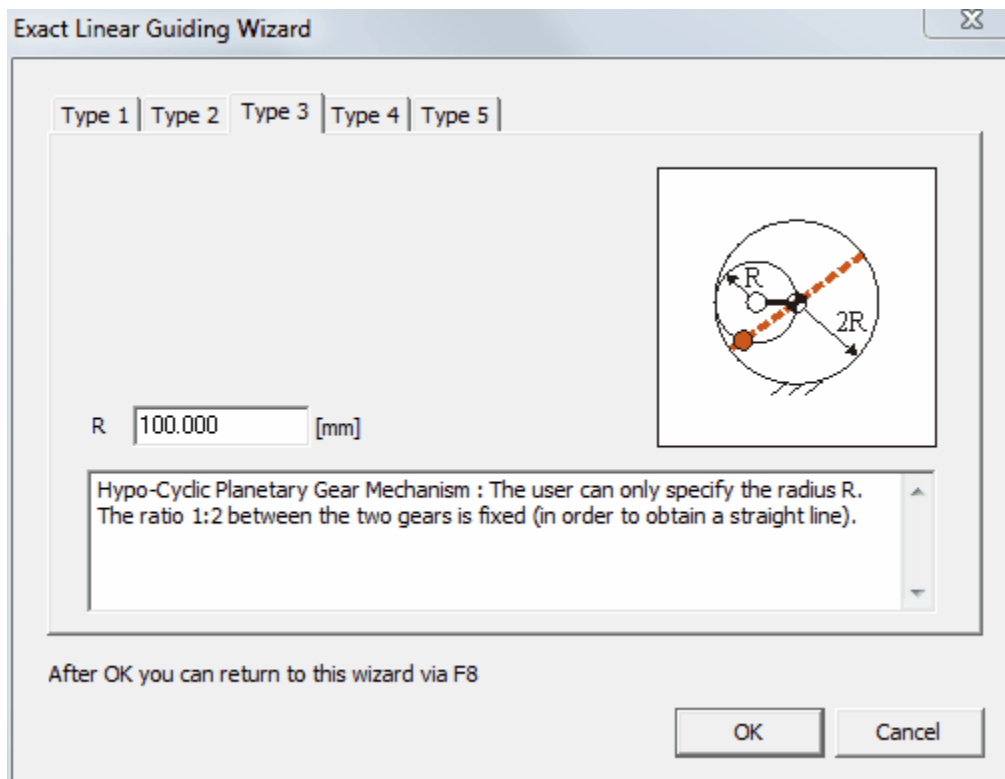
Geared Double Crank Mechanism : You can specify the values a, b and R.

After OK you can return to this wizard via F8

OK Cancel

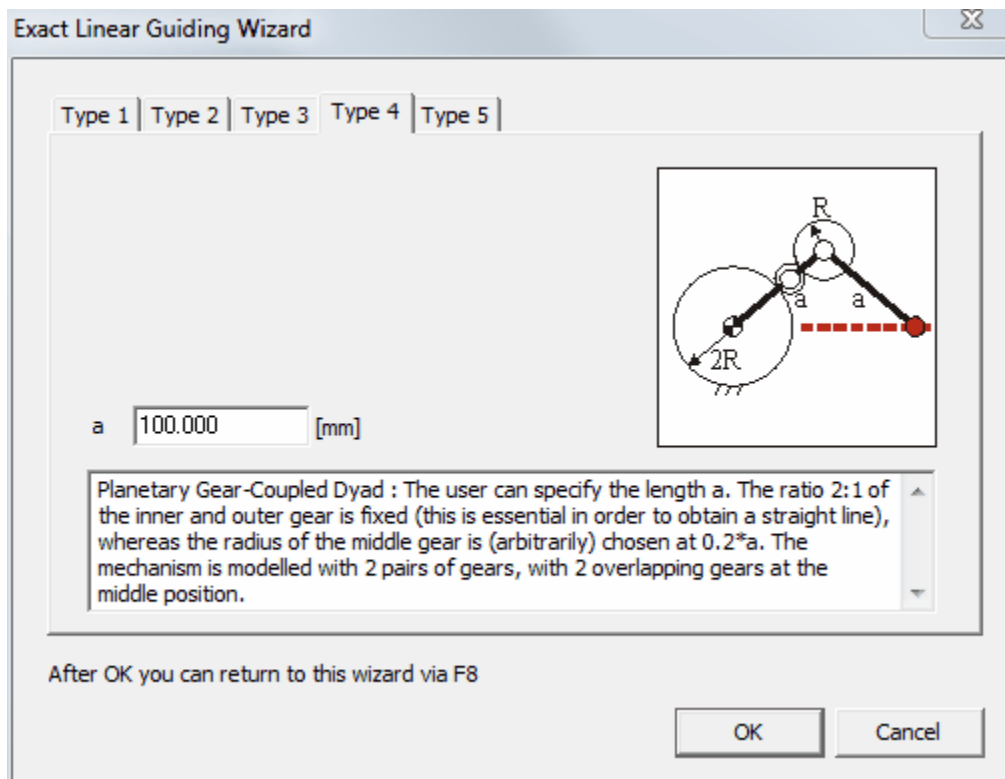
Exact Straight Line Motion Wizard: Geared Double-Crank Mechanism

5.2.3 Hypo-Cyclic Planetary Gear Mechanism

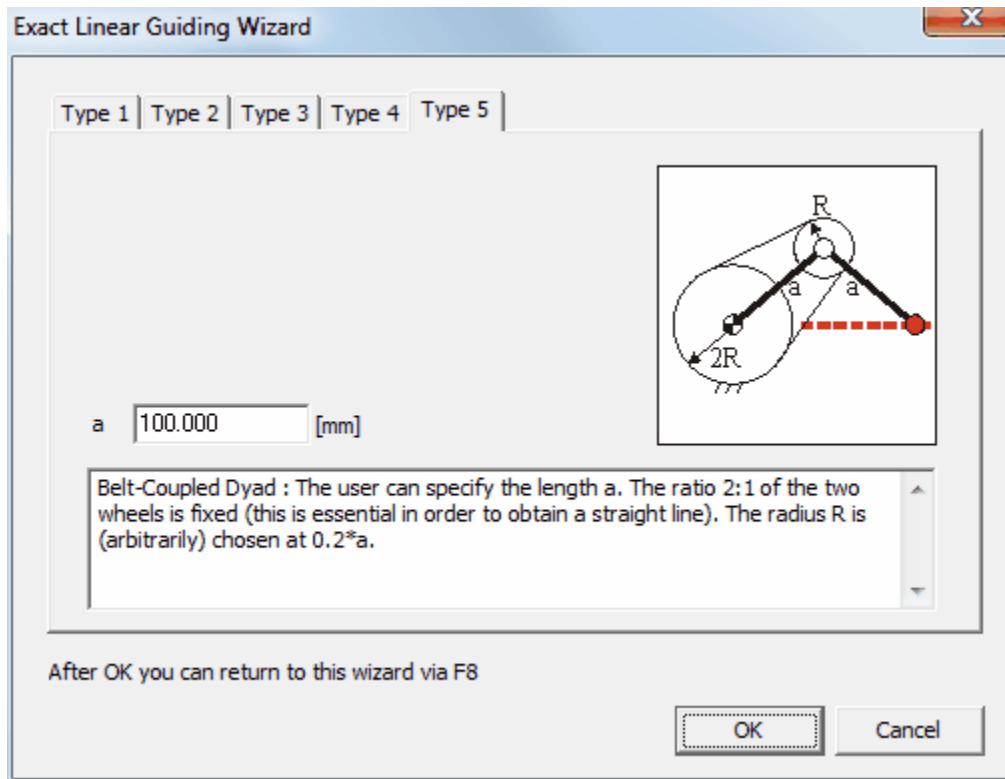


Exact Straight Line Motion Wizard: Hypo-Cyclic Planetary Gear Mechanism

5.2.4 Planetary Gear-Coupled Dyad



5.2.5 Belt-Coupled Dyad



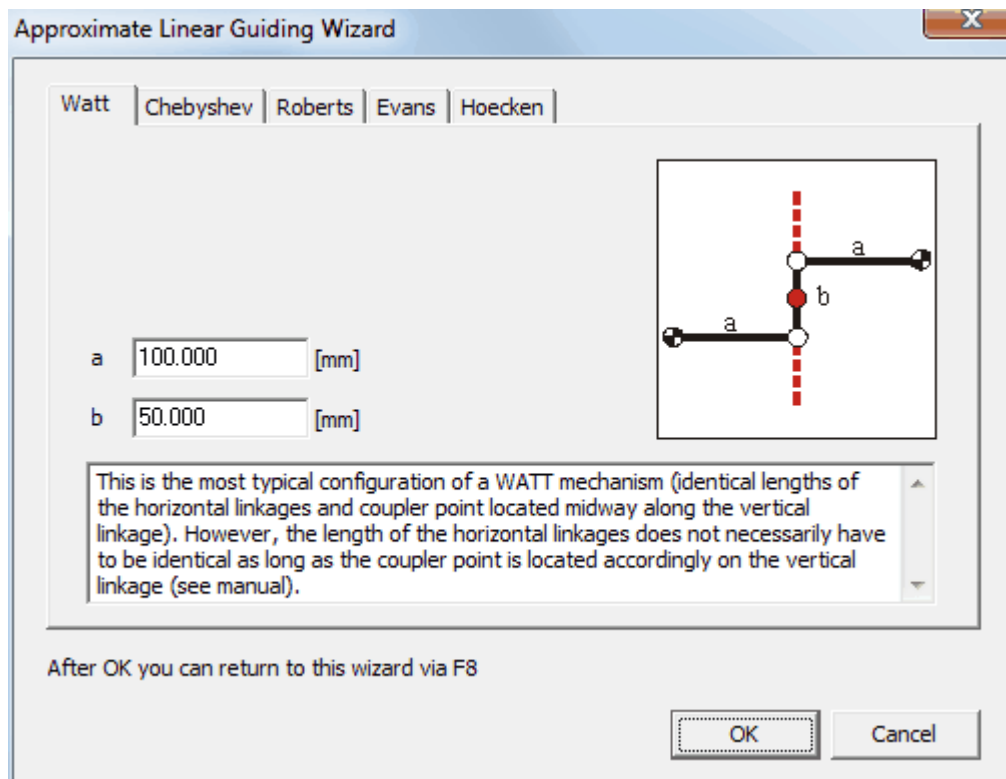
Exact Straight Line Motion Wizard: Belt-Coupled Dyad

5.3 Approximated Straight Line Mechanism

The menu File>Wizard>Approximated Linear Guiding invokes the dedicated dialog, which allows you to generate a number of parametrically predefined approximated straight line 4-bar mechanisms in a very easy way. The five mechanisms are named after their inventors:

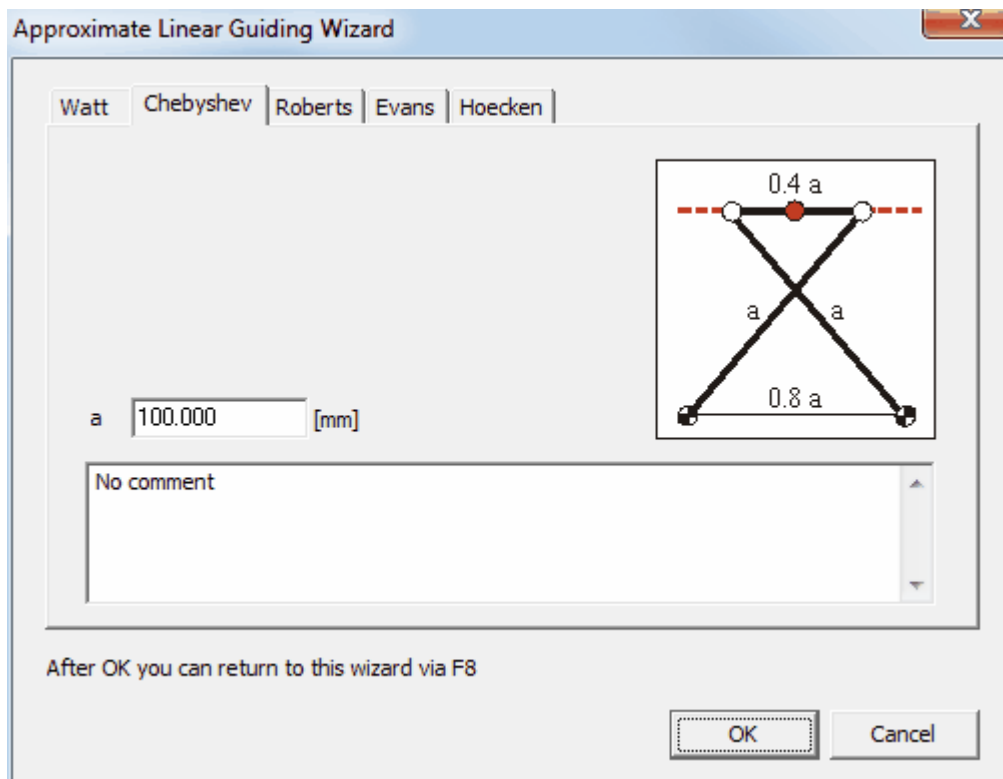
- Watt
- Chebyshev
- Roberts
- Evans
- Hoecken

5.3.1 Watt



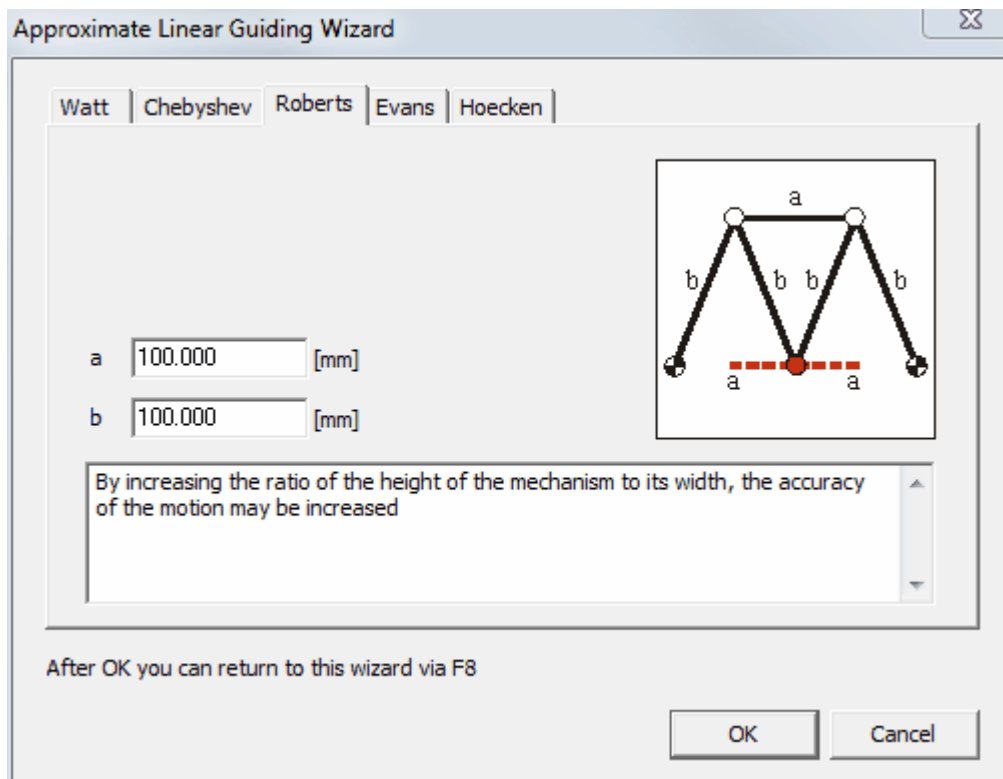
Approximated Straight Line Motion Wizard: Watt

5.3.2 Chebyshev



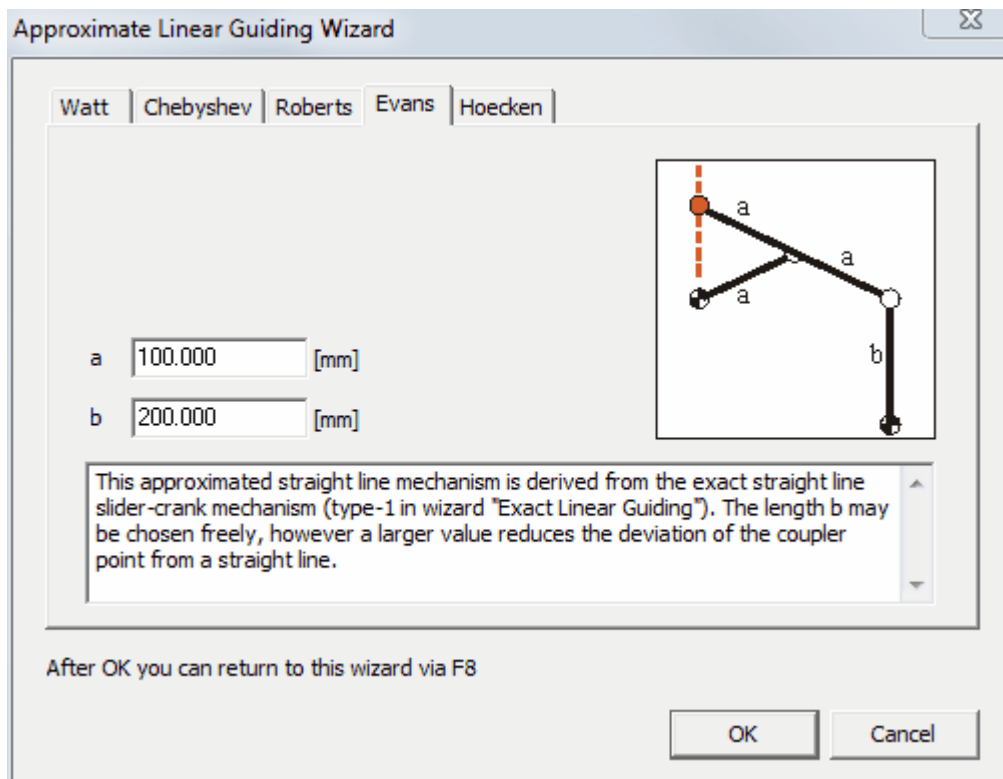
Approximated Straight Line Motion Wizard: Chebyshev

5.3.3 Roberts



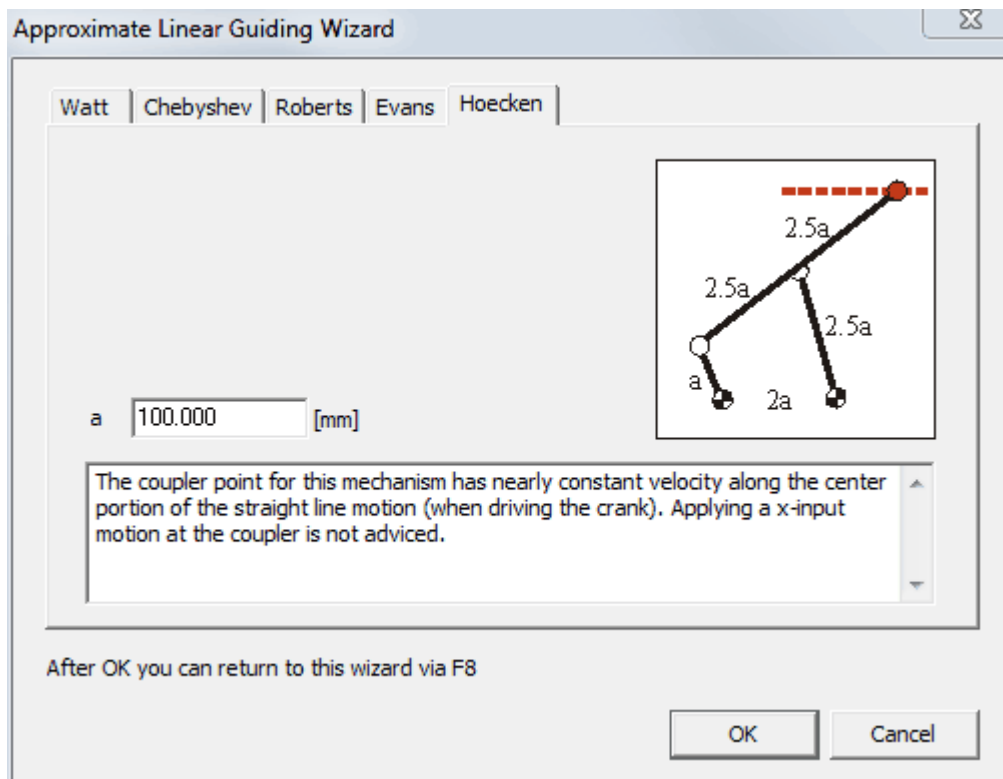
Approximated Straight Line Motion Wizard: Roberts

5.3.4 Evans



Approximated Straight Line Motion Wizard: Evans

5.3.5 Hoecken



Approximated Straight Line Motion Wizard: Hoecken

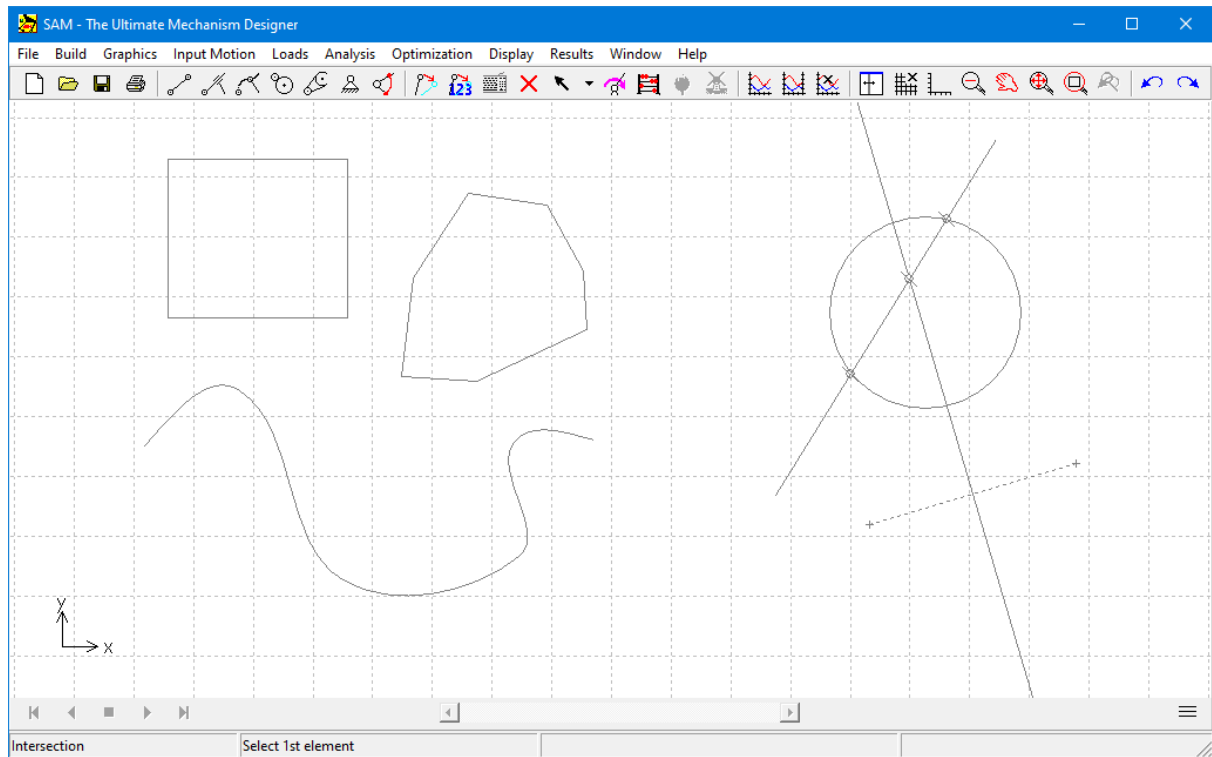
Chapter



6

6 Graphics

SAM lets you draw/manipulate/group various graphics shapes. These can also be imported via File>Import DXF. Individual shapes or a group of shapes can also be attached to any beam or slider element, such that it moves together with the element.



Available shapes (point, line, circle, rectangle, polyline, Bezier) and construction aids (centerline, intersection)

Chapter



7

7 Modelling Considerations

7.1 Units

The use of correct engineering units in analysis programs is always a very interesting issue. To make life easy we have provided ultimate flexibility, meaning that at any time you can change the unit system in the menu File>Preferences>Units and SAM will automatically re-scale all available data and display the chosen units.


Note: When **importing** external data (motion file, force definition or optimization reference file) the user can indicate how to interpret the data in that file. It can be either in accordance with the SI(rad) unit system which implies that the data are converted to the currently active unit system or the data can be imported as-is, meaning the data is already defined in the currently active unit system. When **exporting** data, the currently active unit system is applied unless stated otherwise.

7.2 Number Representation

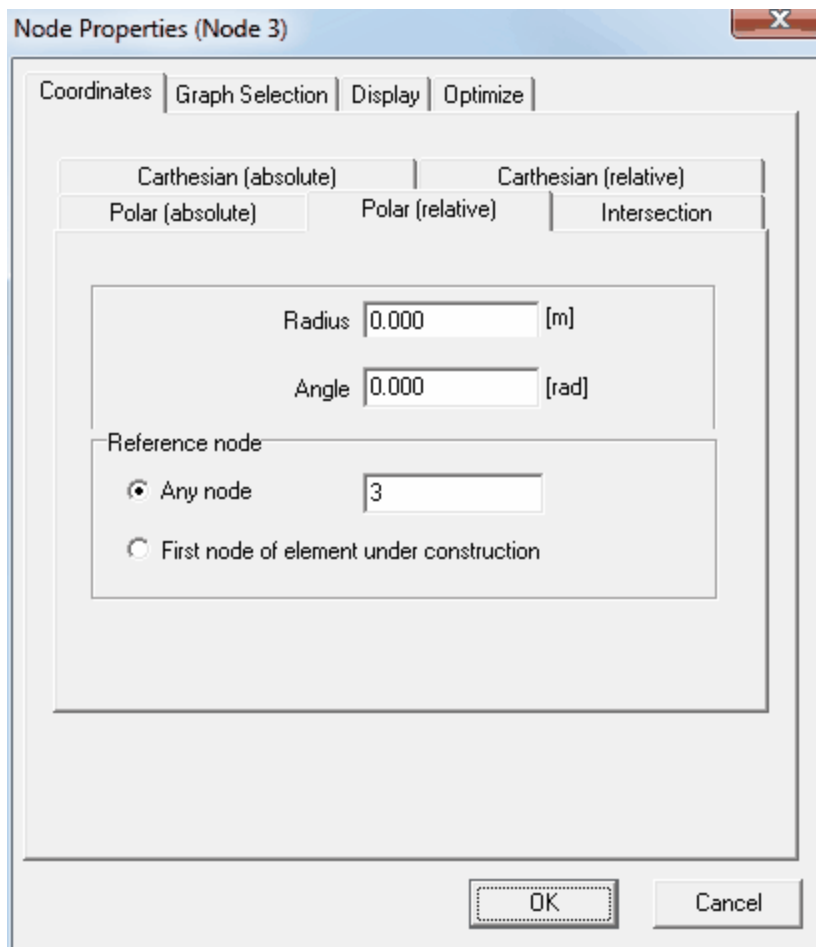
The user can set how decimal numbers should be displayed (File>Preferences>Numbers/Units). You can choose between fixed-point and floating-point representation and you can also set the number of decimals. When exporting results, a separate setting can be made in the Results>Export dialog that only affects the exported data.

Note: The number of digits that you select only affects the way the number is displayed. The internal precision equals that of the floating point processor, so any digits that you typed in are preserved (but perhaps not displayed if you have selected a smaller number of digits).

7.3 Exact Nodal Coordinates

In the process of building a mechanism you will often use the mouse to position a node. However, at a certain moment you will definitely want to enter exact nodal coordinates. Simply hitting the spacebar while you are dragging a node will show a dialog which offers various options for "constructing" exact nodal coordinates. You can also use the  button to modify the coordinates of a node.

In general, pressing the <SPACE> bar allows manual entry or selection at any stage.



Dialog for the "construction" of exact nodal coordinates

The (relative) Cartesian and polar coordinate option needs no further explanation. The last option "intersection" can locate the intersection of two circles with center nodes N1 and N2 and radii L1 and L2, respectively. You can select between the two intersections via the checkbox button.

CAD-data can be imported in DXF-format and the mouse can snap to all imported points. This feature also facilitates the creation of exact nodal coordinates.

7.4 Motion File

In case a combination of standard input motion laws is not sufficient to describe a certain input motion you can work with a motion file in standard ASCII format that allows you to specify a table of stepwise motion data.

The preferred unit system of a motion file is SI(rad) meaning m, m/s, m/s² and rad, rad/s, rad/s². However, as of SAM 8.2 one can also select to import motion data based on the currently active unit system.

Such a motion file can be generated in various ways : by hand, via some sort or spreadsheet program or by exporting analysis results generated by SAM.

7.4.1 Format

A motion file is a simple ASCII text file that defines the motion profile at specified moments in time in the following format:

0	0	vo	ao	If this row is absent, the default values are vo=ao=0 at t=0.
t1	u1	v1	a1	
:	:	:	:	
ti	ui	vi	ai	
:	:	:	:	
tn	un	vn	an	

Ui is the displacement/rotation at t=ti relative to the initial position of the mechanism (by definition, uo is always equal to zero); vi and ai are the corresponding velocity and acceleration.

IMPORTANT NOTE: The preferred engineering unit system used for motion file is the SI(rad) system. As of SAM 8.2 the user can define whether an **import** motion file is defined in SI(rad) and conversion is required or in the import motion file is already defined in the currently active unit system. In case motion data is **exported** to a motion file, an automatic conversion takes place and data is stored in the motion file according to SI(rad) engineering units.

Example: In case the following data is used for a translational motion and the user indicated that the data is in SI(rad), it defines a displacement of 0.5m in 2.0 seconds. The velocity has a constant value of 0.25 m/s, whereas the acceleration equals zero.

```
0.0 0.00 0.25 0
0.2 0.05 0.25 0
0.4 0.10 0.25 0
0.6 0.15 0.25 0
0.8 0.20 0.25 0
1.0 0.25 0.25 0
1.2 0.30 0.25 0
1.4 0.35 0.25 0
1.6 0.40 0.25 0
1.8 0.45 0.25 0
2.0 0.50 0.25 0
```

7.4.2 Importing a motion file

In the input motion dialog you can specify the motion file that you want to use.

Please note that the user can indicate how to interpret the data in the motion file. The first option is to interpret the data as being consistent with the SI unit system (m,m/s,m/s² and rad, rad/s, rad/s²) meaning that during the import process a conversion is made to the currently active unit system. The second options to import the data as-is without any conversion, which mean that the data must be defined in the currently active unit system.

7.4.3 Exporting a motion file

Analysis results can be exported as a motion file via Results/Export and the proper selection in the dialog box that appears. In this way motion data that is generated in one analysis can be used as input in another analysis.

When exporting analysis results in the motion file format, you must make sure that you currently have a valid selection of export data. A valid set consists of U_x , V_x , A_x (or U_y , V_y , A_y) in case of displacement data. The selection must exclusively consist of these three data items of one node. Other valid selections are A , V_a , A_a (for angle) and E , EV , EA (for elongation). Please make sure that no other items are selected next to the set that you need for the motion file.

The preferred unit system of the motion file is SI (m, m/s, m/s² and rad, rad/s, rad/s²), therefore a conversion from the currently active unit system to the SI unit system takes place during the exporting process.

7.5 Multiple Input Motions

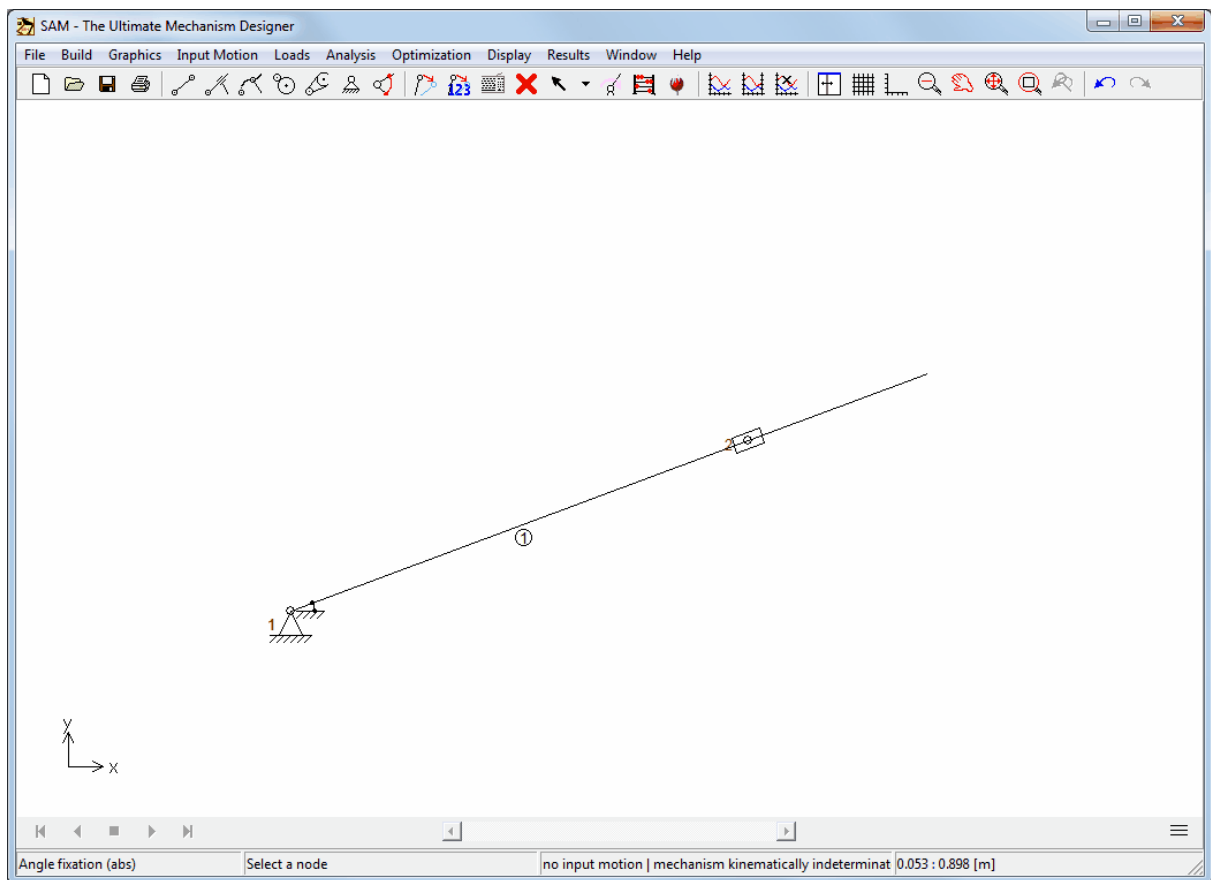
Each input motion generates a vector of discrete timestamps at which the movement/force of the mechanism must be evaluated. A global vector of discrete timestamps is generated by merging all the unique timestamps from each individual time vector. Timestamps that are close enough to satisfy the threshold requirement are treated as one.

7.6 Linear Actuator

A linear actuator such as a hydraulic cylinder is modeled as follows. First a beam element is modeled. Then, the elongation of this element is defined as input in the menu Input>Elongation.

7.7 Inclined Linear Guiding

In some cases one wants to analyze a mechanism with an inclined linear guiding, which cannot be modeled by "fixing" a node because this approach only works for horizontal or vertical linear guiding. However, an inclined linear guiding can be modeled on the basis of the slider element. By fixing both the node and the angle of the base point of the slider, the moving node can only slide along a fixed line.

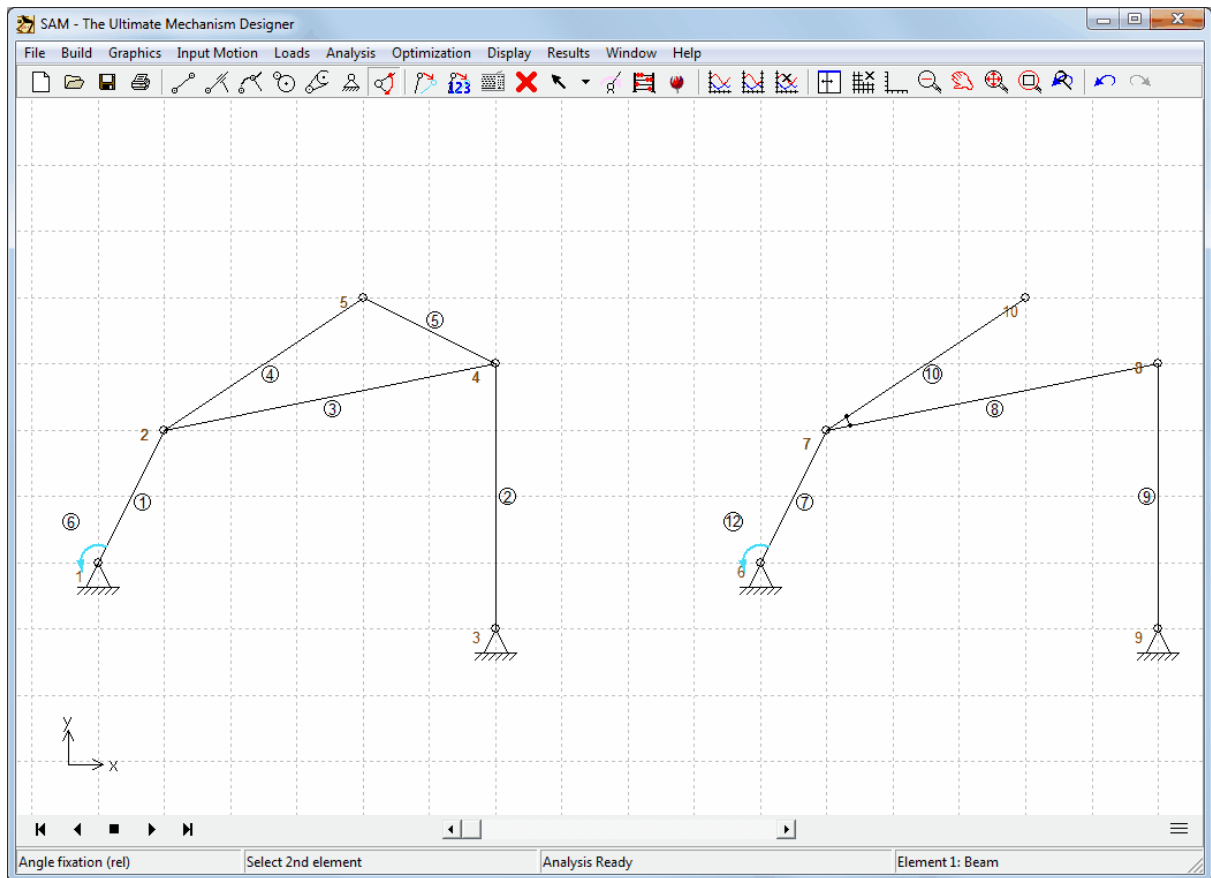


Modeling of an inclined linear guiding using a SLIDER element

7.8 Building Multi-Node Linkages

It is sometimes desirable or necessary to build a rigid linkage with more than two connecting points. This can be achieved by an assembly of several standard beam elements. In the case of a 4-bar mechanism with a coupler point, two approaches can be adopted:

1. Two beam elements with a rigid connection (coinciding nodes plus fixation of relative angle in menu Build>Fix Relative Angle)
2. Assembly of three beam elements, which are pin-joined. The area of the triangle may not become zero, therefore this approach cannot be used when the three nodes are collinear.



Two ways of modeling a coupler point

7.9 SAM/CAD design process

Depending on your design philosophy there are two entry points in an integrated SAM/CAD design process :

1. A conceptual design is made in SAM and the relevant data is exported in DXF-format to the CAD program for further processing. This will then lead to a mechanism design in the CAD environment. In the next step the mechanism design is transferred to SAM in DXF-format. In SAM the various items can be transformed (translated, rotated) and they can be attached to already existing elements.
2. A mechanism design is made directly in the CAD environment, which is then transferred to SAM in DXF-format. In SAM the conceptual mechanism can be constructed on the basis of the imported drawing. Next, groups of graphical items can be attached to elements.

Once DXF-data is imported it is part of the SAM project file.

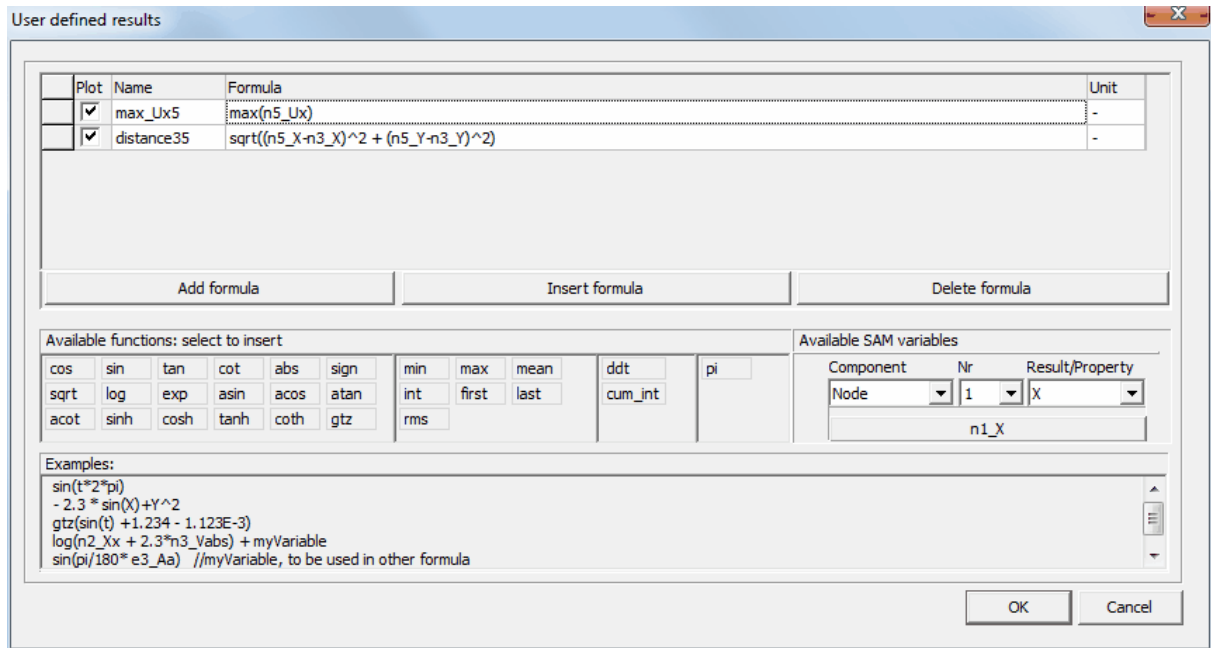
Initially the graphical entities of each imported DXF-file are treated as one group.

Only the following DXF-entities are supported : POINT, LINE, CIRCLE, ARC and TEXT.

Please refer to chapter "A Guide Tour / Running a sample project" for an example of the use of graphics.

7.10 User Defined Results

The sophisticated formula parser offers the possibility to combine simulation results in multiple ways to derive user defined results. One can define just one or a series of results. Each of the user defined results can be a single number (for example the maximum horizontal displacement) or a vector as function of time (distance between two moving nodes as function of time). A single number result is actually also stored as a vector as function of time with all identical values and can be plotted (yielding a straight line).



Plot: here you can switch plotting in the graph on/off

Name: choose any name according to the following rules:

- names are case insensitive
- allowable characters: a..z, A..Z, 1..9, _ (underscore),
- names should not start with a number
- spaces are not allowed (leading/trailing spaces will be deleted)
- existing SAM variables are not allowed (this also includes the variable "t" that is reserved for time)

Formula: can be edited by-hand or by selecting functions from the function list in combination with SAM variables from the variable selection tool

- the standard priority rules apply within each formula
- multiple formulas can be nested and the sequence of the formulas can be arbitrary

$$\text{var1} \mid \text{sqrt}(\text{var2})$$

$$\text{var2} \mid a^2 + b^2$$

Unit: a user-defined string that is used to identify the units of a variable when it is plotted.

- SAM does not perform any automated unit calculation based on the units of the variables and the formula.

Typical mixed math-physical expressions are:

- $1+(2-5)*3+8/(5+3)^2 \text{ sqrt}(2)$
- $(a+b)*(a-b) \ x^2+3*x+1$
- $256.33*\exp(-t/12) \ (1+(2-5)*3+8/(5+3)^2)/\text{sqrt}(5^2+3^2)$
- $2+3x+2x^2 \ 0.25x + 3.5y + 1$
- $(12.3)/(856.6) \ (-1)^{(2n+1)}*x^n/n!$
- $\text{sqrt}((n5_X-n3_X)^2 + (n5_Y-n3_Y)^2)$ // distance between node 5 and node 3
- $e2_A - e3_A$ // relative angle between element 2 and element 3

SAM variables are reserved names and can not be used to identify a user defined result. The list consists of:

- t (time)
- ni_xxx (property xxx of node i)
- ej_yyy (property yy of element j)
- examples: n1_Vabs (absolute velocity of node 1), e3_Ar (angle of element 3 relative to initial position of mechanism)

Constant numbers can be integers, decimal, or exponential

- examples: 2; -3234; 1.3333; -0.00025; 1.2345E-12

Functions are called by their function name followed by parentheses that hold the argument. Function arguments can be: numbers, variables, expressions, or even other functions (exception to this rule are the functions min, max, mean, int and ddt, that can not be used as arguments)

- trigonometric functions: cos, sin, tan, atan, ... (the trigonometric functions sin(x), cos(x), ... assume an argument x in RADIANS, whereas the reverse functions yield an output in RADIANS)
- sqrt, log, exp, sign
- gtz(x): if (x>0) then 1.0, else 0.0 (Greater Than Zero or Heaviside function)
- min, max, mean, int (integrates vector over time from 0 to end time), first, last, rms
- ddt (differentiation over time), intf (integration function over time, last value is identical to int)
- examples: sin(x); cos(2*pi*t+phi); atan(4*sin(x))

The standard order of operations used throughout mathematics, science, technology and many computer programming languages also applies to SAM's formula parser:

- exponents and roots
- multiplication and division
- addition and subtraction

This means that if a mathematical expression is preceded by one operator and followed by another, the operator higher on the list should be applied first. The commutative and associative laws of addition and multiplication allow terms to be added in any order and factors to be multiplied in any order, but mixed operations must obey the standard order of operations.

Nesting of multiple formulas is allowed and the sequence of the formulas can be arbitrary (the parser determines automatically the right processing order)

```
var1 | sqrt(var2)
var2 | a^2+b^2
```

Implicit multiplication is not supported because of its intrinsic ambiguity. So "xy" stands for a variable named "xy" and not for x*y. The multiplication symbol "*" cannot generally

be omitted.

Minus Sign

The parser recognizes the minus sign of constants, variables and functions.

- examples: x^{-n} ; 10^{-2} ; $x \cdot \sin(a)$; $-5 \cdot -2$

The parser applies the following rule: if the minus symbol is preceded by an operator, then it is treated as a sign; else it is treated as the subtraction operator. This feature simplifies expressions writing. Without this rule, the above expression should be written with parentheses, that is: $x^{(-n)}$; $10^{(-2)}$; $x \cdot (-\sin(a))$; $-5 \cdot (-2)$

Internal precision

Like many other standard Windows application SAM uses 15 digits internal precision.

Chapter



8

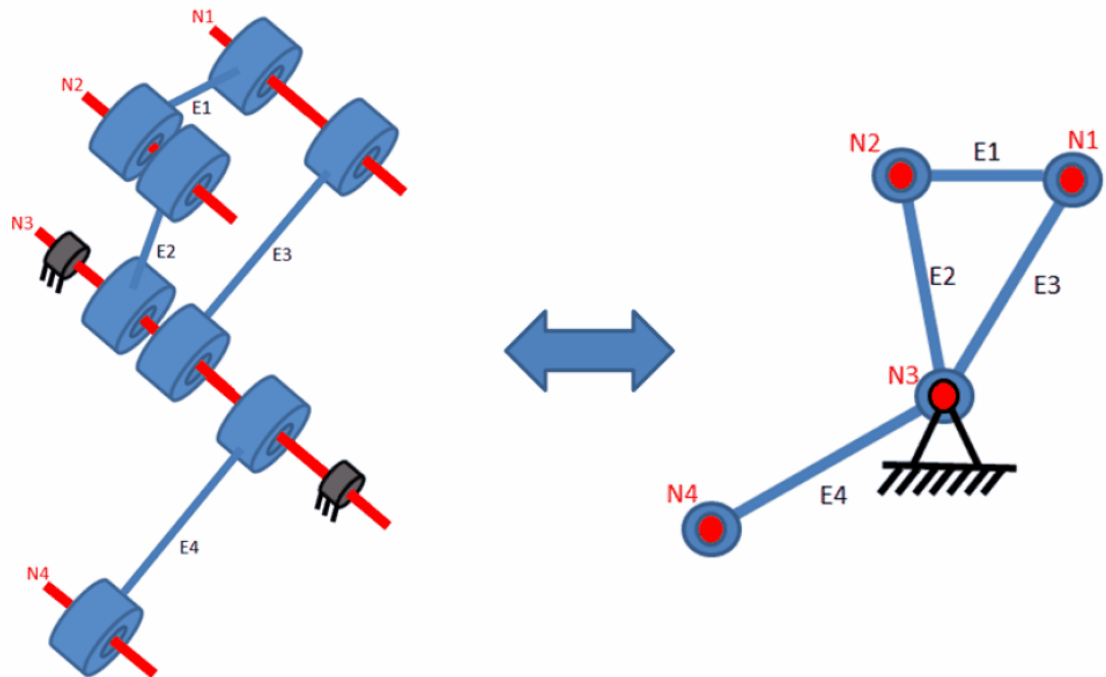
8 Kinetostatics

SAM can perform a kinetostatics analysis, which is a quasi-static force analysis based on the kinematic motion of the mechanism. In each position the force equilibrium due to inertia forces (acceleration, gravity), external forces/torques and spring, damper and friction forces is calculated. In the kinetostatics analysis, the forces result from the specified motion, whereas in a real dynamic analysis the motion that results from the specified forces are calculated via integration procedures. Results of such an analysis are:

- driving force/torque
- reaction forces support nodes
- bearing forces in elements
- internal forces in elements
- required or transmitted power

8.1 Reaction/Bearing Forces

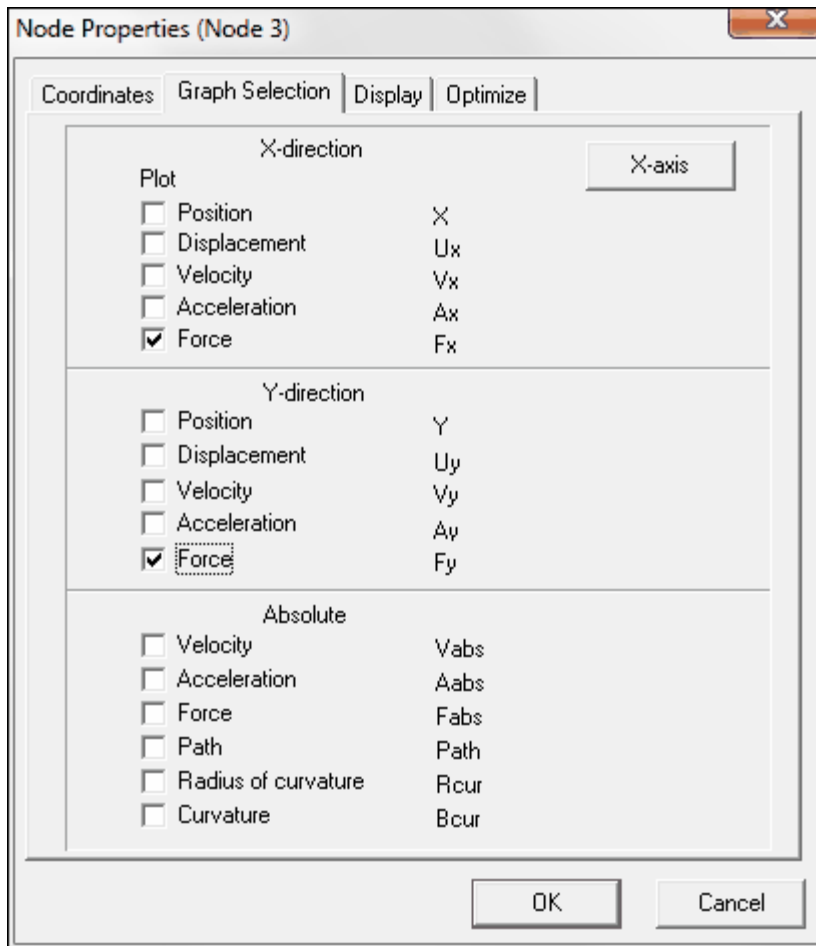
Next to the driving force or driving torque it is also essential to know reaction and bearing forces. To better understand the way reaction and bearing forces are treated in SAM it helps to realize that a mechanism is a collection of elements (blue) that share nodes (red) and to visualize these nodes as shafts that are the interface/connection between multiple elements (see illustration).



Schematic representation of (part of) a mechanism as a collection of elements that are connected via shafts (=nodes)

Reaction forces in fixed node

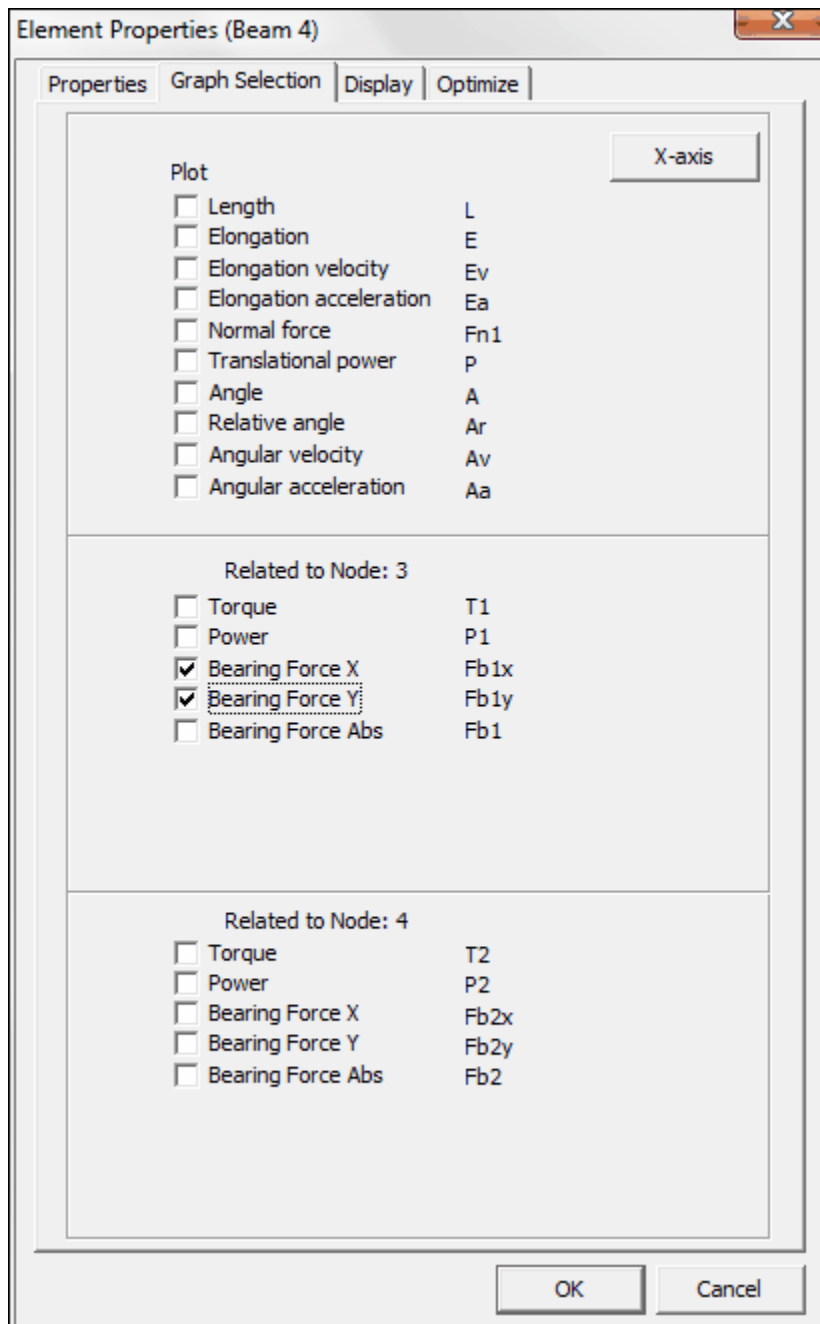
If one needs to know the combined reaction forces that are transferred via node/shaft N3 to the stationary part of the machine one would have to select F_x , F_y and/or F_{abs} in the node selection dialog.



Reaction forces in fixed nodes are selected via the Node Properties dialog (example: reaction forces in node 3)

Bearing forces of an element

If one needs to know the forces that are transferred via a bearing of an element to the connecting shaft (node) - and eventually from there to the rest of the mechanism or a support - one would have select F_{b1x} , F_{b1y} and/or F_{b1} in the element selection dialog.

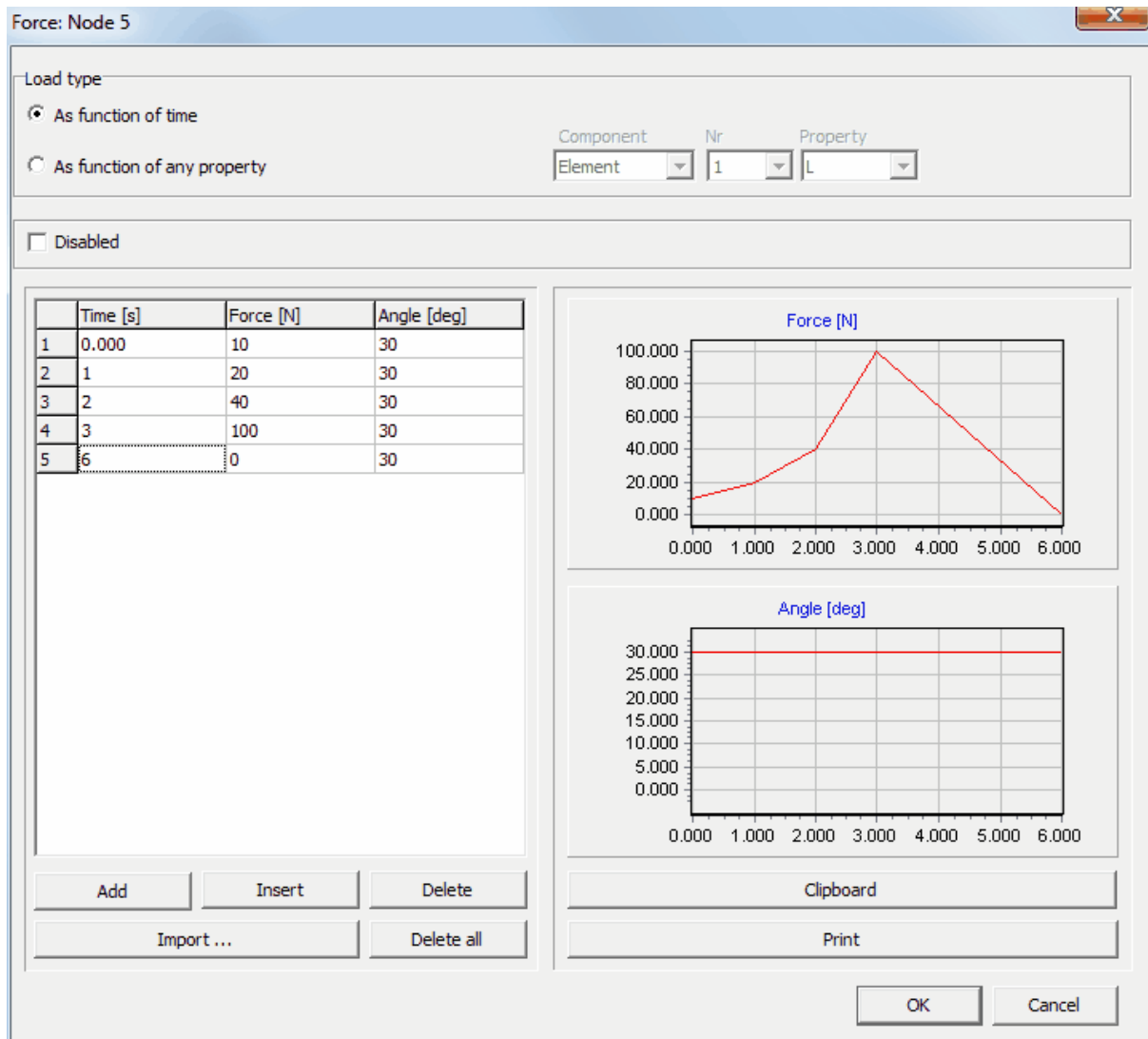


Bearing forces are selected via the Element Properties dialog (example: bearing forces of element 4 at node/shaft N3)

8.2 External Force/Torque

External forces, such as process forces, can be applied as function of time or as function of any motion result, that is calculated by SAM. The same holds for external torques.

The user can define a force (value & direction) as function of time.



Force profile (value and direction) as function of time

Forces/torques can also be modeled as a function of any kinematic property. This feature can be used to model a force that has a fixed value and an angle that corresponds with the angle of an element. This basically mimics a force in a local coordinate system that moves together with the motion of one of the linkages.

Force: Node 5

Load type

☐ As function of time

☒ As function of any property

Component: Element Nr: 1 Property: A

☐ Disabled

	A [deg]	Force [N]	Angle [deg]
1	0.000	150	30
2	360	150	390

Force [N]

Angle [deg]

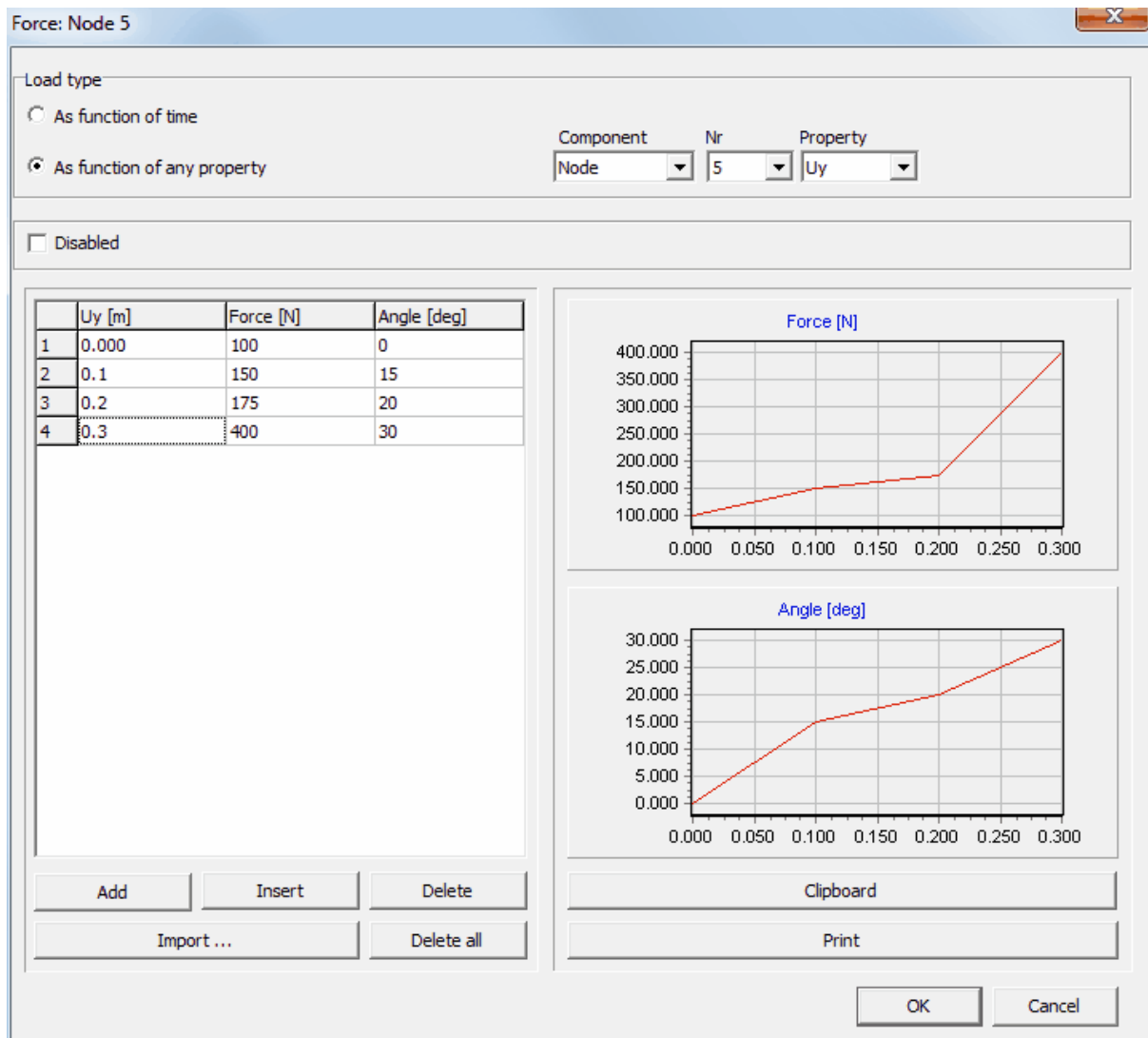
Clipboard

Print

OK Cancel

Force profile as function the angle A of element 1, such that the force is always under a 30 degrees angle relative to the element angle

The fact that the argument of a force can be any kinematic property means that one can also model a force on a node which depends on the y-displacement of another node.



Force profile as function the vertical displacement Uy of node 5

Chapter



9

9 Optimization

9.1 Introduction

The optimization module of SAM offers constrained single-function multi-parameter optimization based on a mix of evolutionary algorithms and Simplex techniques. Constraints are dealt with by treating each violation of a constraint as a penalty that is added to the original cost function. The software offers the option to define own results from the standard set of results via an advanced formula parser. This option is also used to define and add penalties.

Taking the initial design/topology as a starting point one can for example further improve the quality in which the trajectory of a coupler point equals the target trajectory by changing the geometry of the mechanism within pre-defined ranges. Or one can minimize the peak or RMS value of the driving torque of a mechanism by adding a compensating mass and let SAM determine the optimal value of the mass and its position within the allowable range. Just as in the case of the trajectory optimization one can also specify a reference function and minimize the difference between the actual and the reference function. When designing for example fitness equipment one is generally seeking a predefined force as function of displacement.

The goal for optimization can be the minimization or maximization of a variety of properties (peak, RMS, average, ...) or the difference between the actual and the target behaviour of a mechanism, such as :

- trajectory of a node (with or without prescribed timing)
- any motion or force quantity (as function of time or another quantity)

SAM seeks the optimum by modifying the following properties within user-defined ranges :

- geometry of mechanism
- element properties, such mass, spring constant, transmission ratio, ...

The optimization process in SAM is based on a two step approach, consisting of :

- exploration of the design space
- optimization of a specific solution

First, the entire parameter space is explored globally using a combination of a pure Monte-Carlo technique and a so-called Evolutionary Algorithm, which is a optimization technique derived from Genetic Optimization. The top list of such a global exploration are shown in the Explore list box, which displays the value of the optimization function and the corresponding parameters. The individual with the best property is listed at the top.

Next, the designer can select one of the results from the Explore window and start a local optimization. This local search can be either based on a Simplex technique or on a Evolutionary Algorithm with a smaller parameter range centered around the selected solution.

The combination of a global exploration strategy and a local optimization strategy - with the designer in the loop for selecting the mechanism that is further optimized - is believed to give the best trade-off between speed and coverage of the design space. Alternatively, options can also be set in such a way, that a fully automated optimization is performed.

9.2 Optimization Objective

The first step in the optimization of a mechanism is the definition of the optimization target. This is done in the following setup dialog that is invoked via the menu Optimize/Objective.

The 'Setup Objective' dialog box is shown with the following settings:

- Step 1:** Objective Type: Function
- Step 2:** Objective Details: Edit ...
- Step 3:** Objective Reference File (Optional): Edit ...
- Step 4:** Objective Performance Number: RMS
- Step 5:** Penalty Function (Optional):
- Step 6:** Optimization Target: Minimize

Buttons: OK, Cancel

Dialog to define the optimization target

The following choices and setup actions have to be performed :

Step 1: Objective Type

- Function: this can be any motion/force quantity
- Path: this can be the motion of any node
- Path incl. timing: this can be the motion incl. timing of any node

Step 2: Objective Details

Definition of the element/node property or the user defined variable that needs to be optimized. In case of function optimization one also has to define the argument of the function (default is time, but in certain applications one might select another quantity for example if one needs to optimize force as function of displacement).

Step 3: Objective Reference File (Optional)

Definition of a reference or target (**if required !**). In the case of a function optimization this is sometimes not required and can be left blank. For example if the maximum of the driving torque needs to be minimized one does not need to define a reference function. However if a certain force profile as function of displacement is required, that target profile obviously needs to be defined. The format of the reference file depends on the type of optimization:

Function:

argument_1 functionvalue_1

```
argument_2 functionvalue_2
```

```
: :
```

```
argument_n functionvalue_n
```

Path

```
x_1 y_1
```

```
x_2 y_2
```

```
: :
```

```
x_n y_n
```

Path incl. timing

```
time_1 x_1 y_1
```

```
time_2 x_2 y_2
```

```
: : :
```

```
time_n x_n y_n
```

Just before starting the import process the user can define whether the external data in the reference file are interpreted as being in SI(rad) units and require conversion into the currently active unit system or the data are already defined in the current unit system.

In order to achieve maximum flexibility, a reference path will not be automatically closed. To define a closed path the first points needs to be repeated at the end of the file.

Step 4: Objective Performance Number

Selection of property of the objective, expressed in one single number, that needs to be optimized. This can be a variety of properties (minimum, maximum, absolute maximum, range, average, RMS, standard deviation,) and it depends very much on the type of optimization problem which of these choices are relevant.

Step 5: Penalty Function (Optional)

If one needs to perform a constrained optimization, the violation of the constraint can be expressed as a penalty function (or actually a penalty number) that is added to the cost function. For that purpose one can select a user-defined result from the list of available user-defined results.

In case the optimization target has been set to: Maximize(cost function+penalty) it is essential that the penalty must be a negative value in order to make a solution that violates a constraint less favorable.

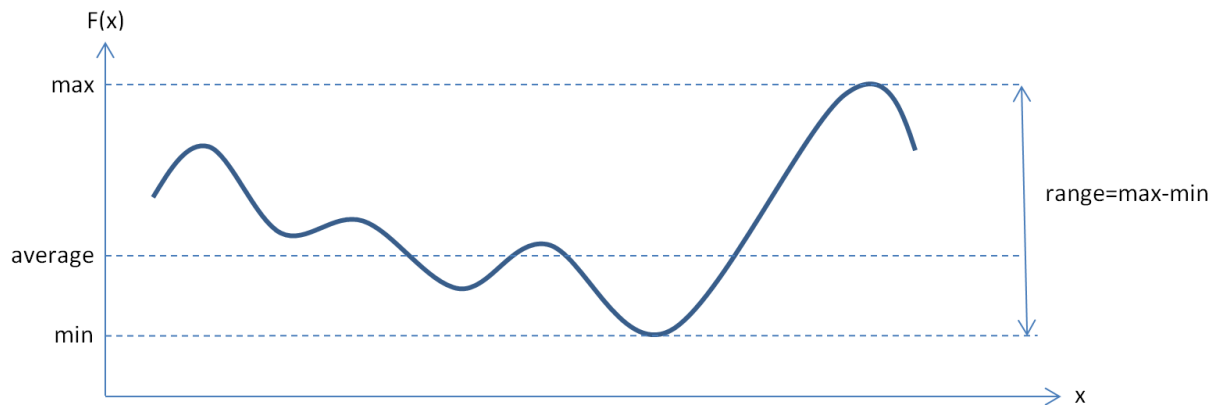
Step 6: Optimization Target

Finally, one has to choose whether the selected property needs to be minimized, which is the case in the majority of optimization problems, or maximized.

9.2.1 Objective Performance Number

The Objective Performance Number indicates the property of a curve, expressed in one single

number, that needs to be optimized. This can be a variety of properties (minimum, maximum, absolute maximum, range, average, RMS, standard deviation,) and it depends very much on the type of optimization problem which of these choices are relevant.



Min, Max is the extreme value of the data set.

Range is equal to the difference between Min and Max.

MaxAbs = $\max(\text{abs}(F(x)))$ is typically used when one wants to minimize the maximum deviation of an actual curve relative to a reference curve without caring about whether it is a negative or positive number.

Mean or Average of a data set is simply the average value.

Standard Deviation (Sigma) of a data set is the square root of its variance. Informally, it measures how far a set of numbers are spread out from their average value.

RMS = Root Mean Square of a data is defined as the arithmetic mean or average of the squares of a set of numbers. The RMS is also known as the quadratic mean. For a varying electric current, RMS is equal to the value of the constant current that would produce the same average power dissipation in a resistive load.

9.2.2 Reference or Target Definition

The target function or path (with or without timing) is defined in a simple ASCII file based on the following format definition which depends on the type of optimization:

Function

```
argument_1    functionvalue_1
argument_2    functionvalue_2
:             :
argument_n    functionvalue_n
```

Path without timing

```
x_1    y_1
x_2    y_2
:       :
x_n    y_n
```

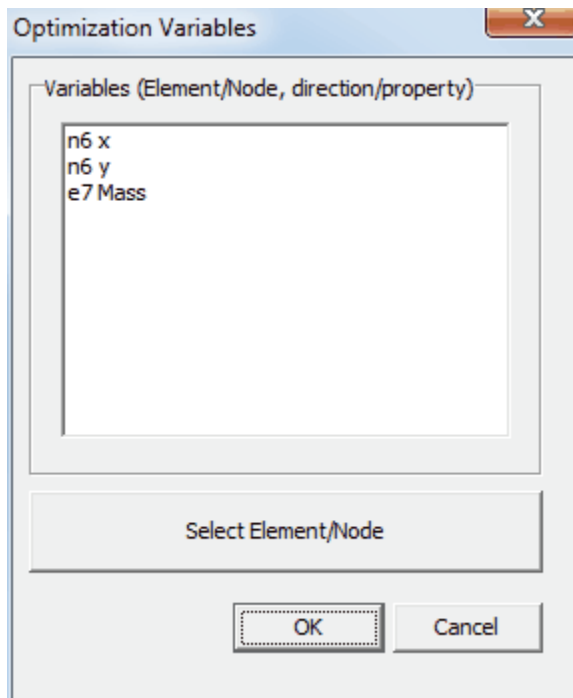
Path including timing

time_1	x_1	y_1
time_2	x_2	y_2
:	:	:
time_n	x_n	y_n

1. Depending on the choices prior to the import of data, the target or reference data from an external file will be interpreted in SI(rad) units followed by a conversion to the currently active unit system or will be imported as-is assuming that the data is already provided in the currently active unit system.
2. The target path runs from the first point to the last point and is NOT automatically closed to give the user maximum control over the target definition. In order to close the target path, the first point needs to be repeated at the end of the reference file.

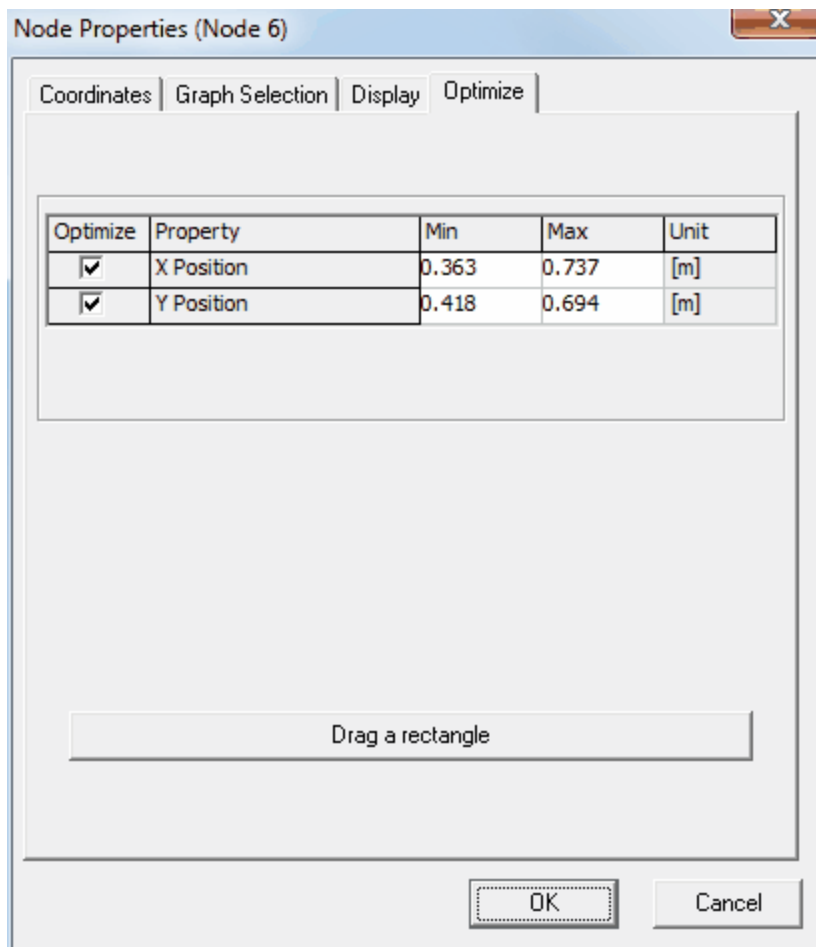
9.3 Parameter Space

The second step in the optimization of a mechanism is the definition of the parameters and the range within which the parameters can be varied in order to optimize the objective function. The menu Optimization/Parameters invokes the following dialogs, which shows the currently selected parameters plus a button to select additional parameters.



Dialog showing currently selected parameters plus button to select additional parameters

Once the Select button is pressed one can for example select a node and define which of the two coordinates of the node can be varied during optimization and within which boundaries. Next to the manual entry of boundary values one can also press the button called "Drag a rectangle" and define a rectangular region with the mouse.



Dialog optimization settings of a node

9.4 Constraints

Constraints are inequalities that limit the feasible space of design parameters (optimization variables) directly or indirectly. Typical kinds of constraints are:

- Direct limits (upper and/or lower bound) of a design parameter. For instance, the length of a binary link may not be zero or negative, or a designer specifies explicitly limits for the length of a link. This can be checked without even running a simulation.
- A boundary for a motion quantity. For instance, the point P of a coupler plane must stay in a certain region of the xy-plane.
- The overall feasibility of the kinematic parameter space. For instance, it is not possible to give one link any arbitrary length without assembly problems.

In many cases, constraints can be converted towards an unconstrained optimization problem with the help of penalty functions.

The idea is to add a penalty value to the cost function in order to make it less favorable when the constraint is violated. A distinction can be made with respect to feasibility of the violation. When a small violation can be accepted, the exterior type can be applied. The penalty will be added as soon as

the violation has begun. In case that the parameter space outside the boundary is not feasible, the interior type must be applied, the closer the distance to the boundary, the more penalty.

In case the optimization target has been set to Maximize (cost function+penalty) , it is essential that the penalty must be a negative value in order to make a solution that violates the constraint less favorable.

9.5 Optimization Options

The menu Optimization/Options invokes the following dialog:

Dialog of standard optimization options

The user can choose between a user-controlled modus and an automatic modus. In the user controlled mode a global exploration of the parameter space is performed based on evolutionary algorithms leading to a set of solutions. The user can then select any of these solutions and further refine it via a local optimization, which can be either based on an evolutionary approach or on a simplex method. In the automatic mode, the best solution of the global search is automatically refined in a local optimization (again either based on an evolutionary or a simplex approach).

The approach that is used for the local optimization is defined under the heading "Local Optimization".

In the next section the user can adapt the settings of the Evolutionary and of the Simplex algorithm. For ease of use, only the most frequently adapted settings are shown in first instance. The entire range of settings, some of which require more theoretical background, are displayed after clicking on **more**.

Optimization Options

Modus

☒ Automatic (global exploration & local optimization) ☒ Optimize all solutions from global exploration

☐ User-Controlled

Local Optimization

☐ Evolutionary Algorithm

☒ Simplex Method

Evolutionary Settings

Nexperiments: 300

Population Size: 50

Nrandom: 50

Selection Pressure: 1.5

Ext.Re-combination: 0.25

Mutation Rate: 0.05

Reduced area (0..1): 0.1

Simplex Settings

Max Iterations: 100

Search within boundaries: ☒

Tolerance (default: 1e-8): 1E-8

Initial range factor (0 - 0.5): 0.1

Input Motion

Reject individual experiments if the successful number of motion steps, expressed in percentage, is less than: 100 %

OK Cancel

Dialog showing the full range of optimization options

Finally, there is also a setting that defines how to deal with experiments that could not complete the full input motion. In most cases it is useful to reject any optimization experiment during which the full motion could not be completed. This failure is typically caused by convergence issues because an certain input motion step can not be executed by the mechanism. However, in some optimization challenges, for example in case one wants to maximize the output motion range of a mechanism without knowing upfront what the corresponding input motion needs to be, it can be useful to accept experiments that did not complete all input motion steps. To facilitate this an extra optimization option has been introduced. It defines the rejection of an experiment based on the percentage of completed motion steps. The default value is 100%, meaning that the entire motion cycle must be completed, but the value can also be set to 75%, 50%, 25% or 10%.

The non-standard settings of the Evolutionary and of the Simplex algorithms can be adapted, but in general, the default values show good results and only expert users should modify these after consultation of specialist literature.

Each of the settings is explained briefly in the next chapter.

9.5.1 Settings Evolutionary Algorithm

Nexperiments: Total number of valid individuals that are generated in the process of the evolutionary search in the design parameter space.

Nrandom: In the chosen implementation an evolutionary search process starts with a random

generation of individuals based on this number.

Population Size: Size of the population. Starting with an empty population it is filled first with Nrandom individuals and then with evolutionary generated offsprings until the population size is reached. Then the evolutionary process of generating offsprings continuous while replacing individuals in the population by better offsprings to maintain the size of the population.

Selection Pressure: The current implementation is based on a linear rank-based fitness assignment, which implies that the chance of a member of the population to be selected for the generation of an offspring is based on its rank in the sorted population rather than on its objective value. The choice of Selection Pressure adds an extra weighing function and can vary from 1.0 (all members of the population have equal chance of selection) to 2.0 (chance of selection decreases linearly with the rank position and is almost zero for the worst individual in the population).

Ext. Re-combination: In the chosen implementation of the evolutionary algorithm, the generation of offsprings is based on a random recombination of the parameter values of the parents. When choosing the standard intermediate recombination process only offsprings with parameters somewhere between those of the parents are generated and gradually a shrinkage occurs of the entire population. This effect can be overcome by extending the selection area beyond the parents. A value of 0.25 means that the area is extended by 25% at both sides and ensures (statistically) that the parameter area of the entire offsprings is the same as the parameter area spanned by the parameters of the parents.

Mutation Rate: After recombination every offspring undergoes mutation. Offspring parameters are mutated by small perturbations with low probability in order to avoid too fast convergence of the population to a local optimum. The probability of mutating a parameter is defined by the mutation rate. As a rule of thumb a mutation rate of $1/n$ (n : number of parameters) generally yields good results.

Reduced Area: When applying the evolutionary algorithm to further refine (local optimization) one specific solution of the global exploration one typically wants to limit the search area and center it around the solution one has chosen to further improve. All parameter ranges in the local search are multiplied by the given factor, which typically ranges from 0.01 to 0.1.

9.5.2 Settings Simplex Algorithm

Max. Iterations: The Simplex algorithm stops when the maximum number of iterations is reached or when the relative improvement drops below a certain value defined by Tolerance.

Tolerance: The Simplex algorithm stops when the relative improvement between two successive steps drops below this level or when the maximum number of iterations is reached.

Initial Range Factor: The first simplex is generated by successively adding a small perturbation to each of the parameters of the starting parameter set. The small perturbation is found by multiplying the search range of each specific parameter by the given factor.

Search Within Boundaries: Whereas the Evolutionary Algorithm searches only within the given parameter boundaries by nature, the Simplex Algorithm has to be restricted to the search area. Sometimes it is interesting to see which solution the algorithm converges to in case the boundaries are neglected.

Chapter



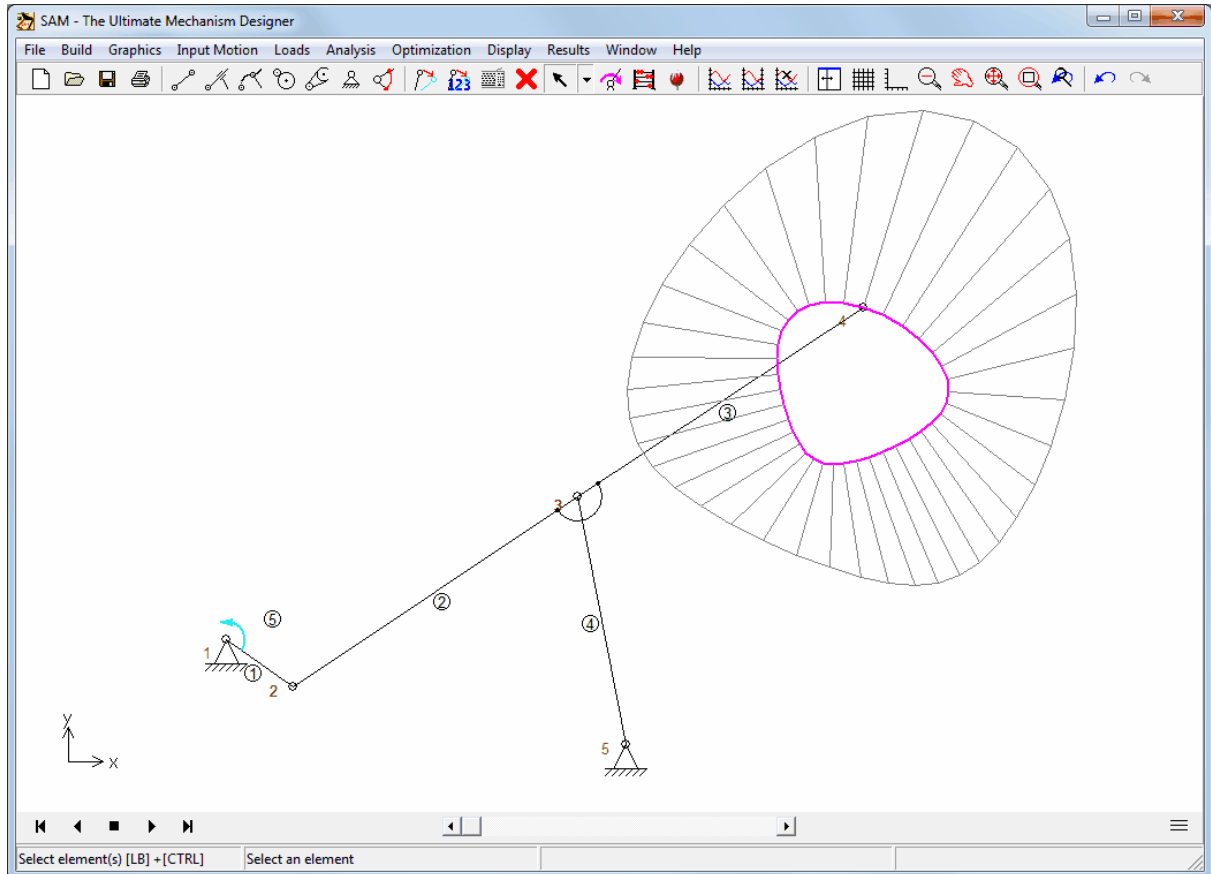
10

10 Examples

10.1 Motion Analysis

10.1.1 Crank Rocker Mechanism

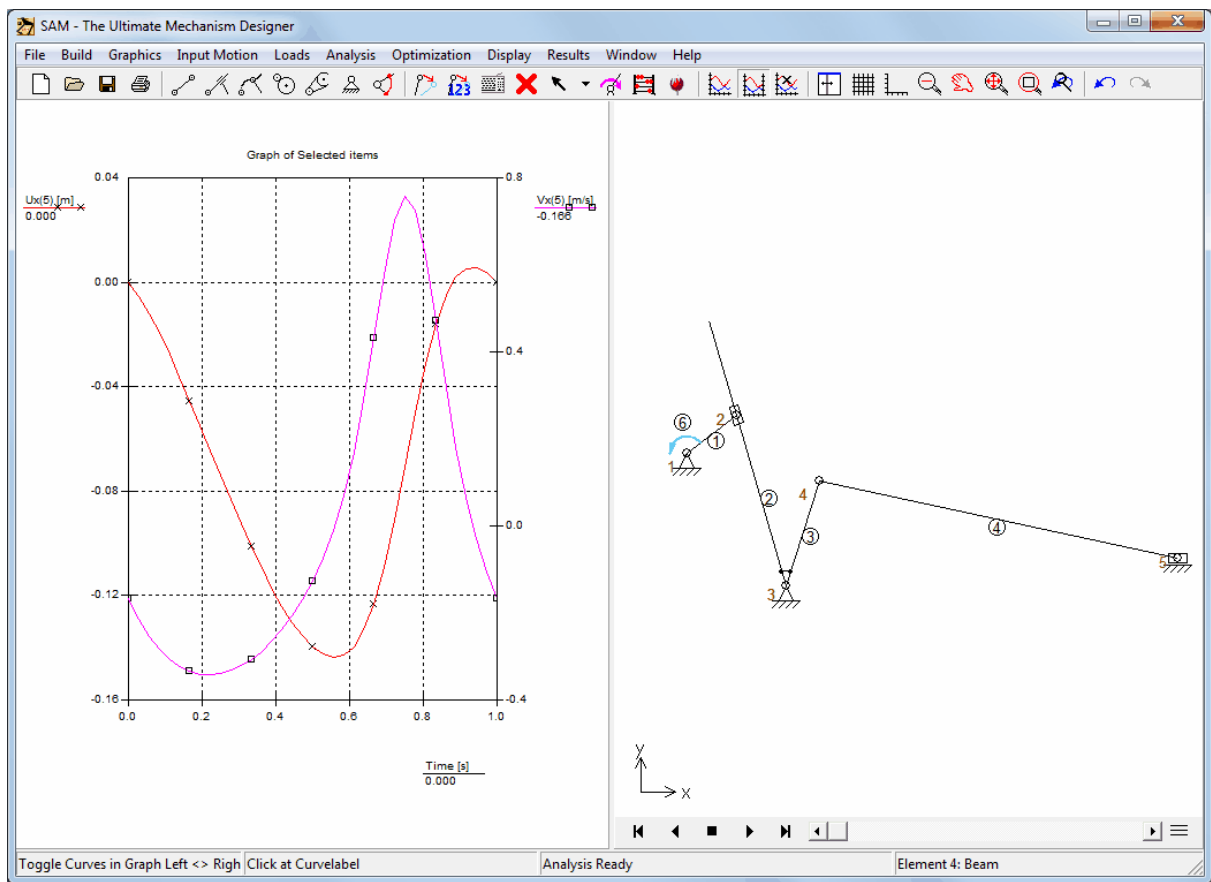
Example file CRRK.SAM defines a crank rocker mechanism, the crank of which is driven with constant speed. The path and velocity of node 4 is analyzed.



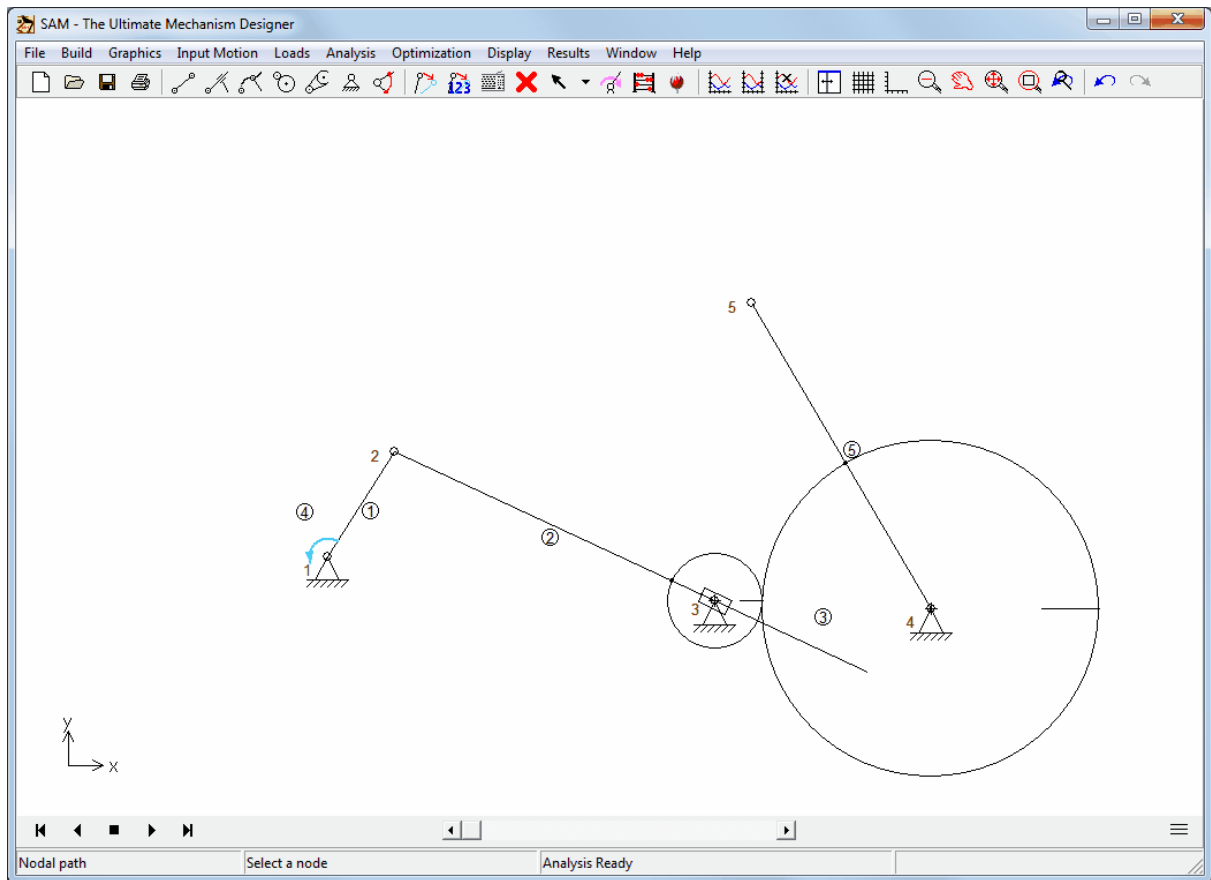
Crank Rocker Mechanism (CRRK.SAM)

10.1.2 Quick-Return Mechanism

In push-up units, a quick return mechanism is often used. Important in these mechanisms is the displacement and velocity diagram of the translating node.

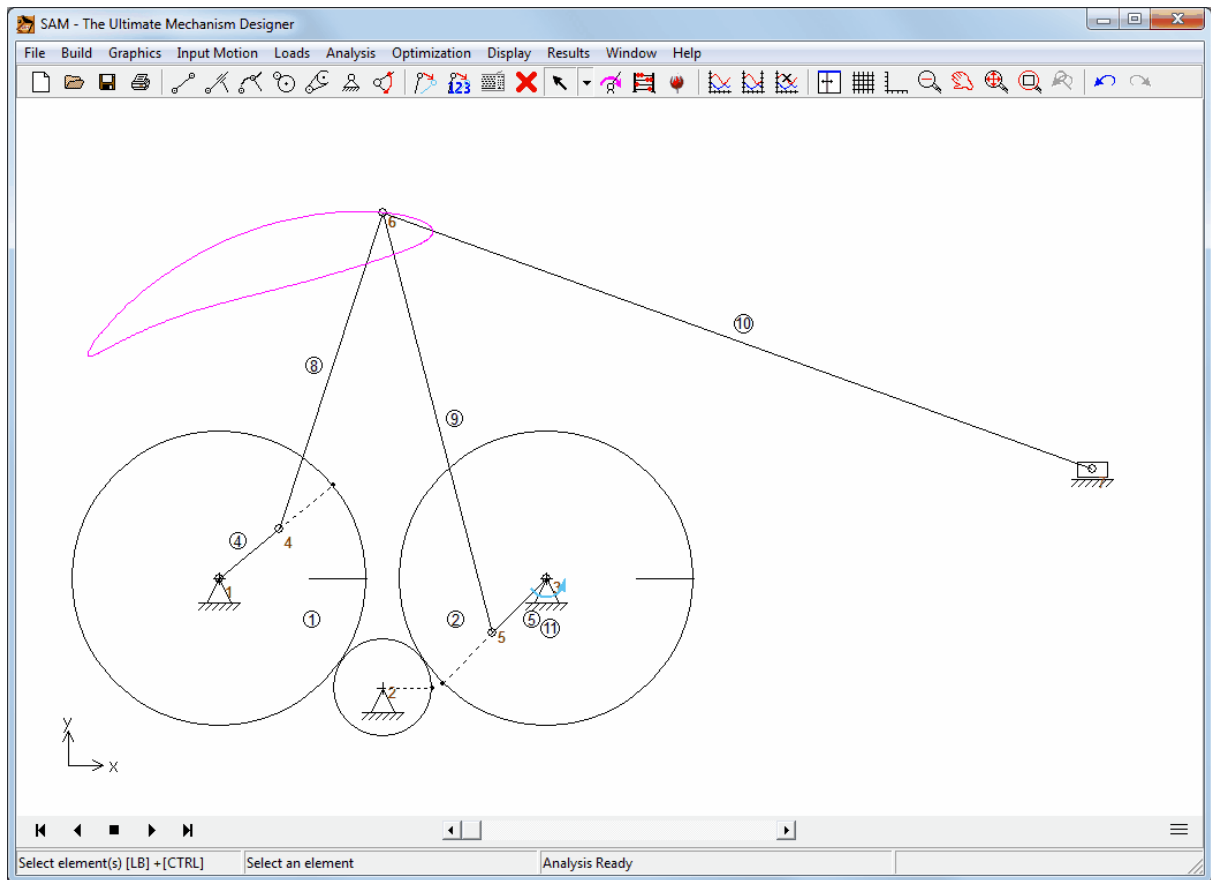
**Quick-Return Mechanism (QKRT.SAM)**

10.1.3 Slider Driven Gear Transmission



Slider Driven Gear Transmission (SLGR.SAM)

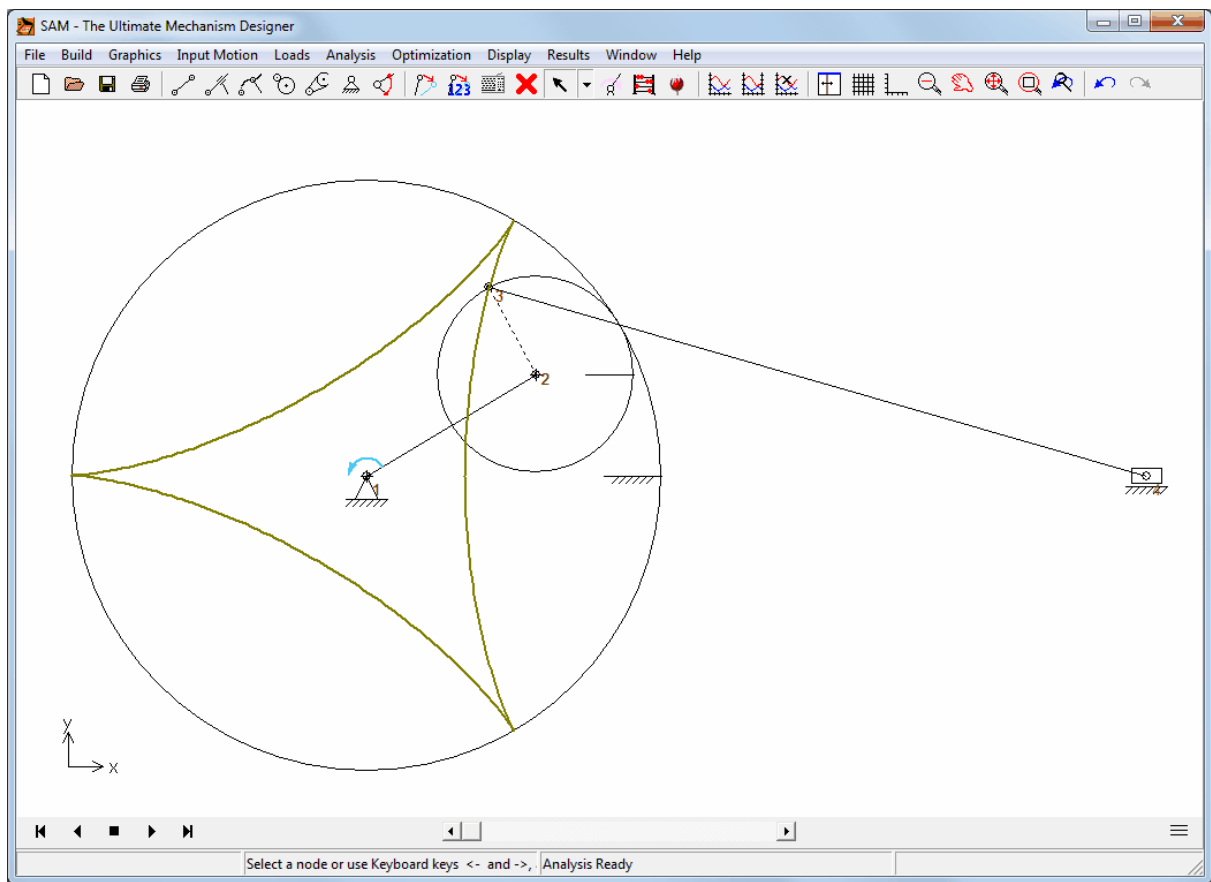
10.1.4 Double Crank Mechanism



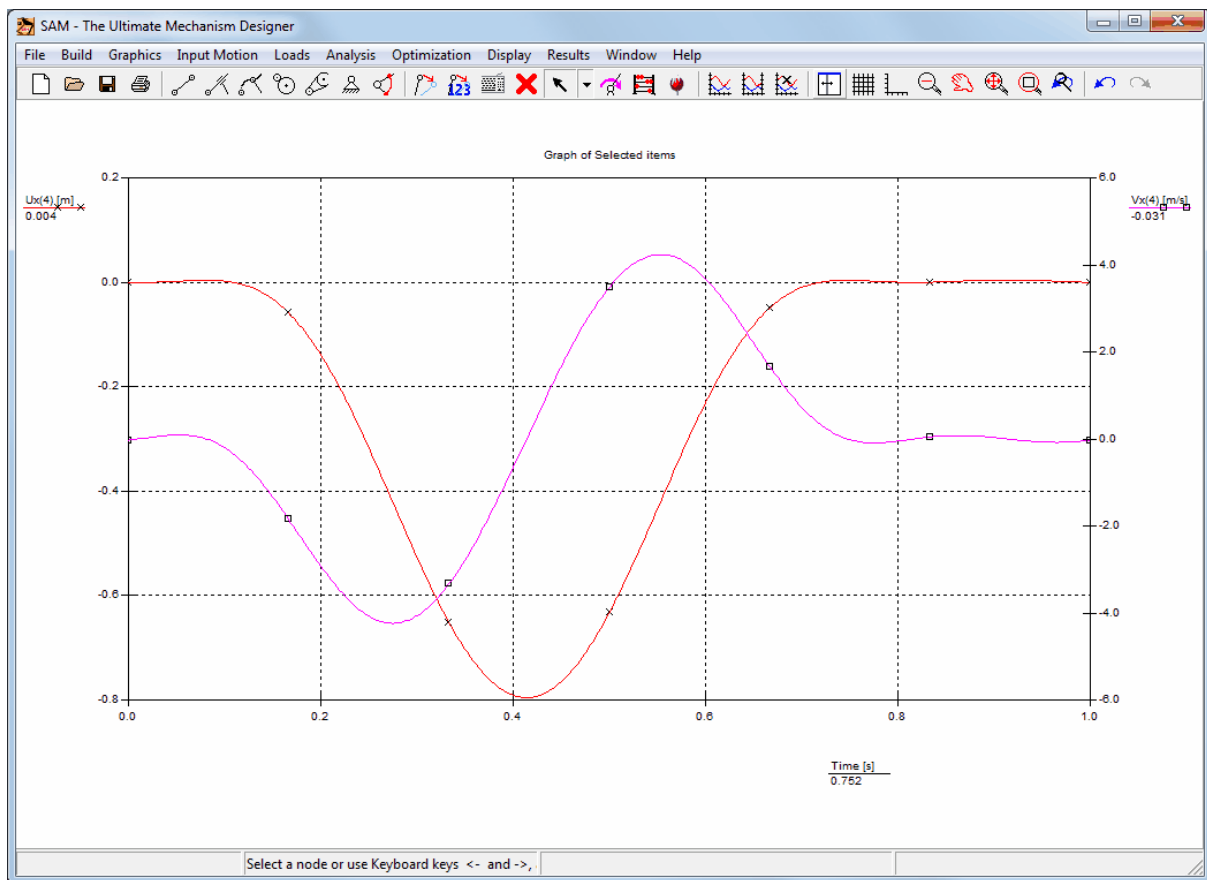
Double Crank Mechanism (DBCR.SAM)

10.1.5 Planetary Mechanism (Hypo-Cyclic)

A hypo cyclic planetary mechanism can be used to generate an approximated dwell. This is achieved by attaching an output linkage to the gear in such a way that the length of that linkage equals the radius of curvature of the connection (gear) node.



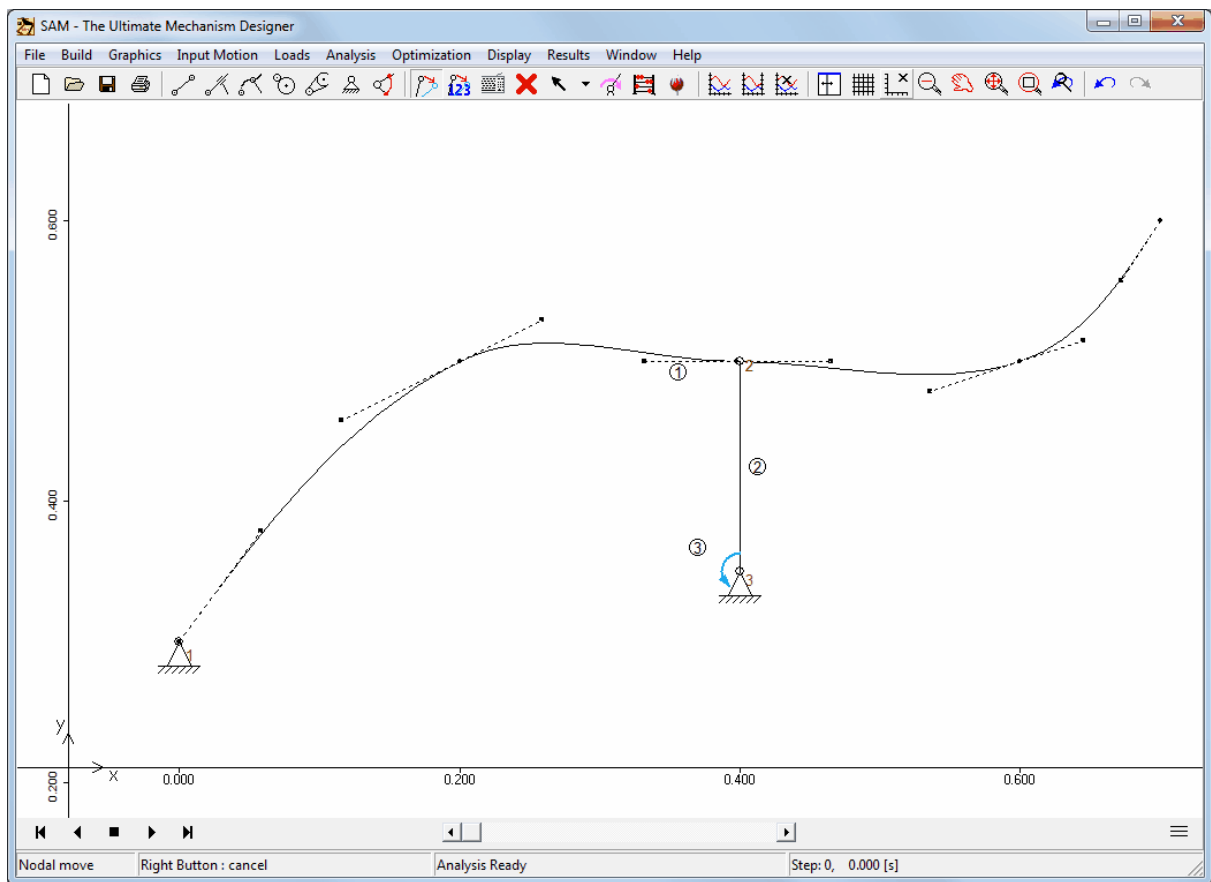
Hypocyclic Mechanism (HYPO.SAM)



Output displacement and velocity of hypo cyclic mechanism

10.1.6 Curved Slider

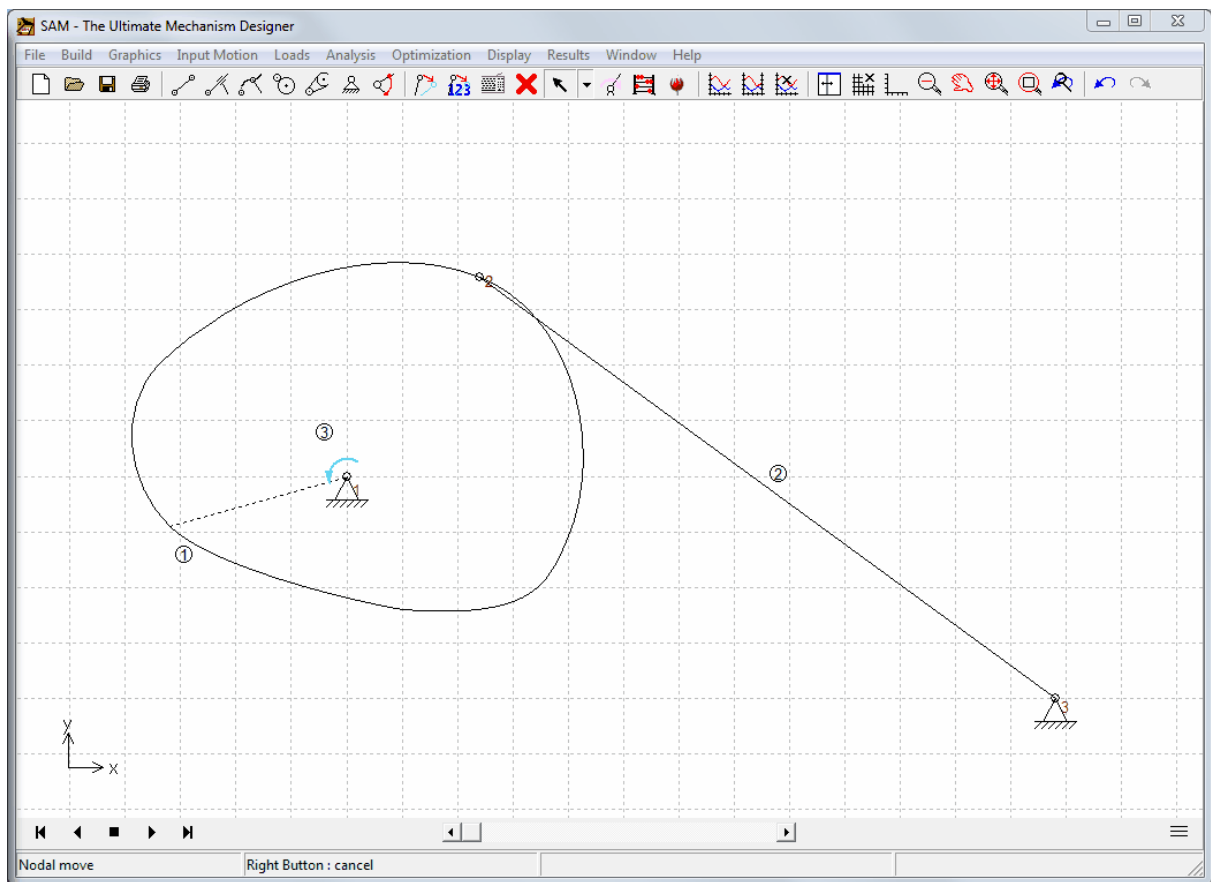
The curved slider element is a very versatile element that can be used to model any sort of non-straight slider based on Bezier segments.



Curved Slider Mechanism (CS.SAM)

10.1.7 CAM mechanism

The curved slider element in its closed version can be used to mimic a CAM profile.

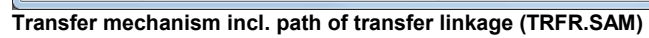


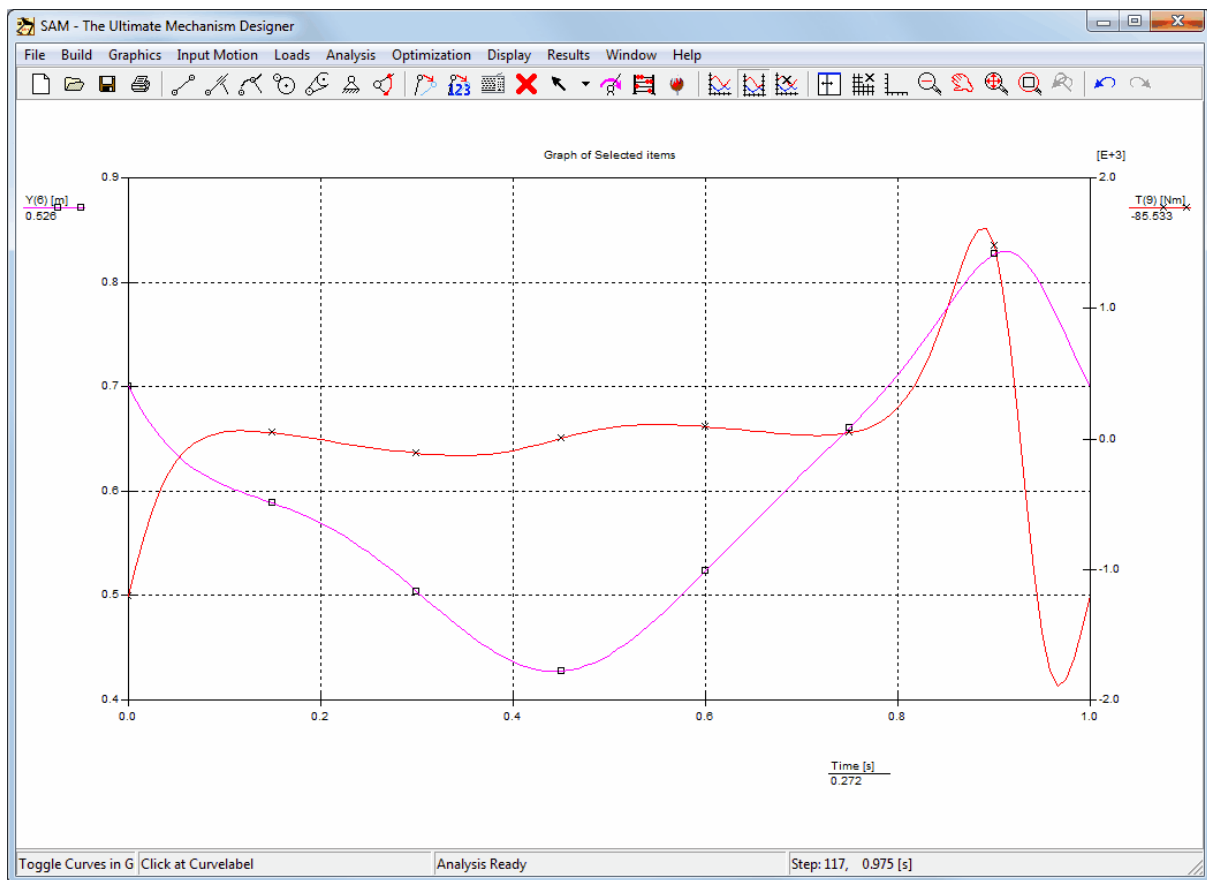
CAM Mechanism (CAM.SAM)

10.2 Force Analysis

10.2.1 Transfer Mechanism (incl. gravity effects)

The transfer mechanism can be used to transfer products without introducing any rotation. Due to the mass of the transfer linkage, which is modeled as discrete masses at the end nodes of that linkage, a certain static driving torque is required.

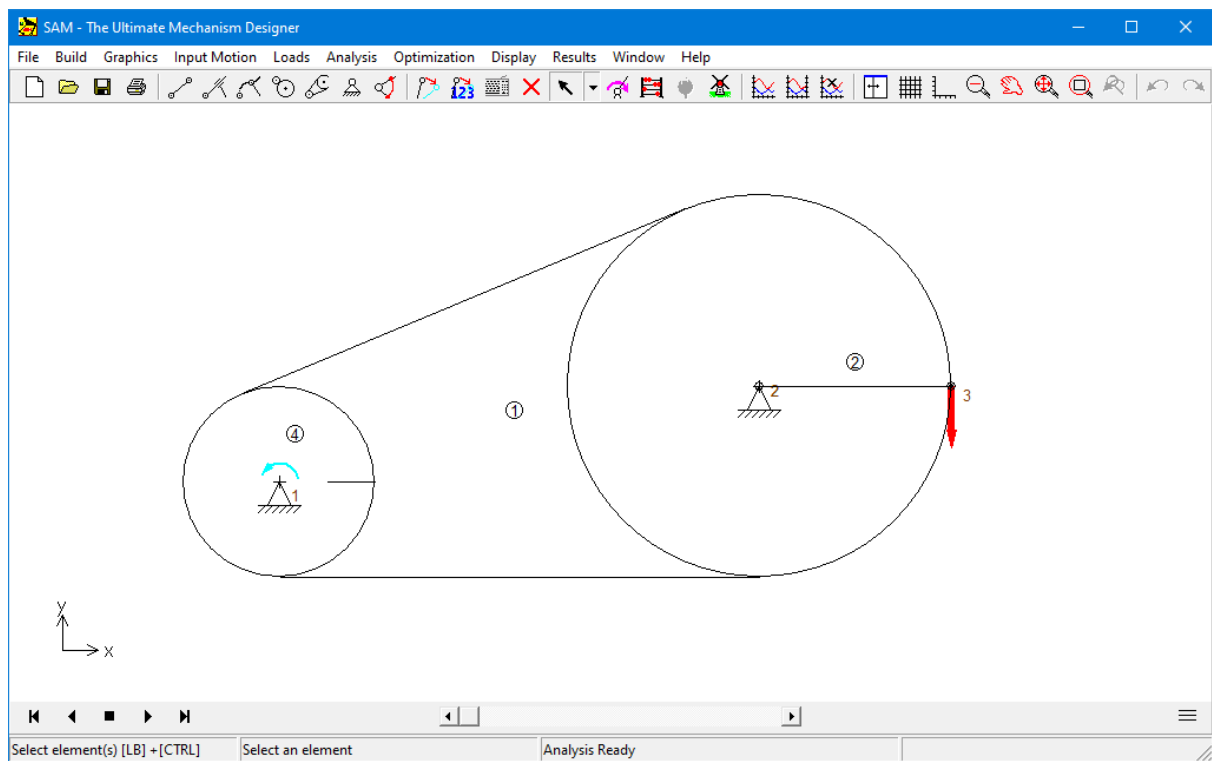




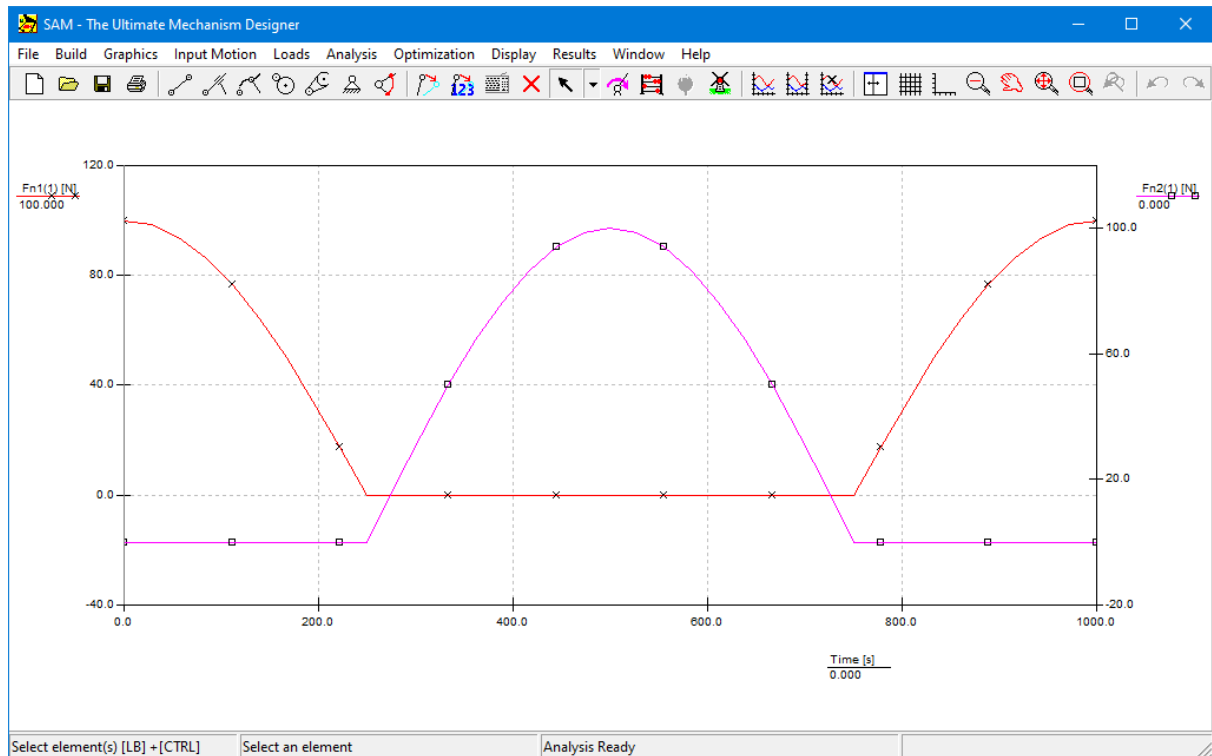
Vertical position transfer linkage & static driving torque

10.2.2 Belt drive with external force

The following example illustrates the force analysis of a belt drive without pretension that is loaded by an external force. Depending on the rotation of the wheel the vertical force acting in node 3 leads to positive or a negative torque on the second wheel. This torque needs to be counteracted by either a positive force (tension) in the upper or a positive force in the lower part of the belt drive, since a non-loaded belt drive can not transfer negative forces (pressure). In case of a tension force in the upper part the force in the lower part is equal to zero and vice versa.



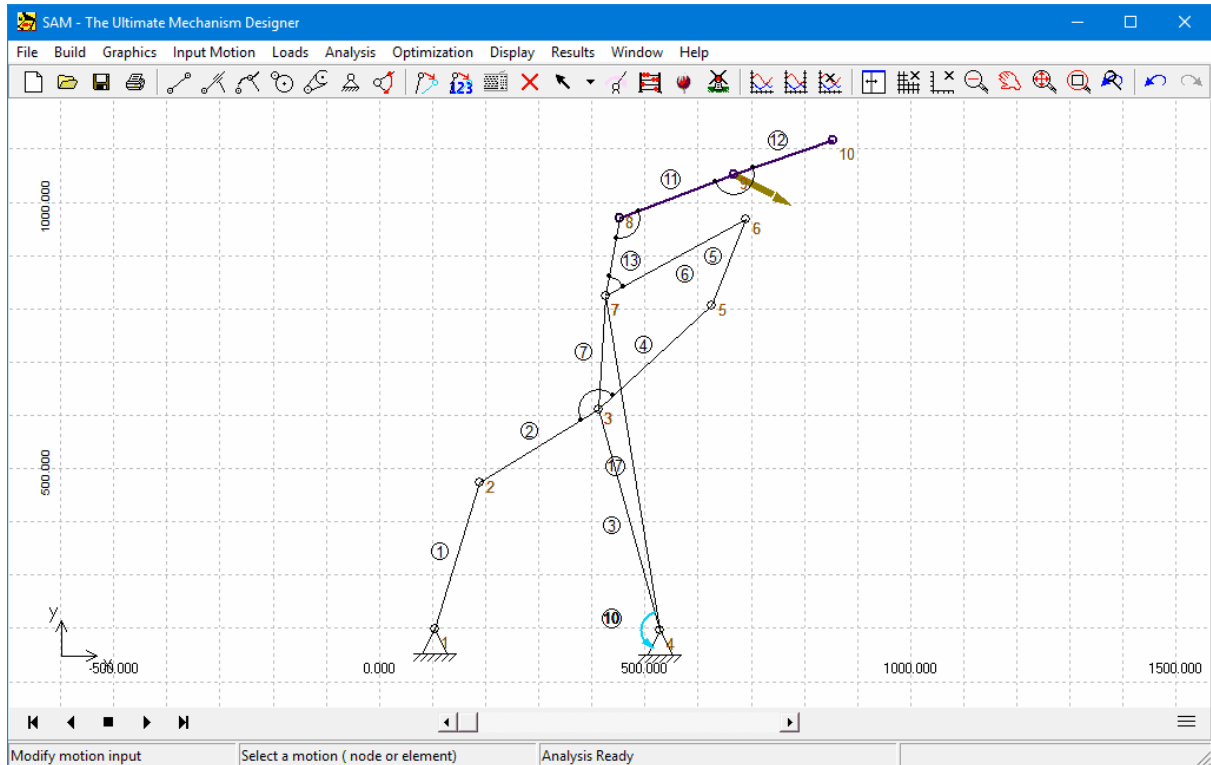
Belt drive with external vertical force in node 3 (FBELT.SAM)



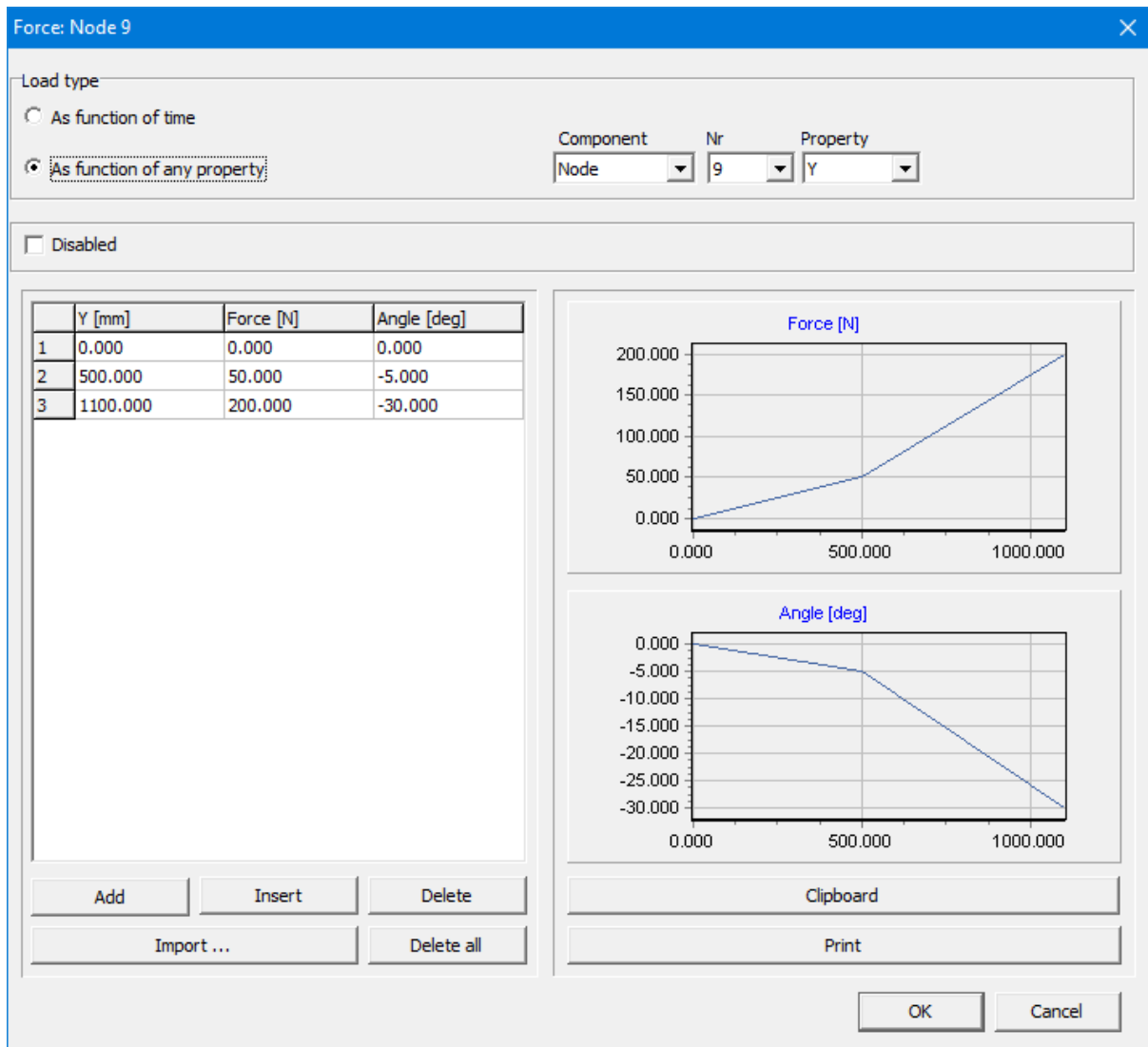
Forces FN1 in upper belt part and FN2 in lower belt part (Note: shifted y-scales)

10.2.3 Height dependent forces on sports car spoiler

Sport cars often have adjustable spoilers in order to generate down force. The value and direction of these forces depends not only on the speed of the car but also on the adjustable height of the spoiler. The spoiler is modeled by the two blue elements 11 and 12, which share node 9. The forces acting on the spoiler at a certain speed of the car have been measured and are modeled as a variable force (value & direction) that depends on the height of node 9.



Spoiler mechanism with a force on node 9 (SCS.SAM)



Force on node 9 as function of the current Y-coordinate of node 9

10.3 Optimization

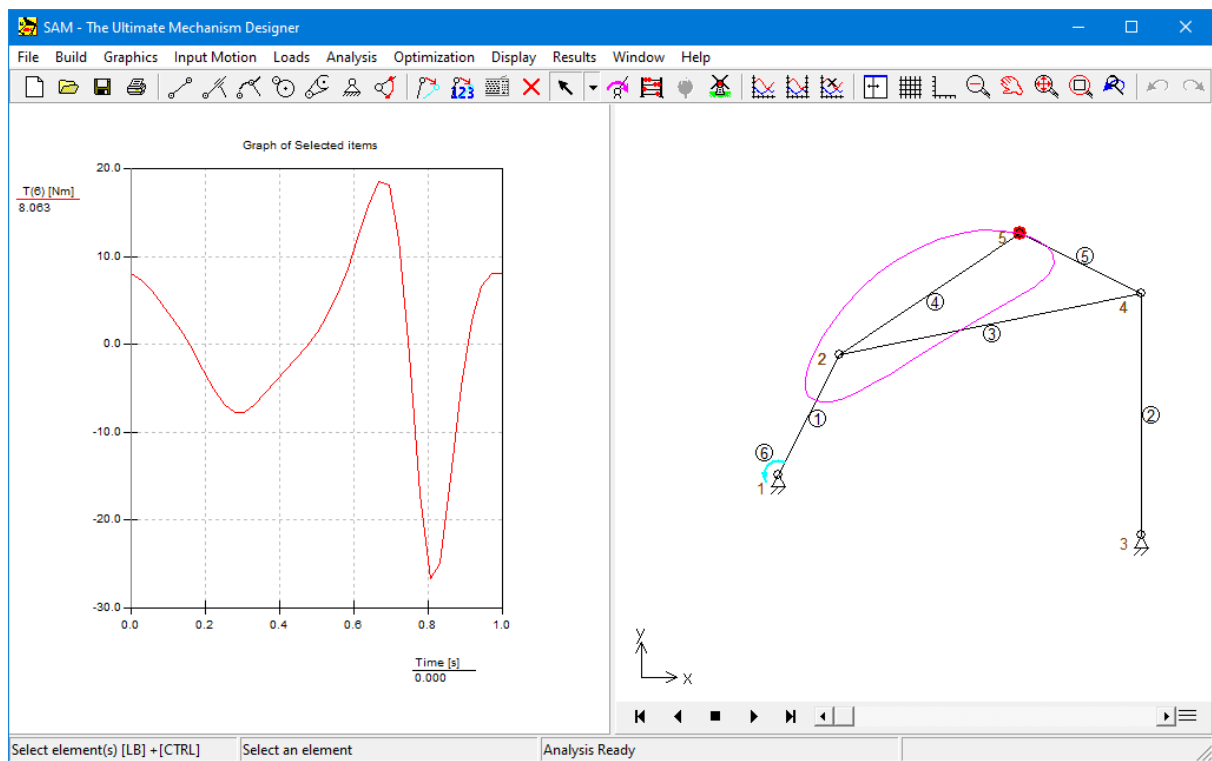
10.3.1 Driving Torque Minimization

As a first example, the reduction of the driving torque of a 4-bar mechanism with mass in the coupler point will be demonstrated. This is achieved by the addition of one single mass in the coupler plane. The mechanism is driven by a constant angular velocity at the crank and gravity effects are excluded from the study. The optimization algorithm must search for the optimal location and value of the compensating mass.

The example is based on the default 4-bar mechanism (4-bar design wizard). In addition to that a mass of 10kg is added to point 5 of the mechanism. The input motion consists of a constant velocity that corresponds to a motion of 360 degree in 1 s. The analysis is divided into 36 intervals.

Reference Mechanism

In the reference situation the driving torque as a function of time will look like depicted in figure below. The absolute maximum of the driving torque equals 26.6 Nm.



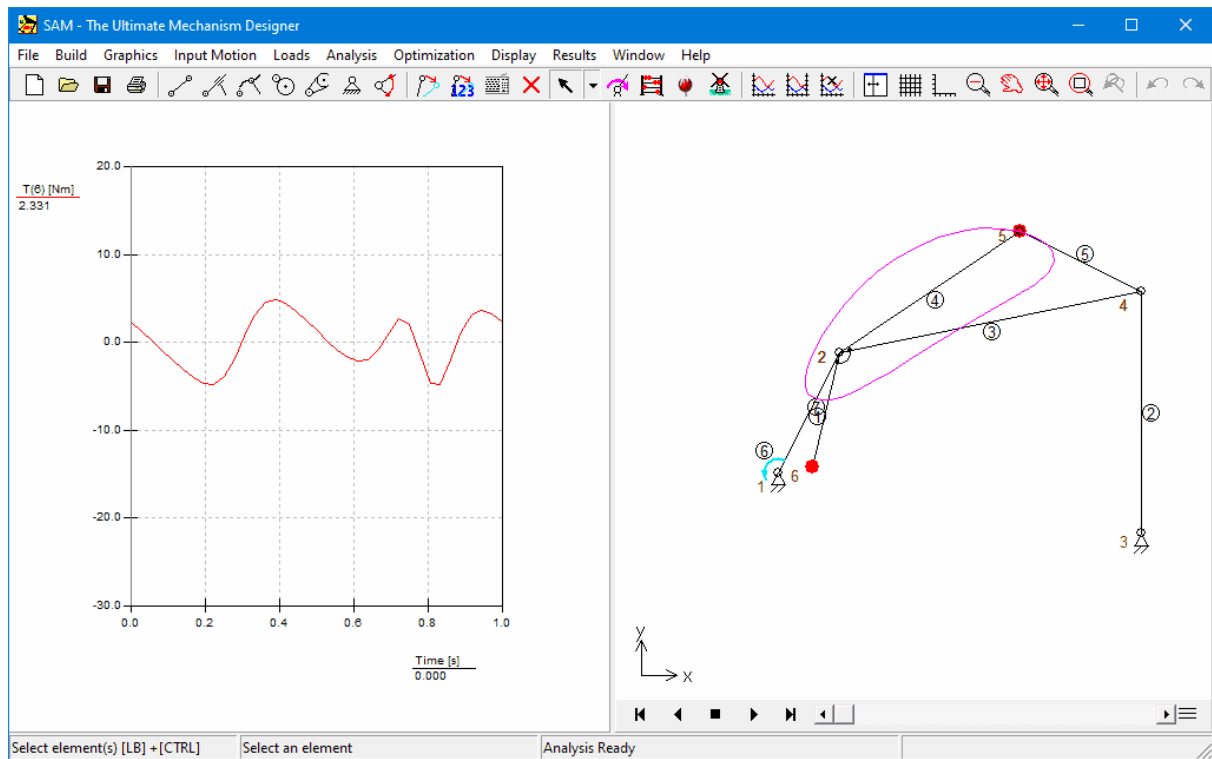
Reference Mechanism (a. Driving Torque b. Mechanism and path of coupler point)

Optimization Parameter Range

The maximum absolute value of the driving torque is reduced by adding a single mass to the coupler linkage. The search region for the mass was set at 0-20kg, whereas the search region for the location was set from -1.0m to + 1.0m in both directions.

Optimized Mechanism

The value ($m=16.5$ kg) and location ($x=0.257$ m and $y=0.531$ m) is derived by the optimization module (fig.14). The corresponding maximum absolute value of the driving force in the optimized situation equals 4.76 Nm.



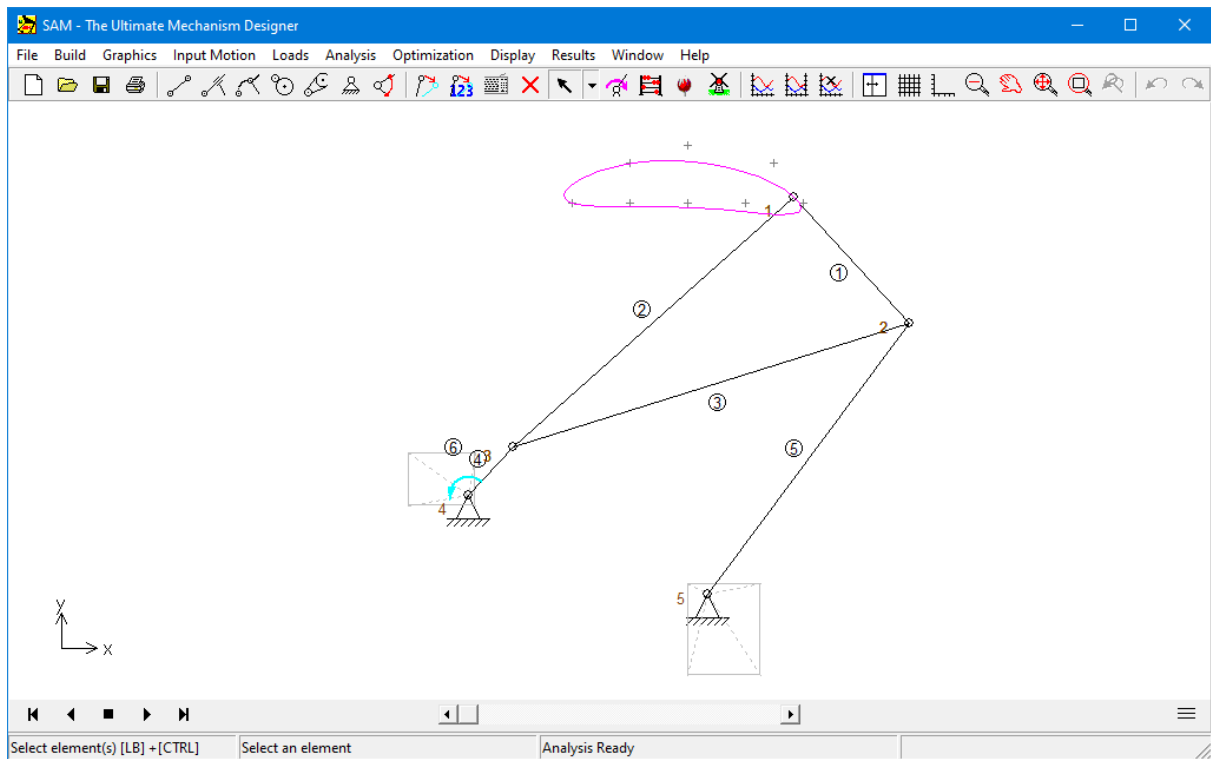
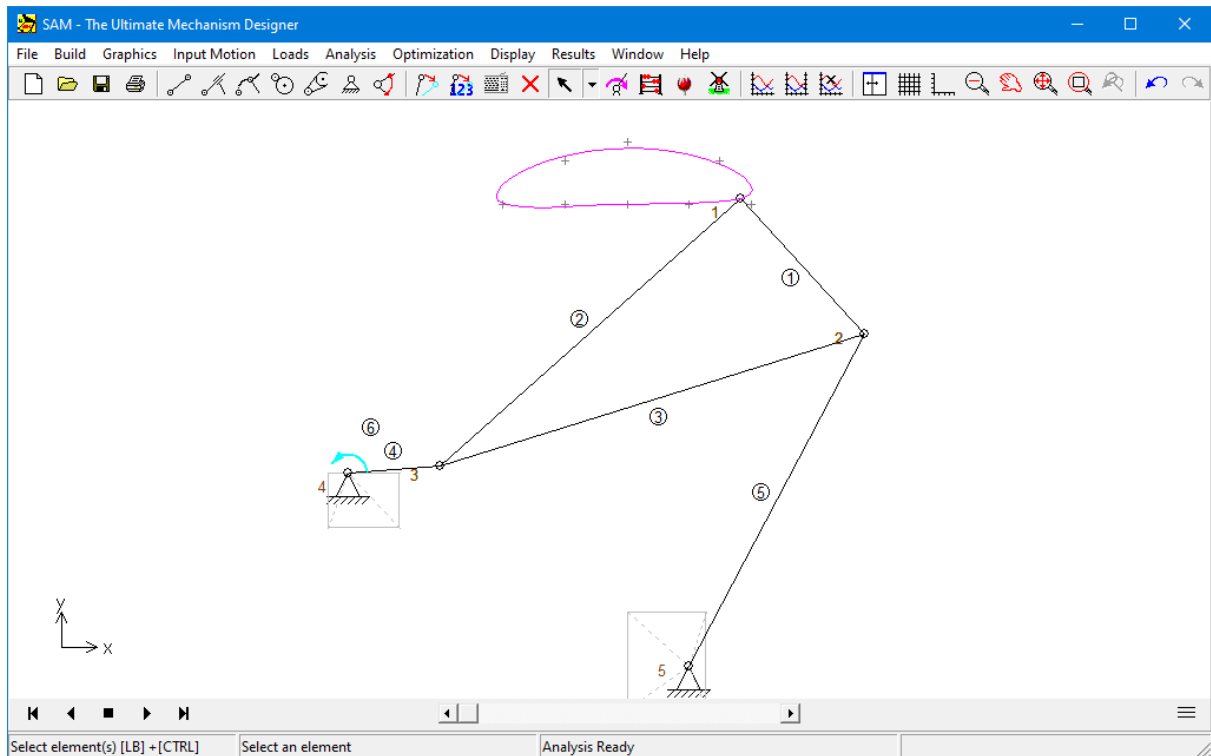
Optimized Mechanism (a. Driving Torque b. Mechanism + compensating mass)

A significant reduction of the driving torque was achieved by adding a compensating mass to the coupler plane. The value of the compensating mass and its location was automatically determined by an optimization procedure based on an evolutionary algorithm.

10.3.2 Optimization of trajectory

SAM can also optimize the parameters of a mechanism such that a desired trajectory is followed as good as possible. The first screenshot below shows the result of a manual attempt to find a 4-bar mechanism with a coupler point motion that mimics the Bezier curve through 8 specified points.

This mechanism is used as a starting point for an optimization in which the RMS-value of the deviation of the actual trajectory from the target curve is chosen as optimization objective function and the positions of the nodes of the mechanism are the design parameters. All moving nodes can be chosen arbitrarily, whereas the support points are limited in their range, as indicated. As can be seen the mechanism that resulted from the optimization process shows significantly better correspondence with the requested target path.

**Manually chosen 4-bar mechanism****Optimized 4-bar mechanism**

10.3.3 Constrained Optimization

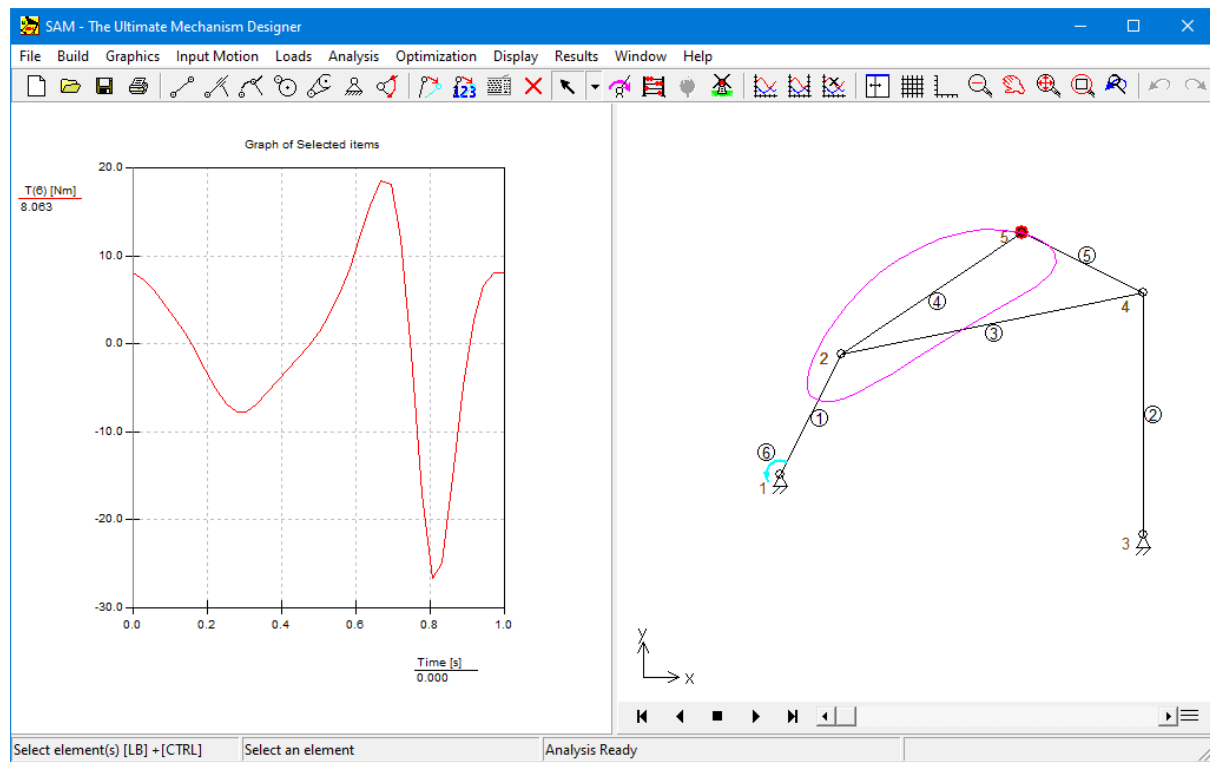
In this example we will show constrained optimization by adding a penalty function to the original cost function in order to incorporate the constraint.

This will be demonstrated using the previous example “Driving Torque Optimization” and adding an extra constraint. The example is based on the default 4-bar mechanism that is generated by the command sequence File>Wizard>4 Bar Mechanism and by accepting the default values. The mechanism is driven by a constant angular velocity at the crank (360 degree in 1 s, divided into 36 intervals) and gravity effects are excluded from the study. A mass of 10kg is added to point 5 of the mechanism.

The optimization algorithm must search for the optimal value and location of the compensating mass. In this example, we want to add the extra constraint, that the length of the extra element, that is added to support the compensating mass, should not be more than 0.05m.

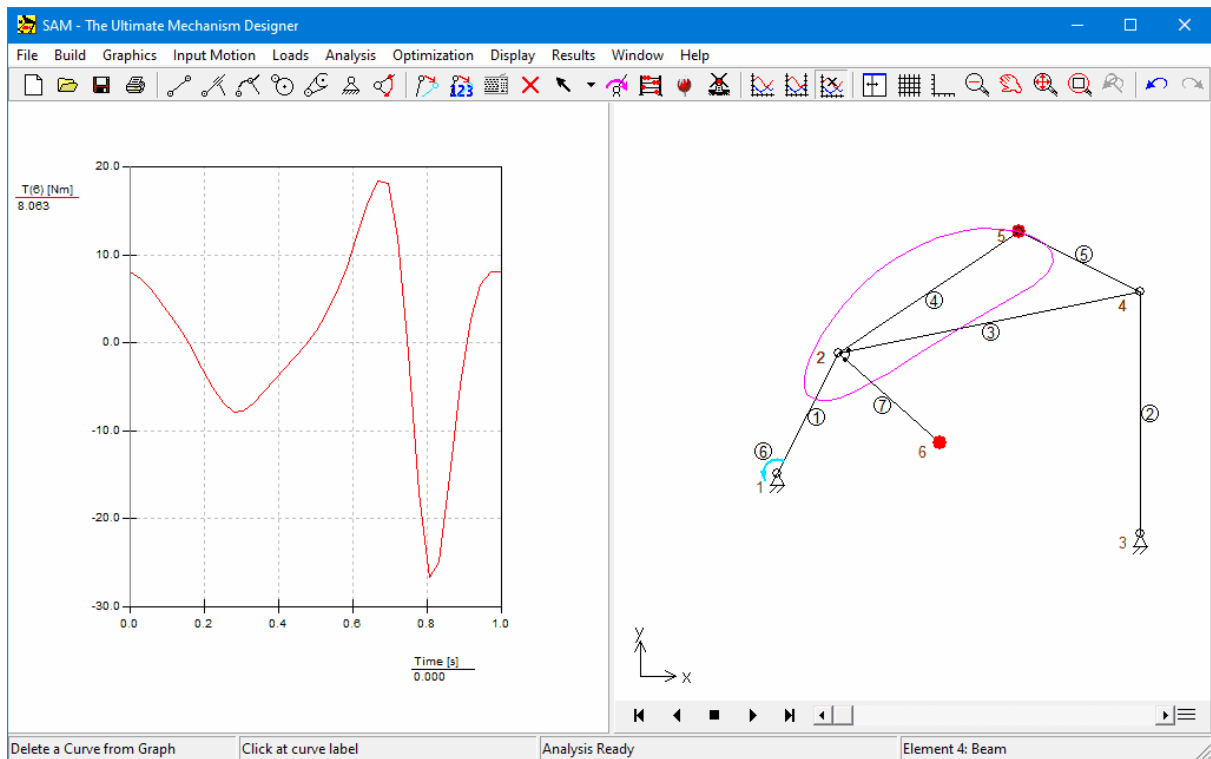
Reference Mechanism

In the reference situation the driving torque as a function of time will look like depicted in the following picture. The absolute maximum of the driving torque equals 26.6Nm.



Reference mechanism

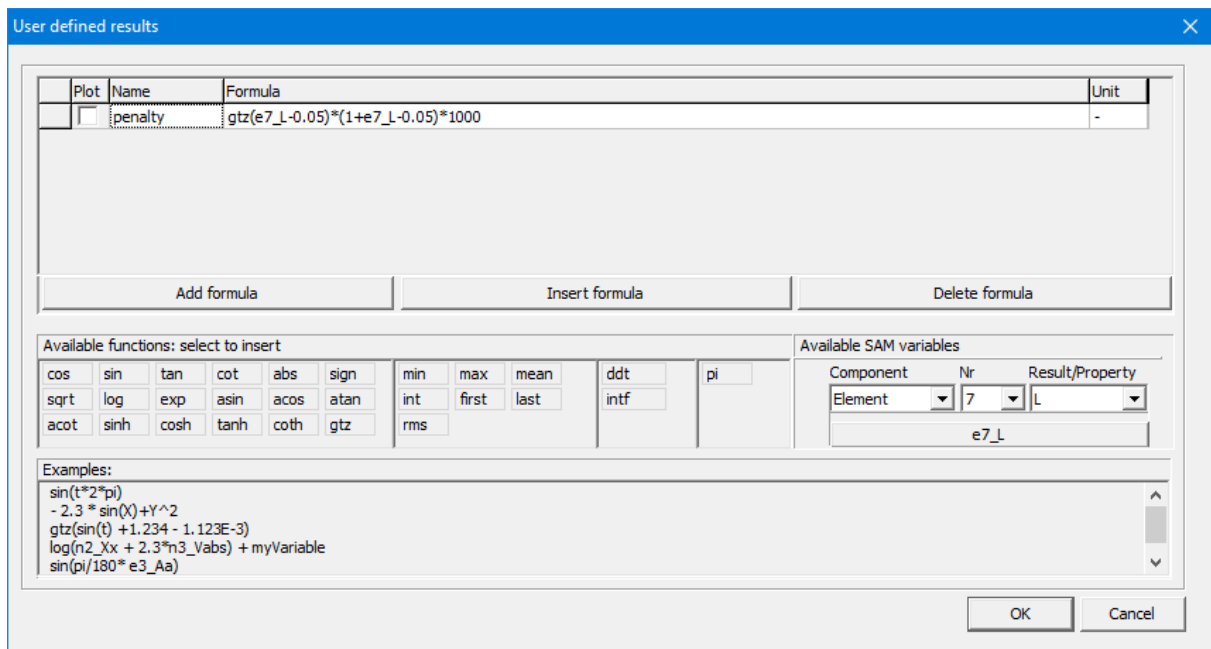
The next step is to add an extra element (nr. 7) at the end of which the compensation mass is located via element properties. The optimization module can't handle optimization of an individual mass and therefore the mass has to be added via element properties.



Reference mechanism including element 7 with the compensation mass

Optimization

In order to incorporate the constraint a user defined variable needs to be created via Results>User defined...



The constraint of the length of element 7 is converted into a penalty

A penalty is defined in case the length of element 7 (SAM variable: e7_L) is larger than 0.05m. The penalty is based on the Heaviside or *Greater Than Zero* function GTZ(x), which yields an output of 1 if $x \geq 0$ and an output of 0 if $x < 0$. In order to achieve a larger penalty for larger deviations from the target the outcome is multiplied by the actual deviation. Finally a weighting function of 1000 is applied based on engineering judgment. The outcome of the penalty function should be large enough to significantly impact the overall cost function but not too large to lead to numerical problems.

Optimization Objective

The settings of the optimization are shown in the following screenshot. The maximum absolute value of torque T1 of element 1 is chosen as objective to minimize given the constraint that the length of element 6 should be smaller than 0.05m. This constraint is translated into a penalty that is added to the value of the cost function.

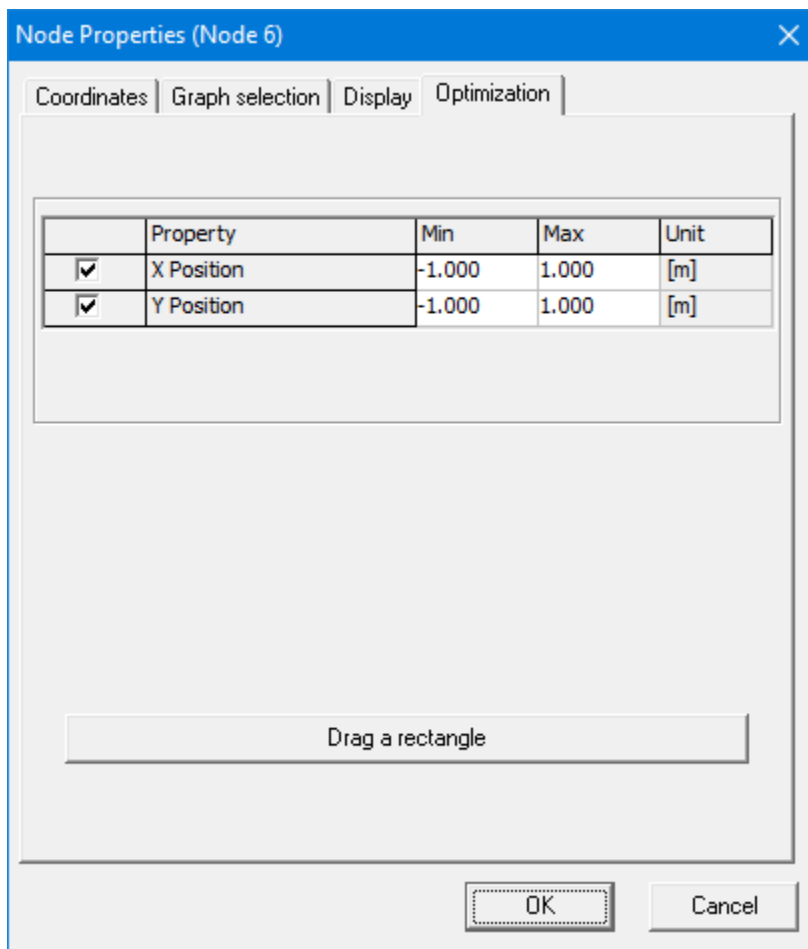
The screenshot shows the 'Setup objective' dialog box with the following settings:

- Step 1: Objective type: Function
- Step 2: Objective details: Edit...
- Step 3: Objective reference file (optional): Edit...
- Step 4: Objective performance number: Maximum abs.
- Step 5: Penalty function (optional): penalty
- Step 6: Optimization target: Minimize

Buttons: OK, Cancel

Optimization Parameter Range

The maximum value of the driving torque (or actually the maximum value of the cost function) is reduced by adding a single mass to the coupler linkage. The search region for the location was set at (-1.0,-1.0) up to (1.0,1.0).



The image shows a software dialog box titled "Node Properties (Node 6)". It has four tabs: "Coordinates", "Graph selection", "Display", and "Optimization". The "Coordinates" tab is selected. Inside the dialog, there is a table with the following data:

	Property	Min	Max	Unit
<input checked="" type="checkbox"/>	X Position	-1.000	1.000	[m]
<input checked="" type="checkbox"/>	Y Position	-1.000	1.000	[m]

Below the table, there is a large rectangular area with the text "Drag a rectangle" centered inside it. At the bottom right of the dialog, there are two buttons: "OK" and "Cancel".

The search region for the mass of element 7 was set at 0-40kg (note: you need to set the location of the mass within the element such that it is located at node 6; which means that depending on the sequence in which element 7 was created – from node 2 to node 6 or the other way around – one needs to enter the value for relative distance to first node of either 0.0 or 1.0).

Element Properties (Beam 7) ✕

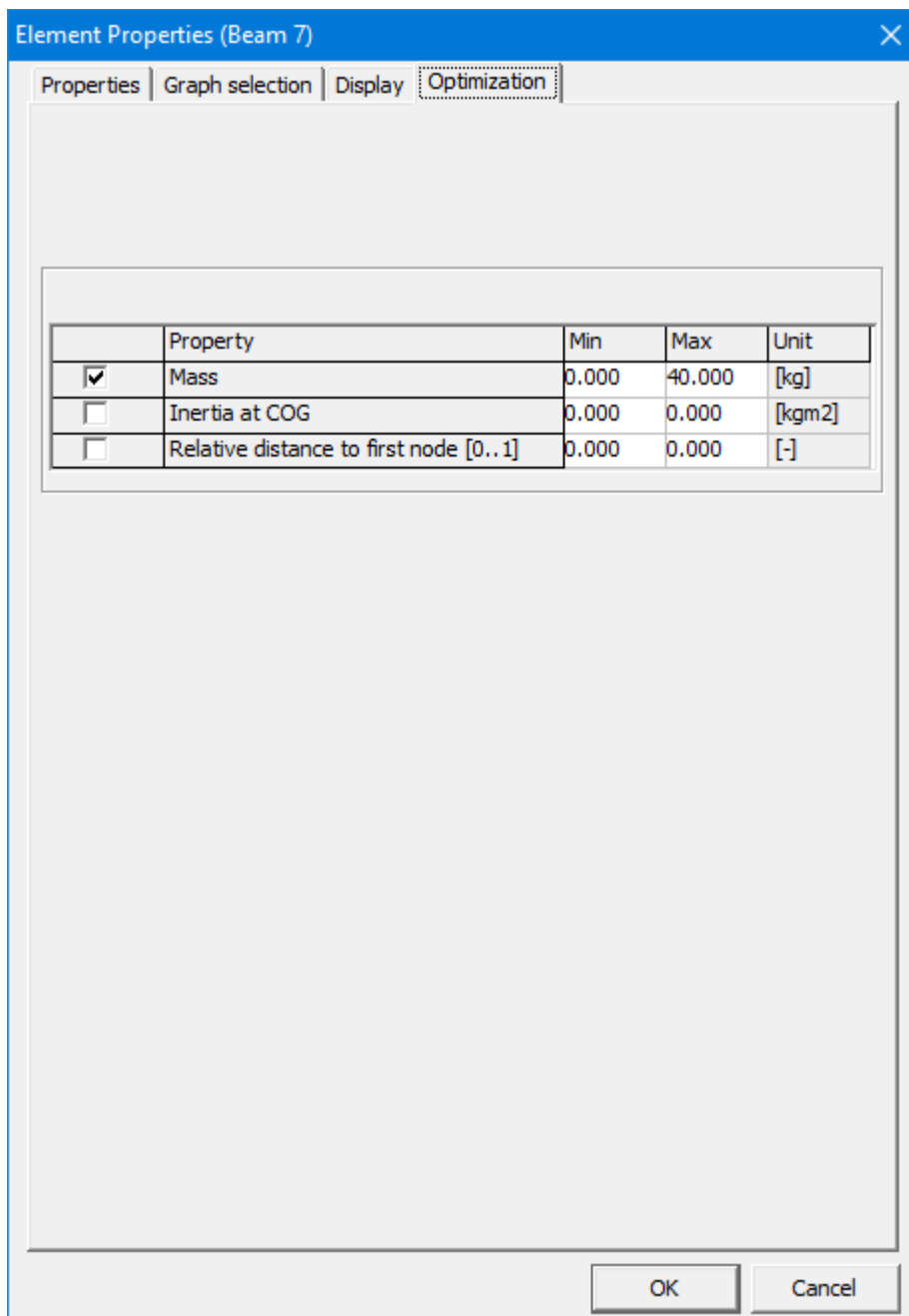
Properties | Graph selection | Display | Optimization

Element Nr	7	
Node 1	2	
Node 2	6	
Length	0.225	[m]
Angle	318.297	[deg]

Mass	0.001	[kg]
Mass moment of inertia (at center of gravity)	0.000	[kgm2]
Relative distance to first node [0..1]	1.000	[-]

Trackbar...

OK Cancel



Optimization Options

For the sake of simplicity, the automatic modus was chosen. In this modus a global exploration is performed based on an evolutionary algorithm using the settings as shown. The best results of that global approach are then further optimized automatically - in this case using the Simplex method as indicated.

Modus

- Choosing AUTOMATIC is the easiest way to do an optimization.
- The checkbox "Optimize all solutions from global exploration" indicates that after the global exploration of the solution space (always done via evolutionary algorithm), ALL best results of the

exploration are further improved via a local optimization. After the global exploration a pool of best results exists and the size of that pool of results is equal to the Population Size defined in the Evolutionary Settings. So in this case 300 evolutionary (or genetic) experiments are done based on a population size of 30 "members". At the end a pool of 30 best members or solutions exist. If the checkbox is checked, all these 30 best results are further optimized via local optimization. If the checkbox is NOT checked, only the best solution found in the global exploration is further improved via local optimization.

Local Optimization

- In general using the Simplex Method for the local optimization yields good results.

Evolutionary Setting

- The evolutionary algorithm starts with filling a pool or population of results via a random process (Population Size indicates the size of the population). This population is then improved via a evolutionary (nature inspired genetic) process. The number of genetic experiments is indicated by Nexperiments.

Simplex Setting

The main setting for the Simplex Method is the max. number of iterations. The default value is generally OK but based on the progress that is monitored online one can decide that a smaller number is sufficient. On the other hand, if one observes significant progress just before reaching the max. number of iterations it is very likely that continuing the process yields better results and it makes sense to increase this number.

In principle the boundaries of the optimization space should always be respected. However, sometimes it is very interesting – and one may be triggered to find new solutions – to allow the algorithm to disrespect these boundaries. This is the reason why the user has a choice.

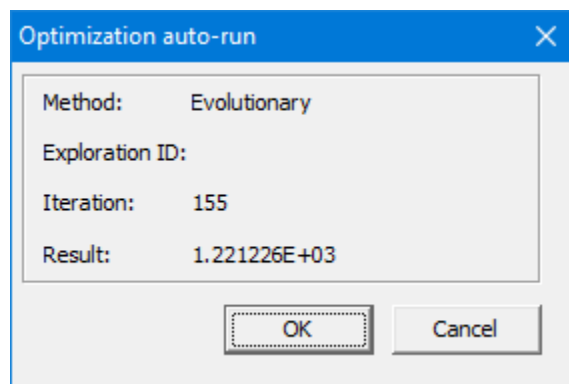
If the process takes too long one can decide to do less experiments, choose a smaller population size (though it should always be at least 20-30 to give the evolutionary algorithm a chance), do less iterations in the local optimization process or decide NOT to further optimize all results from the global search but to only post-process the best result of the global search.

Optimization Run

In order to see how the optimization improves the results it's best to go to Results>Options and turn the AUTOSCALE option OFF. The vertical axes is then fixed to -30Nm to +20Nm.

Global Exploration

The status window shows the process of the global exploration.

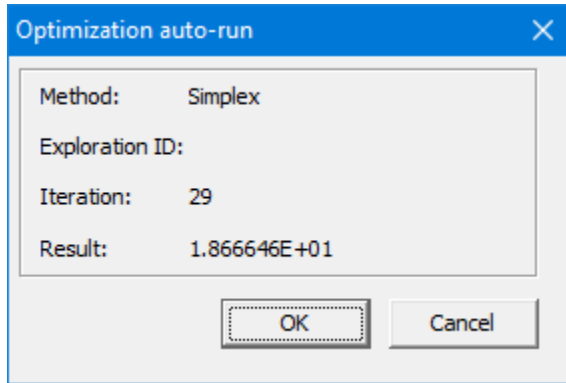


Method: The method for global exploration is always the Evolutionary Algorithm.
 Explore id: Empty (has only meaning when doing the local optimization).

Iteration: Current iteration step in the evolutionary process
Result: Current value of the experiment.

Local Optimization

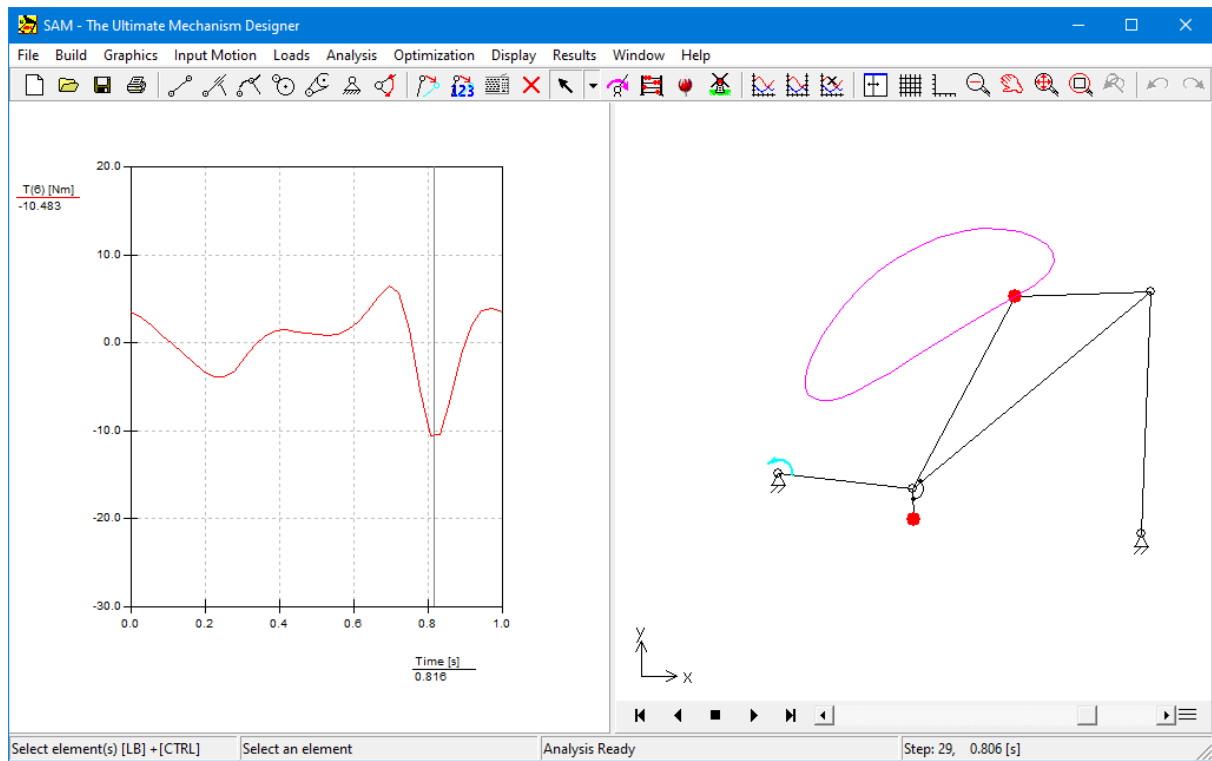
In this case all 30 results of the global exploration are improved via a local optimization process. The status window shows:



Method: Chosen method for local optimization
Explore id: ID of the solution of the global exploration that is currently locally optimized.
Iteration: Current iteration step in the local optimization process
Result: Current value of the optimization goal.

Optimized Mechanism

The value ($m=39.875\text{kg}$) and location ($x=0.28\text{m}$ and $y=0.65\text{m}$) is derived by the optimization module. The corresponding maximum absolute value of the driving force in the optimized situation equals 10.48Nm . This is the best solution that could be found given the constraint that the length of the extra element 6 should be less than 0.05m .



Conclusion

Via the automated optimization procedure a reduction of the driving torque was achieved by adding a compensating mass to the coupler plane and letting the optimization module determine the optimal values. The torque was reduced from 26.6Nm to 10.48Nm. This is the best solution that could be found given the constraint that the length of the extra element 6 should be less than 0.05m. As described in the example "Driving Torque Optimization" the result without this extra constraint was significantly better (4.6Nm).

Chapter

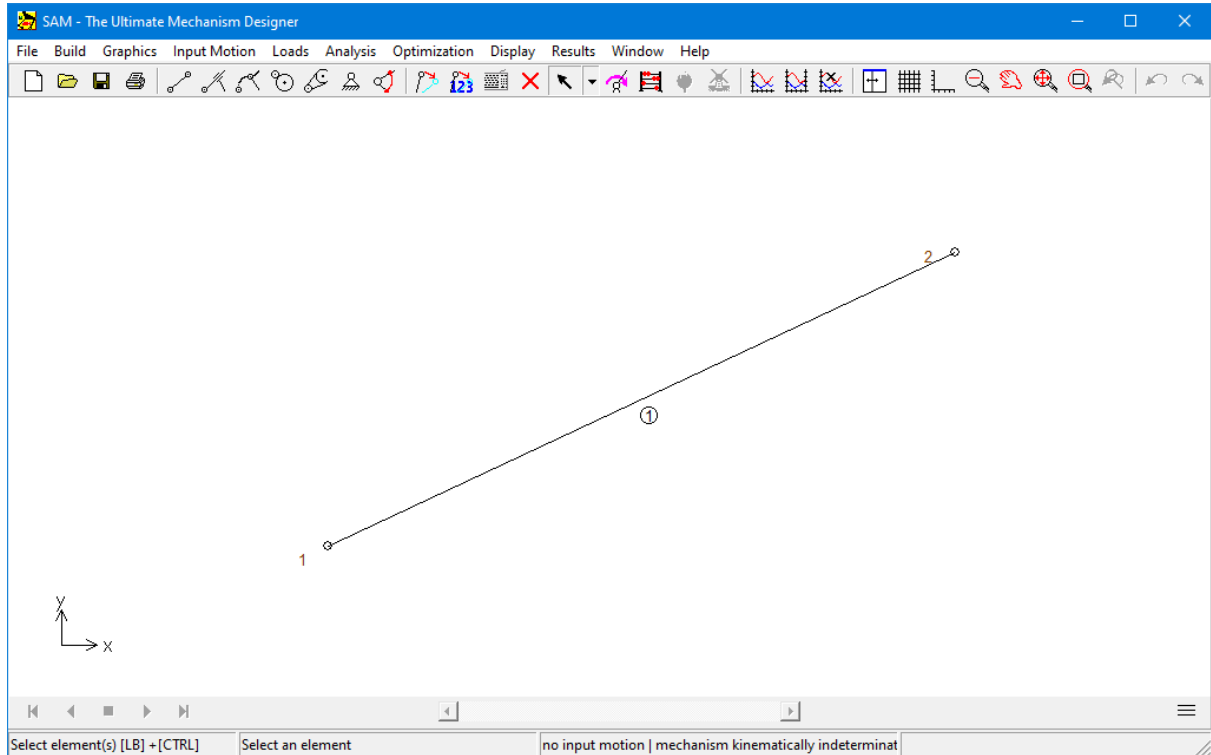


11

11 Elements

11.1 Beam

The beam element is the most basic element that is available in SAM. It is used to model any type of linkage.



Beam element

The beam element is defined by the position of its end nodes, which implies that the length of a beam is not an independent item but rather the result of the node coordinates. The length can be viewed in the property dialog, along with the element and node identification. In the property dialog, the user can also define inertia properties of a beam, such as :

- mass
- location center of gravity
- mass moment of inertia at center of gravity

The center of gravity is defined as (distance with respect to first node)/ (length beam) and it ranges from 0-1. The location is therefore restricted to a position on a line that connects the two nodes and must be between these nodes.

Element Properties (Beam 1)

Properties | Graph selection | Display | Optimization

Element Nr	1	
Node 1	1	
Node 2	2	
Length	1.101	[m]
Angle	25.162	[deg]

Mass	0.000	[kg]
Mass moment of inertia (at center of gravity)	0.000	[kgm2]
Relative distance to first node [0..1]	0.000	[-]

Trackbar...

OK Cancel

Properties of beam element

The user can define which properties of an element need to be calculated and displayed in a graph. In the case of a beam element the following items can be selected :

- length, elongation, elongation speed, elongation acceleration
- absolute and relative angle, angular velocity, angular acceleration
- normal force, normal power

- torque and corresponding power at both nodes

Element Properties (Beam 1)

Properties | Graph selection | Display | Optimization

Plot

X-axis

<input type="checkbox"/> Length	L
<input type="checkbox"/> Elongation	E
<input type="checkbox"/> Elongation velocity	Ev
<input type="checkbox"/> Elongation acceleration	Ea
<input type="checkbox"/> Normal force	Fn1
<input type="checkbox"/> Translational power	P
<input type="checkbox"/> Angle	A
<input type="checkbox"/> Relative angle	Ar
<input type="checkbox"/> Angular velocity	Av
<input type="checkbox"/> Angular acceleration	Aa

Related to Node: 1

<input type="checkbox"/> Torque	T1
<input type="checkbox"/> Power	P1
<input type="checkbox"/> Bearing Force X	Fb1x
<input type="checkbox"/> Bearing Force Y	Fb1y
<input type="checkbox"/> Bearing Force Abs	Fb1

Related to Node: 2

<input type="checkbox"/> Torque	T2
<input type="checkbox"/> Power	P2
<input type="checkbox"/> Bearing Force X	Fb2x
<input type="checkbox"/> Bearing Force Y	Fb2y
<input type="checkbox"/> Bearing Force Abs	Fb2

OK Cancel

Calculable properties of beam element

A positive force Fn1 indicates a tension force, which - in case of flexibility - would result in an elongation of the beam.

In most applications, the length of a beam element is fixed and as a result of that, the elongation and its derivatives are equal to zero. In the absence of elongation, the transmitted power in axial direction is also equal to zero (power=force*velocity).

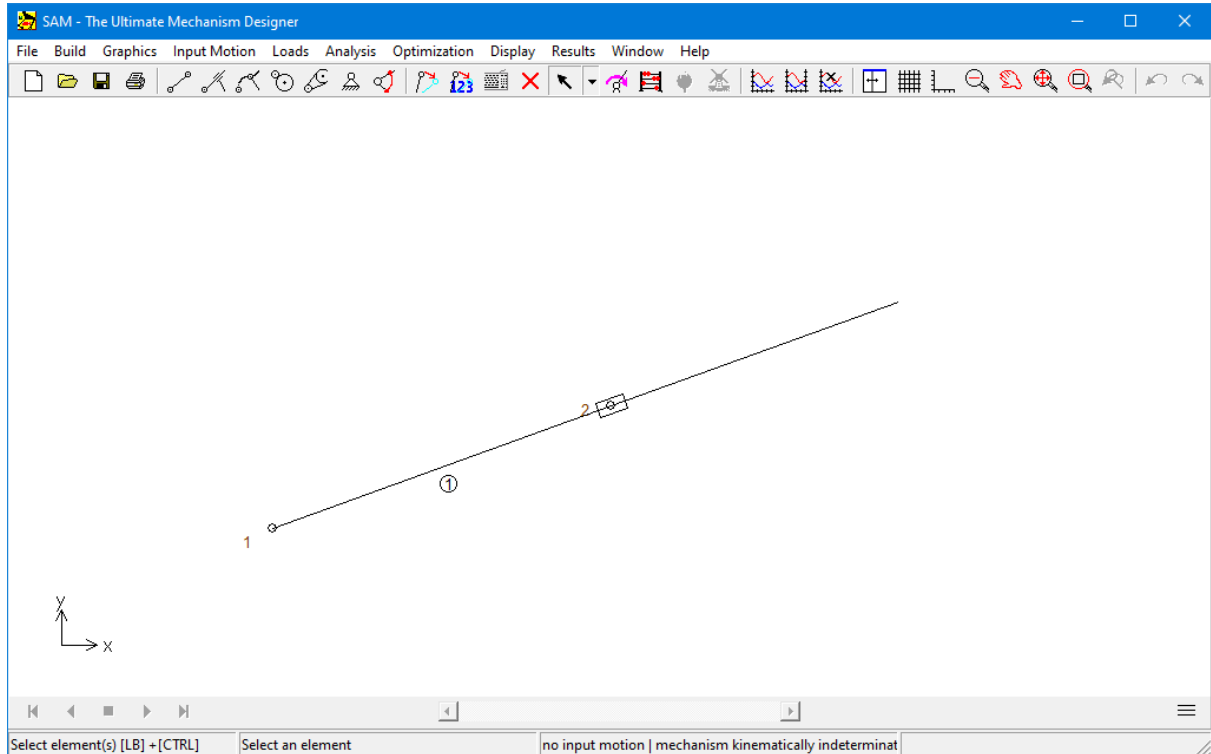
In the case of a linear actuator, which can be modeled as a beam element with prescribed elongation,

these quantities have the anticipated physical meaning.

Apart from the absolute angle of the element and its derivatives, one can also calculate the relative angle w.r.t. to the initial position. Furthermore, the resulting (driving) torque on the element at each of the nodes can be calculated. Finally, if a rotational input is defined at one of the nodes of the element, SAM can calculate the required driving power.

11.2 Slider

The slider element is basically nothing more than a beam element, with a non-fixed length. In order to achieve better graphical visualization, the user can specify the length of the slider.



Slider element

The slider element is defined by the position of its end nodes. The distance between these nodes is not restricted by the slider. As an extra property - only used for better graphical representation - the user can define the length of the slider. These properties, along with the element and node identification can be viewed in the property dialog, in which the user can also define inertia properties, such as :

- mass
- location center of gravity
- mass moment of inertia w.r.t. center of gravity

The center of gravity is defined as (distance w.r.t. first node) / (length element) and it ranges from 0-1. The location is therefore restricted to a position on the slider.

Element Properties (Slider 1) ✕

Properties | Graph selection | Display | Optimization

Element Nr	1	
Node 1	1	
Node 2	2	
Length	0.573	[m]
Angle	19.899	[deg]

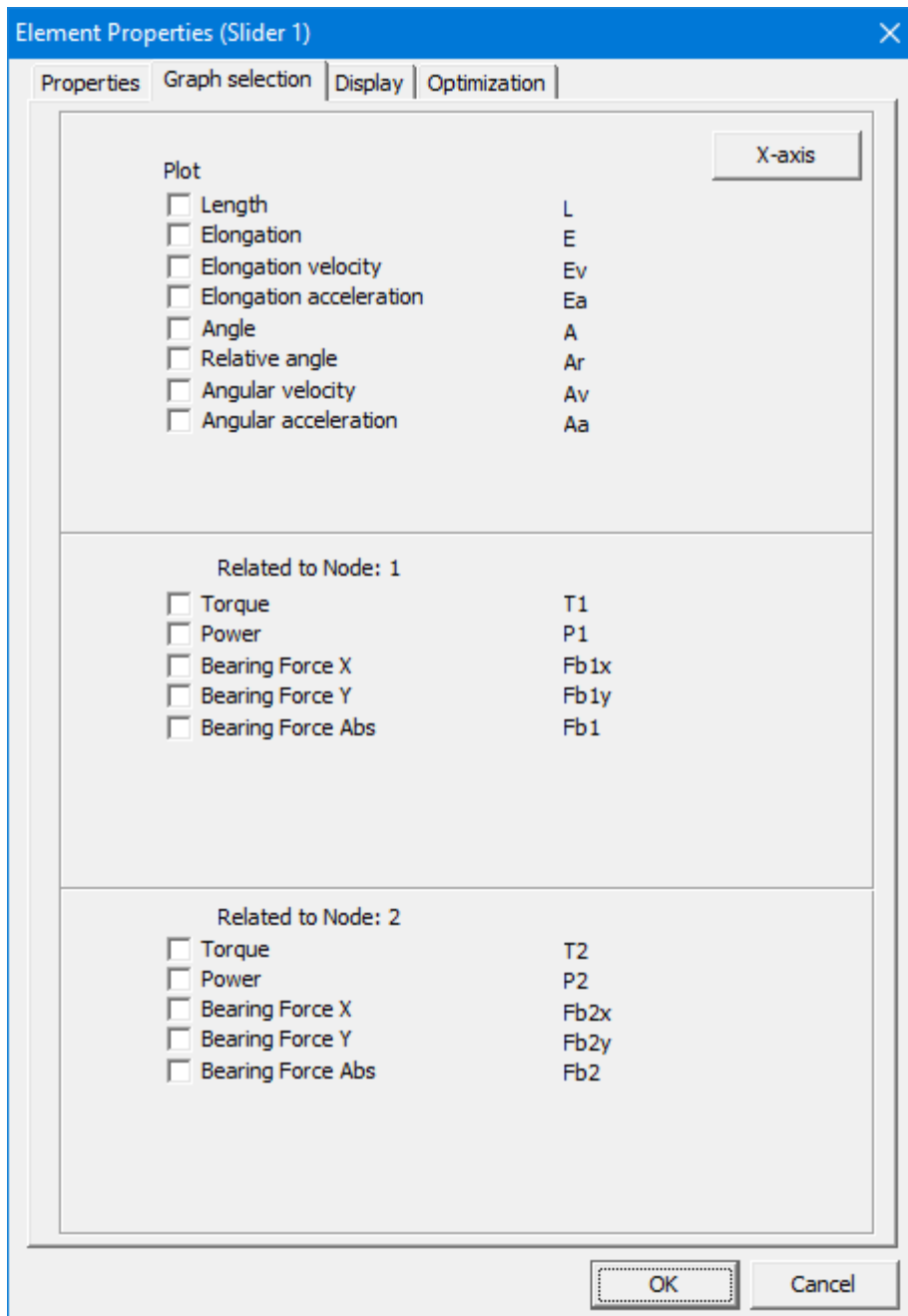
Sliderlength	1.060	[m]
Mass	0.000	[kg]
Inertia at COG	0.000	[kgm2]
Rel.distance to rod-end	0.000	[-]

Trackbar...

Properties of slider element

The following items can be selected as result of the analysis process :

- length, elongation, elongation speed, elongation acceleration
- absolute and relative angle, angular velocity, angular acceleration
- torque and corresponding power at both nodes



Calculable properties of slider element

The output selection dialog of the slider is equal to that of a beam element, except that the item "normal force" and "normal power" is greyed. Since the elongation of a slider is not restricted, normal forces can not be transmitted and the corresponding power is naturally also equal to zero. The item "Length" refers to the distance between the two nodes, whereas elongation and its derivatives have the anticipated meaning.

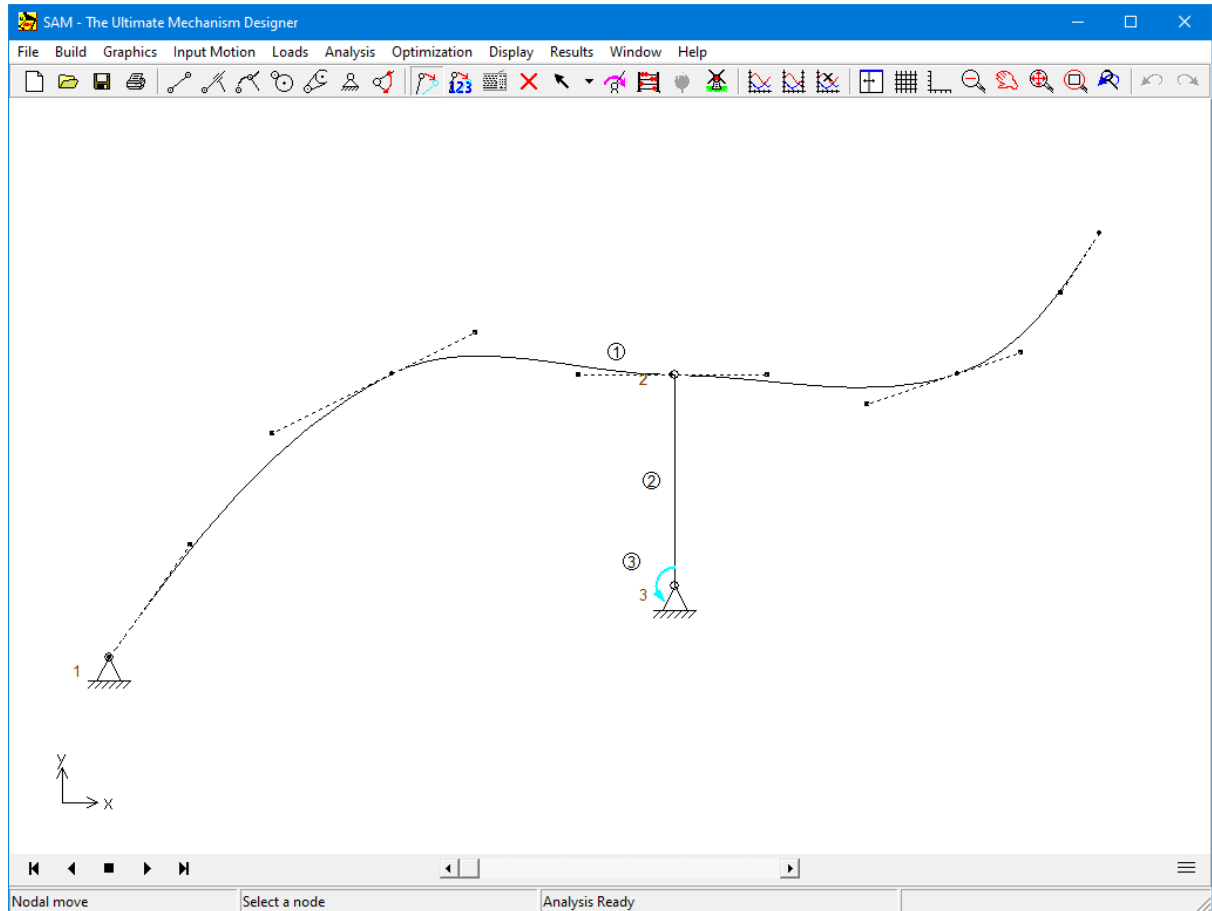
Apart from the absolute angle of the element and its derivatives, one can also calculate the relative angle w.r.t. to the initial position. Furthermore, the resulting (driving) torque on the element at each of the nodes can be calculated. Finally, if a rotational input is defined at one of the nodes of the element, SAM can calculate the required driving power.

11.3 Curved Slider

The curved slider element is a very versatile element that can be used to model any sort of non-straight slider based on Bezier segments. Each segment is defined by two (end) points and two intermediate control points.

The definition of a curved slider element consists of two steps:

- definition of the first and second node, which can be new or existing nodes
- selection of an existing Bezier curve or creation of a new Bezier curve



Curved Slider element 1 with nodes 1 and 2 (a beam element 2 serves as crank in this example)

The curve slider element is defined by the position of its end nodes and the angle corresponding to the first node. The second node is free to move along the curve but not perpendicular to it.

The curved slider element does not have any inertia properties of its own. If one wishes to add inertia an extra beam element has to be attached and inertia needs to be added to that extra element.

The Bezier points are defined in the global coordinate system. The geometric properties of a curved slider element can be adjusted via its element properties. This can be done by editing the table, but it is also possible to import data from a text file that contains a list of x/y coordinates. One can choose between:

- **Import points ...** => Import file contains only points, control points are generated automatically.
- **Import points and control points ...** => Import file must contain points and control points.

If the last point in an import file coincides with the first point, the curve is automatically considered a closed Bezier curve.

Element Properties (Curved Slider 1) ✕

Properties | Graph selection | Display | Optimization

Element Nr	1
Node 1	1
Node 2	2
Nodal Distance	0.517 [m]

	X [m]	Y [m]
1 (point)	0.000	0.300
2 (control point)	0.022	0.408
3 (control point)	0.105	0.484
4 (point)	0.200	0.500
5 (control point)	0.300	0.517
6 (control point)	0.354	0.511
7 (point)	0.409	0.481
8 (control point)	0.482	0.449
9 (control point)	0.517	0.523
10 (point)	0.581	0.544
11 (control point)	0.635	0.566
12 (control point)	0.672	0.557
13 (point)	0.700	0.600

Add

Insert

Delete

Import points ...

Delete all

Import points and control points ...

☒ When moving node 2: also translate the curve.

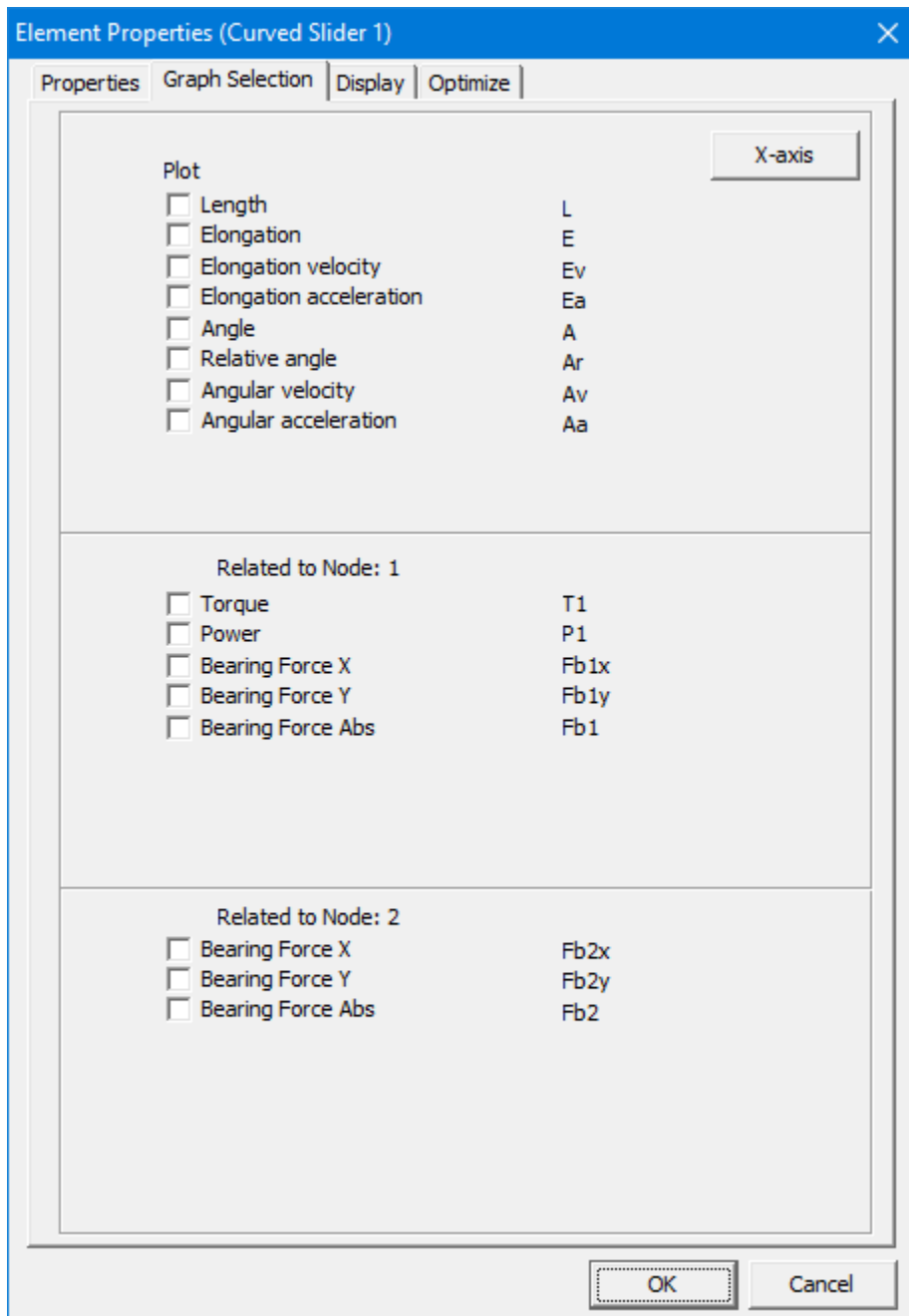
OK

Cancel

Properties of curved slider element

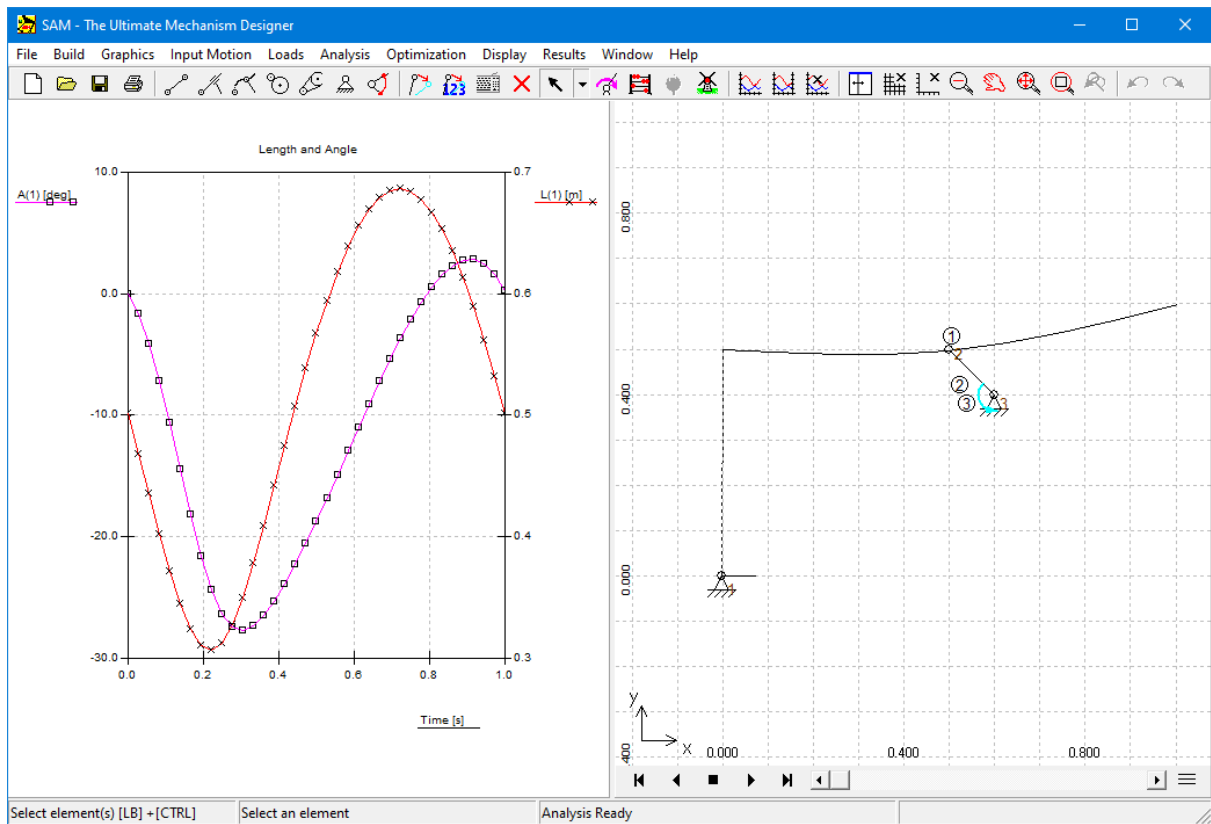
The following items can be selected as result of the analysis process :

- length, elongation, elongation speed, elongation acceleration (length incl. its derivatives is measured along the curve starting from the first point of the curve to the second node)
- absolute and relative angle, angular velocity, angular acceleration
- bearing forces
- torque and corresponding power at first node



Calculable properties of curved slider element

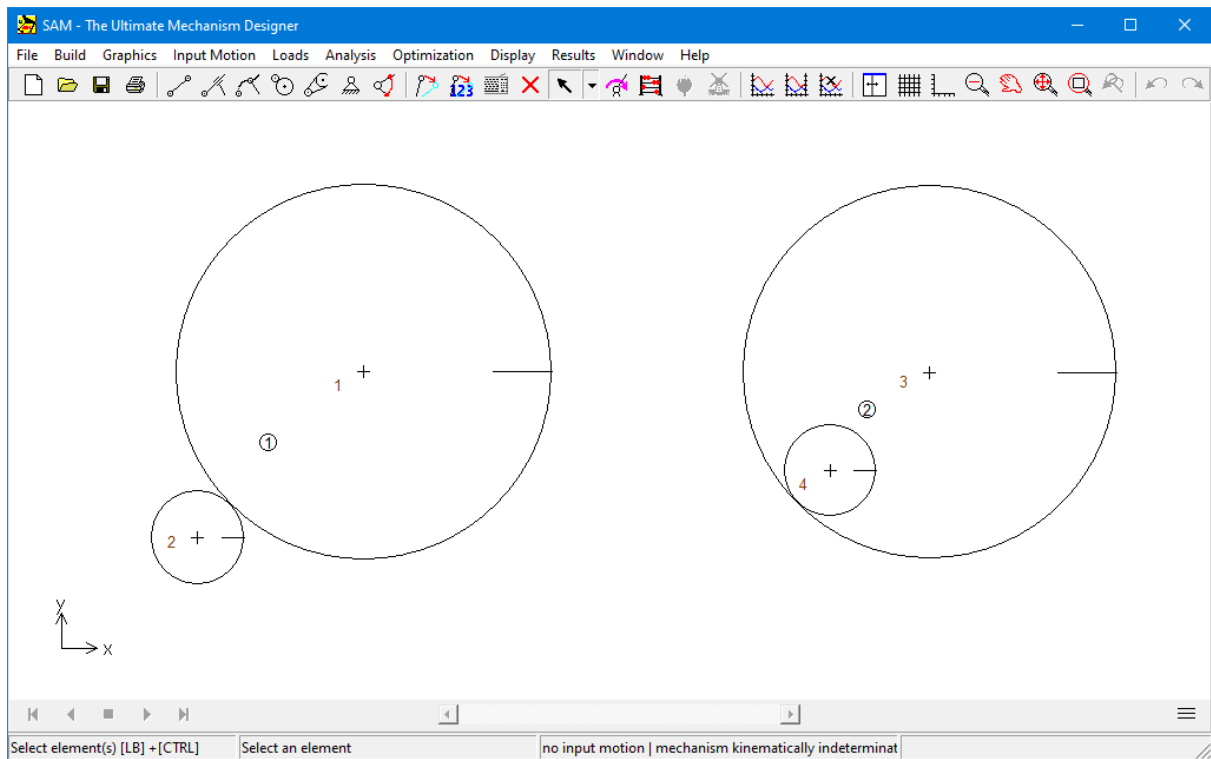
The **length** and its derivatives are measured along the curve starting from the first point of the curve to the second node. The orientation and corresponding **angle** and its derivatives is symbolized by the small line at the first node which in the initial orientation of the mechanism is horizontal by default and the initial angle is equal to zero. This is illustrated again in the following example. In the initial position, the angle is equal to 0 [degree] while the length along it corresponds to 0.501 [m].



Example to clarify the definitions of length and angle.

11.4 Gear

The gear element is used to describe various gear configurations, ranging from standard gear configurations to gears with inner toothing and even planetary gear trains. It is important to note, that the distance between the nodes of a gear element needs to be fixed, either by constraining the nodes or - in case of a planetary gear train - by defining a beam element between these nodes.



Gear element

The gear element is defined by the position of its end nodes and the transmission ratio. Based on the distance between the nodes and the transmission ratio, the radii of both gears is calculated. The properties of the gear element, along with the element and node identification can be viewed and edited in the property dialog. The following data can be edited :

- transmission ratio (not the absolute value of the two entries is important, but only the ratio of these values).
- mass of both gear wheels
- mass moment of inertia of both gear wheels
- pressure angle

Element Properties (Gear 1) ✕

Properties | Graph selection | Display | Optimization

Element Nr	1
Node 1	1
Node 2	2
Nodal Distance	1.765

	Node 1	Node 2
Radius	1.412	0.353
Mass	0.000	0.000
Inertia at COG	0.000	0.000

☐ Internal Tooth

Pressure angle [deg]

Properties of gear element

The following analysis results can be selected:

- absolute and relative angle, angular velocity, angular acceleration
- gear force that is transmitted between the two gears (the two possible force transmission directions are indicated by the force FN1 and FN2, one of which is NEGATIVE (= pressure) and the other one is equal to zero).
- torque and power (if corresponding angle is driven) at each of the two nodes.

Element Properties (Gear 1)

Properties | Graph selection | Display | Optimization

Plot X-axis

<input type="checkbox"/> Normal force (left)	Fn1
<input type="checkbox"/> Normal force (right)	Fn2

Related to Node: 1

<input type="checkbox"/> Angle	A
<input type="checkbox"/> Relative angle	Ar
<input type="checkbox"/> Angular velocity	Av
<input type="checkbox"/> Angular acceleration	Aa
<input type="checkbox"/> Torque	T1
<input type="checkbox"/> Power	P1
<input type="checkbox"/> Bearing Force X	Fb1x
<input type="checkbox"/> Bearing Force Y	Fb1y
<input type="checkbox"/> Bearing Force Abs	Fb1

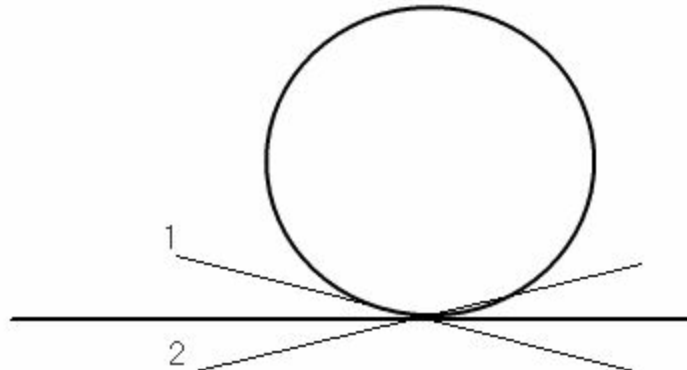
Related to Node: 2

<input type="checkbox"/> Angle	A
<input type="checkbox"/> Relative angle	Ar
<input type="checkbox"/> Angular velocity	Av
<input type="checkbox"/> Angular acceleration	Aa
<input type="checkbox"/> Torque	T2
<input type="checkbox"/> Power	P2
<input type="checkbox"/> Bearing Force X	Fb2x
<input type="checkbox"/> Bearing Force Y	Fb2y
<input type="checkbox"/> Bearing Force Abs	Fb2

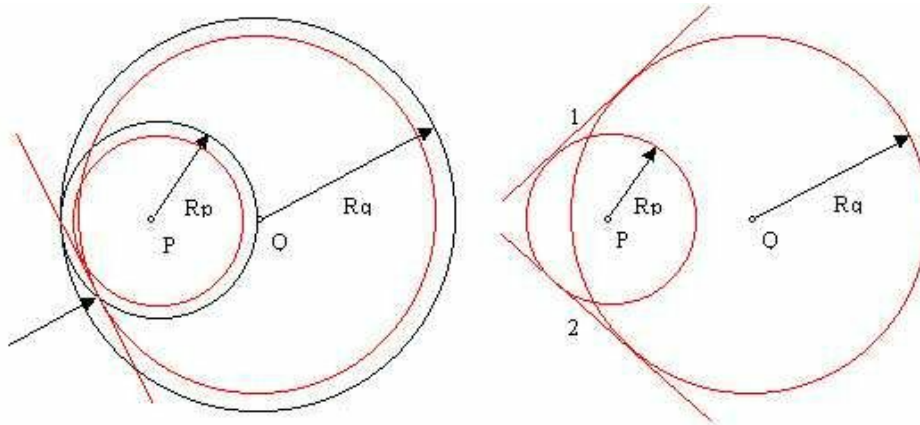
OK Cancel

Calculable properties of gear element

In any gear transmission one can distinguish two possible force transmission directions. In the absence of pre-tension one of these two forces is equal to zero, whereas the other one is **NEGATIVE** (= pressure force). In order to understand the results of the force analysis (internal element forces Fn1 and Fn2) it is essential to have a clear definition of these two contactlines. Contactline 1 of a gear element which is defined from node P to node Q always comes into contact with wheel Q on the left-hand side of the line PQ (see example).



Definition of element forces F_{n1} and F_{n2} in a standard gear transmission



Definition of element forces F_{n1} and F_{n2} in a gear transmission with internal tooth

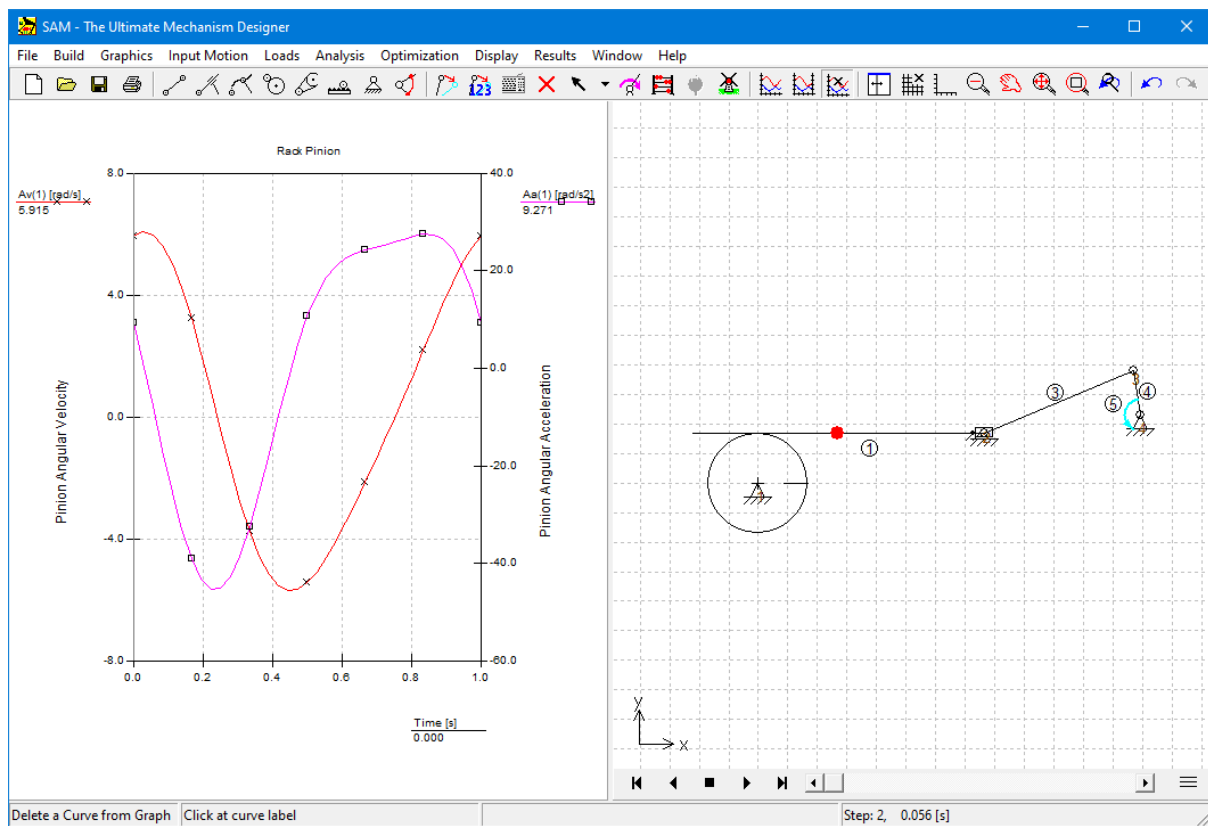
11.5 Rack Pinion

The rack pinion element is used to describe even the most complex rack pinion configuration. Building a rack pinion is a four-step process:

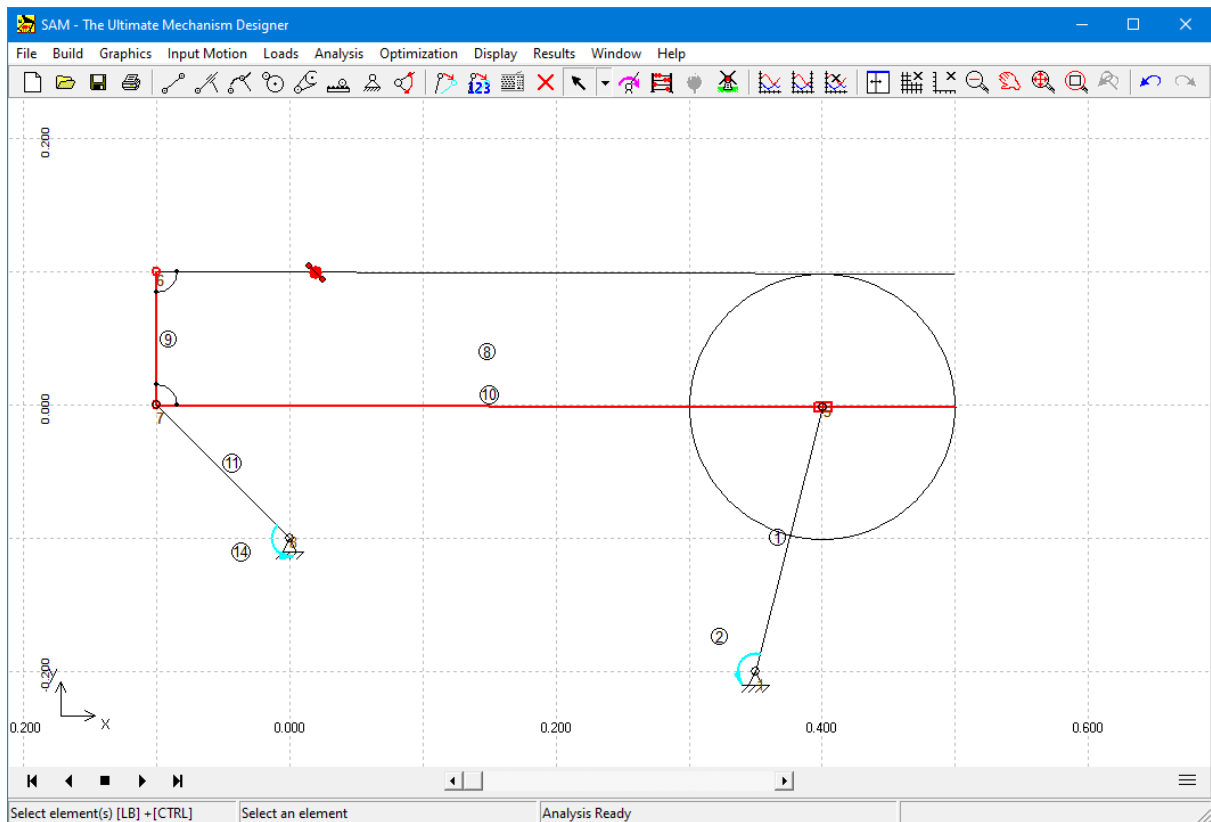
1. click on the center of the pinion gear
2. define the gear radius
3. click on the start of the rack
4. choose between the two orientation options of the rack and define the length of the rack.

All properties can be modified afterwards using the element properties dialog.

Similar to the gear pair, that only defines the rotational relation between the involved gears but not the distance between the gears, the rack pinion element imposes only one constraint. The distance from rack to the center of the pinion is not fixed and needs to be constrained by external fixation(s) or by additional elements, as indicated in the examples below.



Example of a simple mechanism with a rack pinion element (note: angle of the rack is fixed in node 2)



More complex mechanism in which beam 9 and slider (both indicated in red) constrain the distance between pinion and rack.

The rack-pinion element is defined by the position of its end nodes and the radius of the pinion. The properties of the rack-pinion element, along with the element and node identification can be viewed and edited in the property dialog. The following data can be edited :

- length of the rack (only for visual display)
- radius of the pinion
- mass and mass moment of inertia of pinion
- mass (+location) and moment of inertia around center of mass of the rack
- pressure angle
- choice of tangent

Element Properties (Rack-Pinion 8) ✕

Properties | Graph selection | Display | Optimization

Element Nr	8	
Node 1	5	
Node 2	6	
Length	0.600	[m]
Angle	-0.003	[rad]

Length	0.600	[m]
Radius	0.100	[m]
Pinion mass	0.000	[kg]
Pinion mass moment of inertia	0.000	[kgm ²]
Rack mass	1.000	[kg]
Rack mass moment of inertia	0.500	[kgm ²]
Relative distance to node of rack [0..1]	0.200	[-]
Pressure angle	0.000	[rad]

☐ Opposite tangent

Properties of rack-pinion element

The following analysis results can be selected:

- absolute and relative angle, angular velocity, angular acceleration
- force that is transmitted between the rack and the pinion (the two possible force transmission directions are indicated by the force Fn1 and Fn2, one of which is NEGATIVE (= pressure) and the other one is equal to zero).
- torque and power (if corresponding angle is driven) at each of the two nodes.

Element Properties (Rack-Pinion 8) X

Properties | Graph selection | Display | Optimization

Plot X-axis

☐ Normal force (left) Fn1

☐ Normal force (right) Fn2

Pinion

☐ Angle A

☐ Relative angle Ar

☒ Angular velocity Av

☒ Angular acceleration Aa

☐ Torque T1

☐ Power P1

☐ Bearing Force X Fb1x

☐ Bearing Force Y Fb1y

☐ Bearing Force Abs Fb1

Rack

☐ Angle A

☐ Relative angle Ar

☒ Angular velocity Av

☒ Angular acceleration Aa

☐ Torque T2

☐ Power P2

☐ Bearing Force X Fb2x

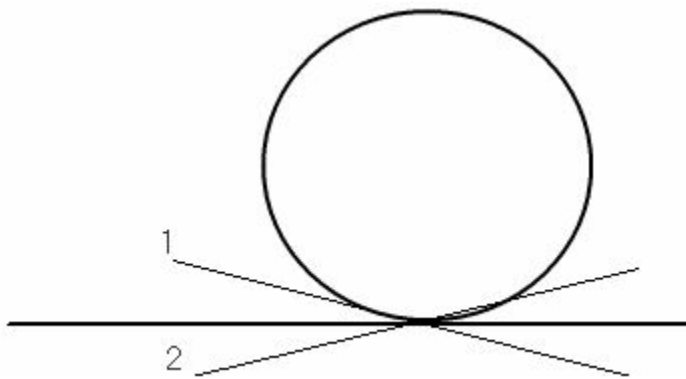
☐ Bearing Force Y Fb2y

☐ Bearing Force Abs Fb2

OK Cancel

Calculable properties of rack-pinion element

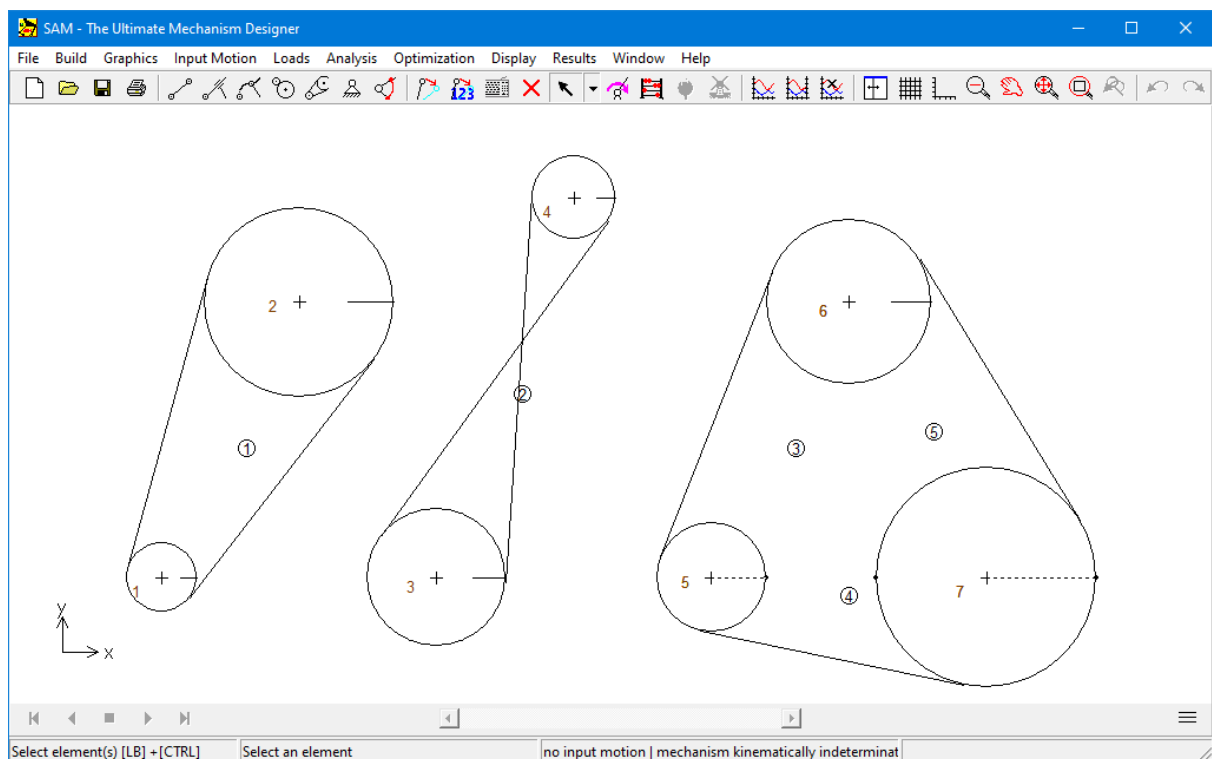
In any gear pair or rack-pinion one can distinguish two possible force transmission directions. In the absence of pre-tension one of these two forces is equal to zero, whereas the other one is **NEGATIVE** (= pressure force). In order to understand the results of the force analysis (internal element forces Fn1 and Fn2) it is essential to have a clear definition of these two contact lines.



Definition of element forces F_{n1} and F_{n2} in a rack-pinion

11.6 Belt

The belt element is used to describe various sort of belt configurations, ranging from standard belt to crossed belts or a belt that passes several wheels. It is important to note, that the distance between the nodes of a belt element needs to be fixed, either by constraining the nodes or - in case of a non-stationary belt - by defining a beam element between these nodes.



Belt element

The belt element is defined by the position of its end nodes, the radii of the wheels and a flag that indicates whether the belt is standard or reversed. Furthermore, the user can indicate whether both parts of the belt should be drawn or only one (results in nicer graphics in case of a belt that passes along a number of wheels). The properties of the belt element, along with the element and node identification can be viewed and edited in the property dialog. At each wheel the user can define :

- radius
- mass
- mass moment of inertia

Element Properties (Belt 1)

Properties | Graph selection | Display | Optimization

Element Nr: 1
Node 1: 1
Node 2: 2
Nodal Distance: 0.559

	Node 1	Node 2
Radius	0.064	0.172
Mass	0.000	0.000
Inertia at COG	0.000	0.000

☒ Closed
☐ Reversed
☐ Opposite Belt

OK Cancel

Properties of belt element

The following items can be selected as result of the analysis process :

- absolute and relative angle, angular velocity, angular acceleration

- belt force that is transmitted between the two wheels (the two possible force transmission directions are indicated by the force Fn1 and Fn2, one of which is POSITIVE (= tension) and the other one is equal to zero).
- torque and power (if corresponding angle is driven) at each of the two nodes.

Element Properties (Belt 1)

Properties | **Graph selection** | Display | Optimization

Plot X-axis

☐ Normal force (left) Fn1

☐ Normal force (right) Fn2

Related to Node: 1

☐ Angle A

☐ Relative angle Ar

☐ Angular velocity Av

☐ Angular acceleration Aa

☐ Torque T1

☐ Power P1

☐ Bearing Force X Fb1x

☐ Bearing Force Y Fb1y

☐ Bearing Force Abs Fb1

Related to Node: 2

☐ Angle A

☐ Relative angle Ar

☐ Angular velocity Av

☐ Angular acceleration Aa

☐ Torque T2

☐ Power P2

☐ Bearing Force X Fb2x

☐ Bearing Force Y Fb2y

☐ Bearing Force Abs Fb2

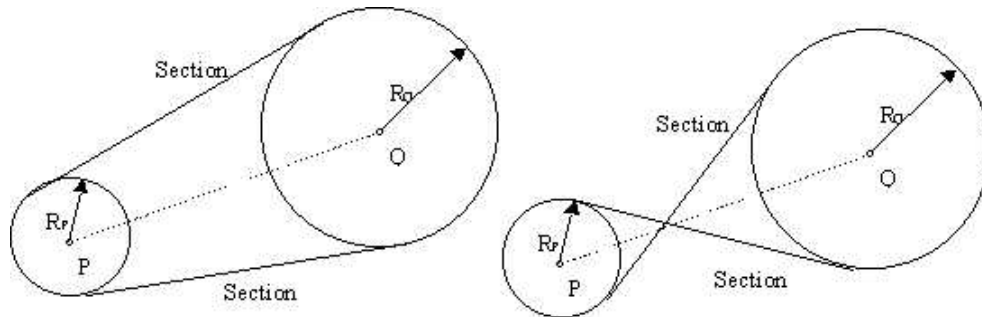
OK Cancel

Calculable properties of belt element

In any closed belt transmission one can distinguish two sections of the belt. It is assumed that there can only be tension forces in each section (pressure can not be present). In order to understand the results of the force analysis (internal element forces Fn1 and Fn2) it is essential to have a clear definition of these two sections. Section 1 of a belt element which is defined from node P to node Q

always comes into contact with wheel Q on the left-hand side of the line PQ (see example).

In case of an open belt drive only part 1 is present (which is then assumed to be able to transmit both positive and negative forces).



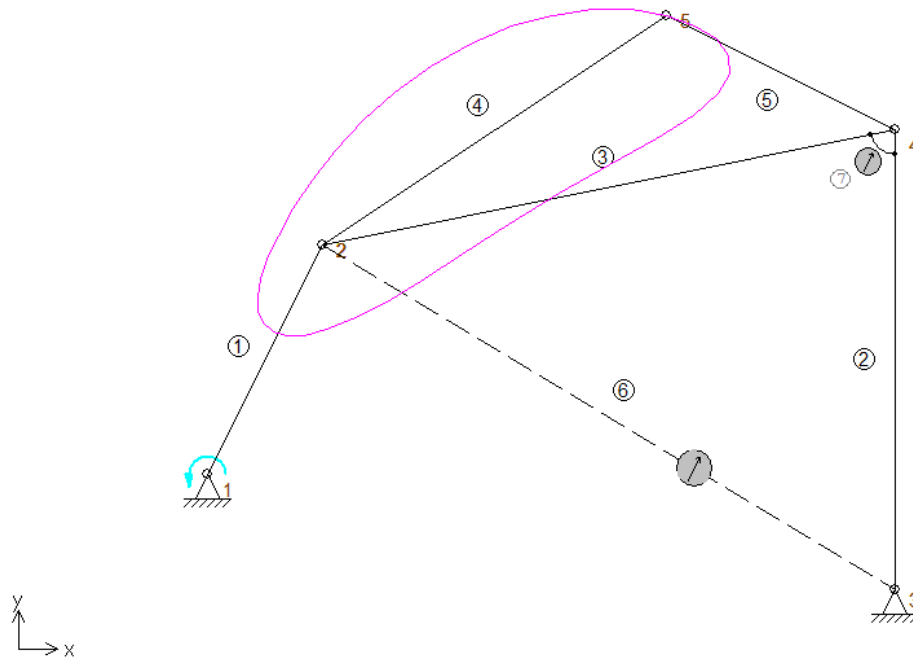
Definition of element forces F_{n1} and F_{n2} in a belt transmission

11.7 Sensor

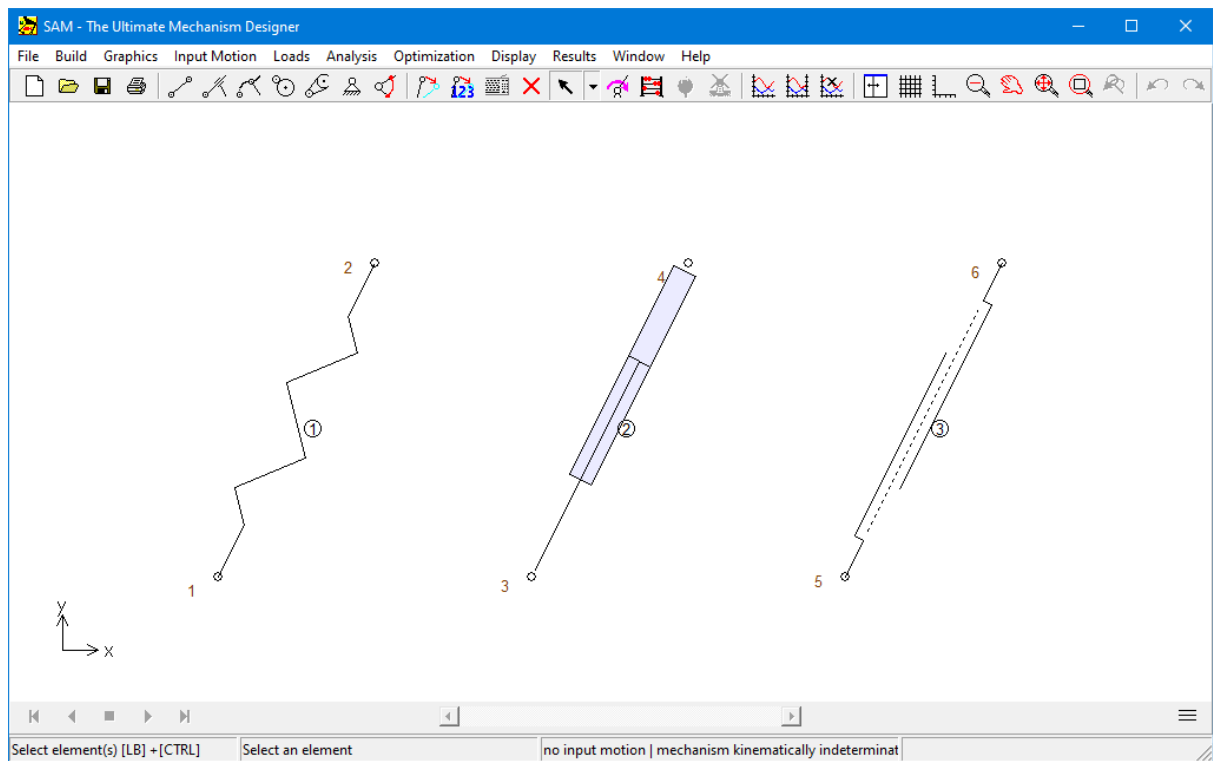
Under BUILD > EXTRA ELEMENTS you will find a (translation) sensors and a rotation sensor. Both sensors can be added to a mechanism like an ordinary element. Their sole purpose is to extract certain data from a model (like the distance between two points or the relative angle between two elements). They have no influence on the kinematics or kinetostatics of a mechanism.

The result of the sensor depends on the order in which the involved elements/nodes are picked. The positive direction of the sensor is always from the first element/node to the second element/node. The rotational sensor measures the angle between two elements at a certain node while considering both elements to start at that node (even if in the element definition it is actually the end node). The results of the rotational sensor are wrapped such that the measured angle in the initial position is in the range of -180 to +180 degrees.

In the example element 6 is a sensor that measures the distance between nodes 2 and 3, whereas the rotation sensor (element 7) measures the relative angle between beam elements 2 and 3.



11.8 Spring, Damper, Friction



Translational spring, damper and friction element

The spring, damper and friction element are closely related. Each of these elements has no influence on the motion of the mechanism and is only relevant in the kinetostatics analysis to model forces that result from springs, damper or friction. The shape of each element is defined by the position of its end nodes.

The properties of each of these elements, along with the element and node identification can be viewed and edited in the property dialog. Since spring, damper and friction have different mechanical properties, that the user has to define, each of them has a different property dialog.

The spring element has two mechanical input properties :

- stiffness
- preload in the initial position of the mechanism OR its unloaded length (the user can choose which of the two ways of defining the state of the spring in the initial position of the mechanism is preferred).

Element Properties (Spring 1)

Properties | Graph selection | Display | Optimization

Element Nr	1
Node 1	1
Node 2	2
Length	0.559 [m]
Angle	63.435 [deg]

<input checked="" type="radio"/> Stiffness	1.000	[N/m]
<input type="radio"/> Preload		[N]
<input type="radio"/> Unloaded Length	0.600	[m]

OK Cancel

Properties of spring element

The damper element possesses two input properties :

- damping coefficient
- stroke of the damper (visualization only)

Element Properties (Damper 2)

Properties | Graph selection | Display | Optimization

Element Nr	2
Node 1	3
Node 2	4
Length	0.559 [m]
Angle	63.435 [deg]

Damping	1.000 [Ns/m]
Stroke	0.373 [m]

OK Cancel

Properties of damper element

The friction element is used to model pure coulomb friction. It is defined by :

- constant friction force
- length of the stroke (visualization only)

Element Properties (Friction 3)

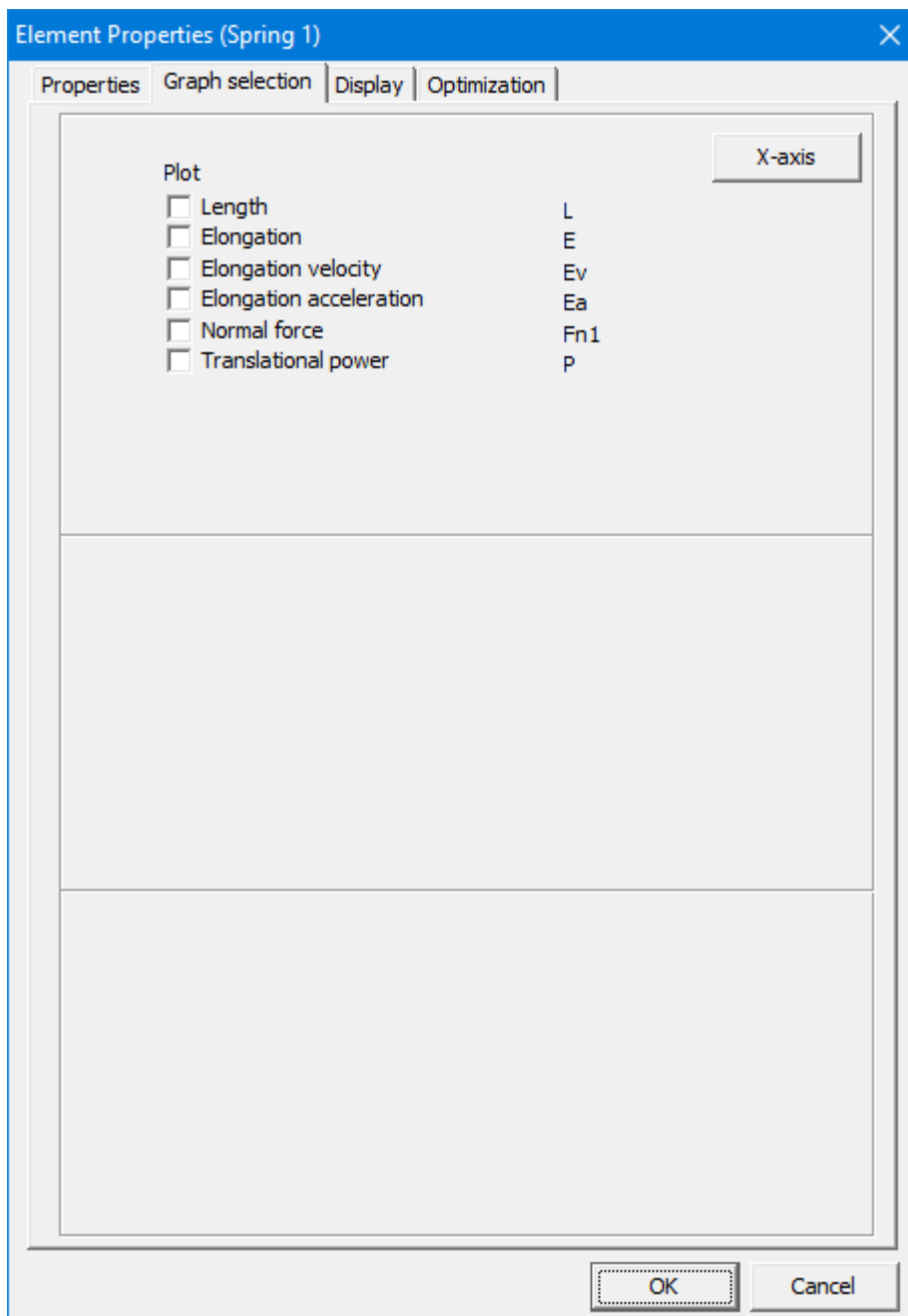
Properties	Graph selection	Display	Optimization
Element Nr	3		
Node 1	5		
Node 2	6		
Length	0.559	[m]	
Angle	63.435	[deg]	

Friction	1.000	[N]
Stroke	0.329	[m]

Properties of friction element

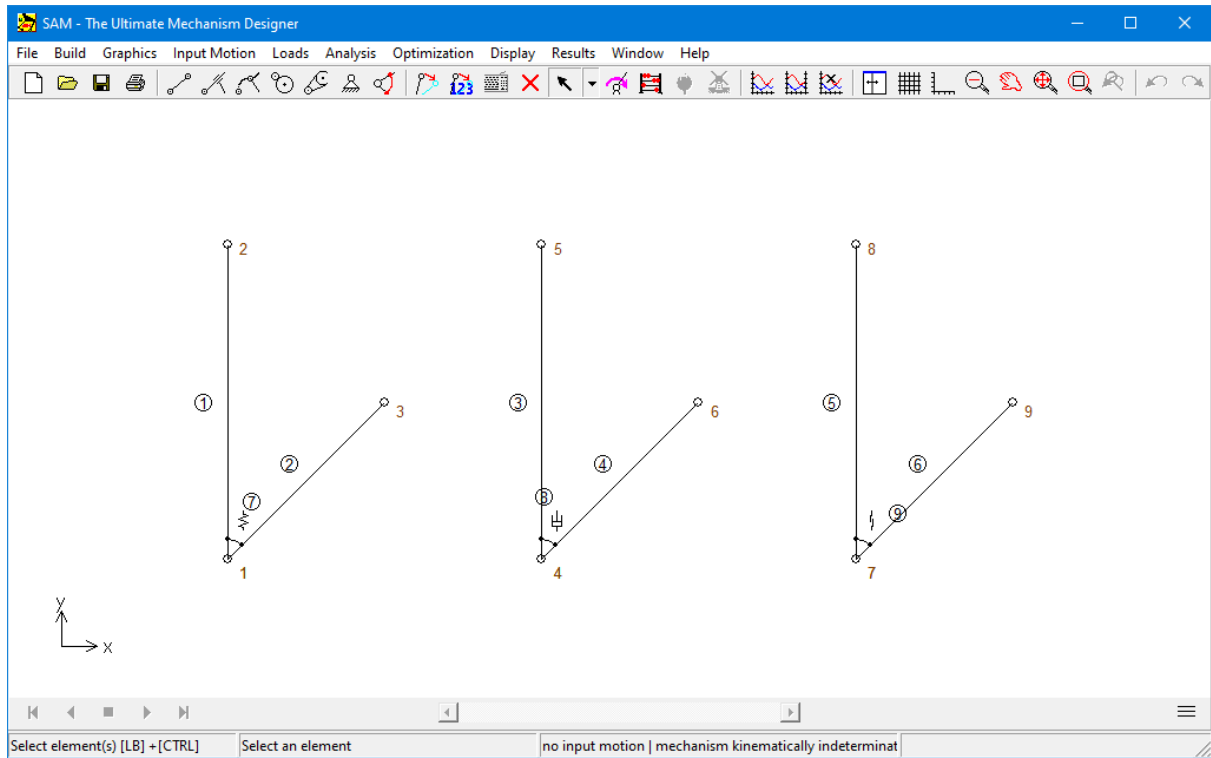
The output choices of each of these elements is identical :

- length and its derivatives
- normal force (a positive value of the force FN1 indicates a tension force) and corresponding power



Calculable properties of spring, damper and friction element are identical

11.9 Rotational Spring, Damper, Friction



Rotational spring, damper and friction element

The rotational spring, damper and friction element are closely related and can therefore be treated together. Just like the translational equivalents, these elements have no influence on the motion of the mechanism and are only relevant in the kinetostatics analysis. However, there are some important differences with respect to the translational equivalents that make a separate treatment of this group necessary.

The most important difference is the fact, that the rotational spring, damper and friction can not exists on their own. They can only be attached to already existing elements.

Creating a rotational spring, damper or friction between an existing element and the "world" is done by clicking somewhere on the workspace instead of clicking the second element.

The properties of each of these elements, along with the element and node identification can be viewed and edited in the property dialog. Since each of them has different mechanical properties, there are three different property dialogs.

The rotational spring element has two mechanical properties :

- rotational stiffness

- preload in the initial position of the mechanism.

Element Properties (Rotation Spring 7)

Properties | Graph selection | Display | Optimization

Element Nr	7
Node 1	1

Stiffness	1.000	[Nm/rad]
Preload	0.000	[Nm]

OK Cancel

Properties of rotational spring element

The rotational damper element possesses one functional property, namely its damping coefficient.

Element Properties (Rotation Damper 8)

Properties | Graph selection | Display | Optimization

Element Nr	8
Node 1	4

Damping 1.000 [Nms/rad]

OK Cancel

Properties of rotational damper element

The rotational friction element is used to model pure coulomb friction, which is defined by the value of the friction torque.

Element Properties (Rotation Friction 9)

Properties | Graph selection | Display | Optimization

Element Nr	9
Node 1	7

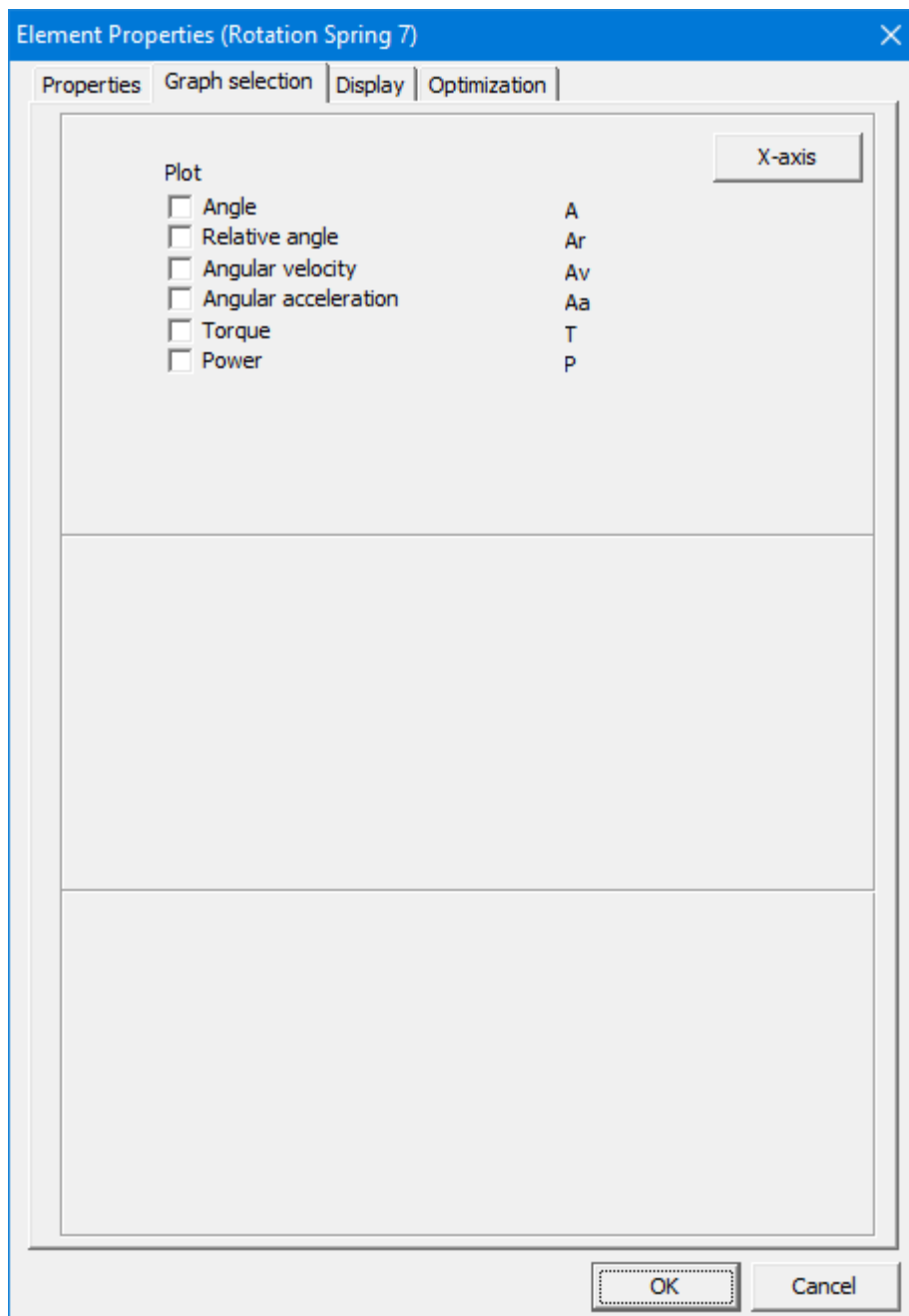
Friction 1.000 [Nm]

OK Cancel

Properties of rotational friction element

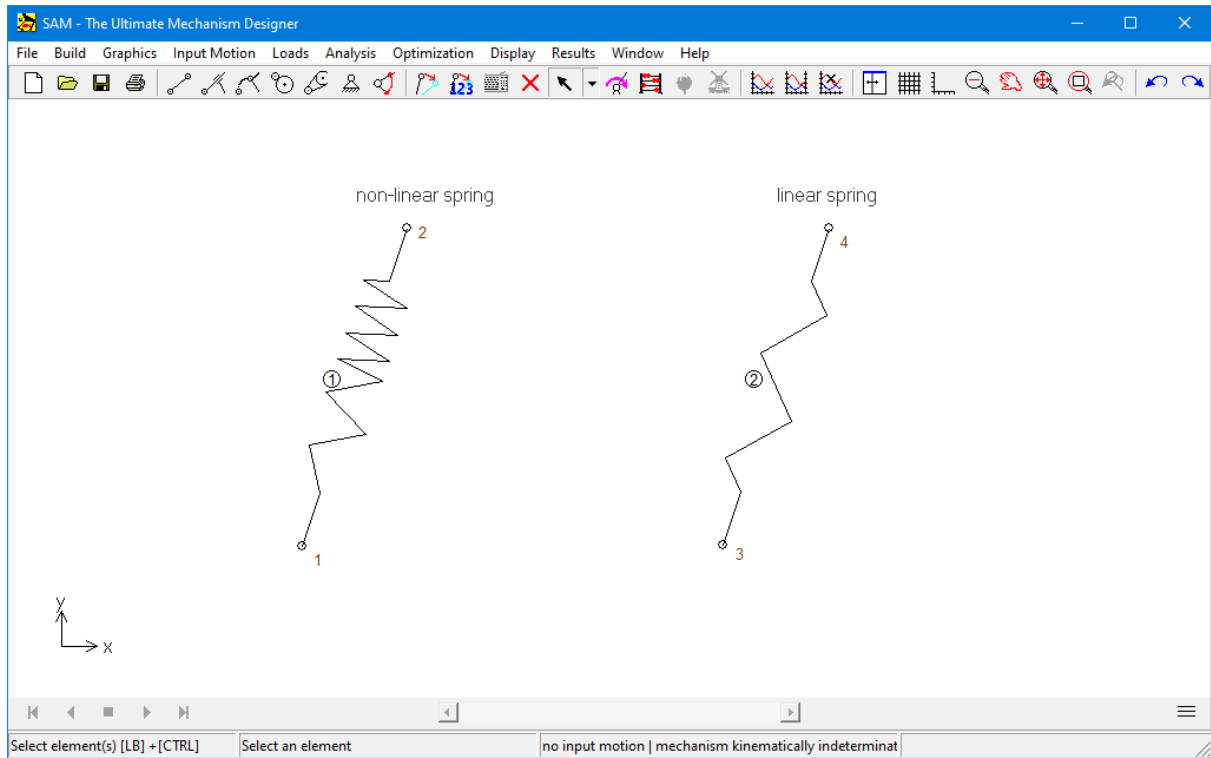
The output choices of each of these elements is identical :

- relative angle and its derivatives
- torque and corresponding power



Calculable properties of rotational spring, damper and friction element are identical

11.10 Non-linear spring



Non-linear spring (and a standard linear spring)

The non-linear spring is very similar to the linear spring in terms of functionality, however one can specify a non-linear relation between spring force and elongation of the spring. This non-linear characteristics is defined in the following way :

- Unloaded length (length of the spring in the absence of any load)
- Elongation-force table (note: elongation relative to the unloaded situation of the spring). Outside of the elongation range defined in the table, the force is assumed to be equal to zero.

Please note that (in line with all other elements) the elongation result of the non-linear spring is calculated relative to the initial position of the mechanism and NOT relative to the unloaded length of the spring. This difference is important in case of a pre-load of the spring in the initial position of the mechanism.

Element Properties (Non-linear Spring 1) ✕

Properties | Graph selection | Display | Optimization

Element Nr 1
Node 1 1
Node 2 2

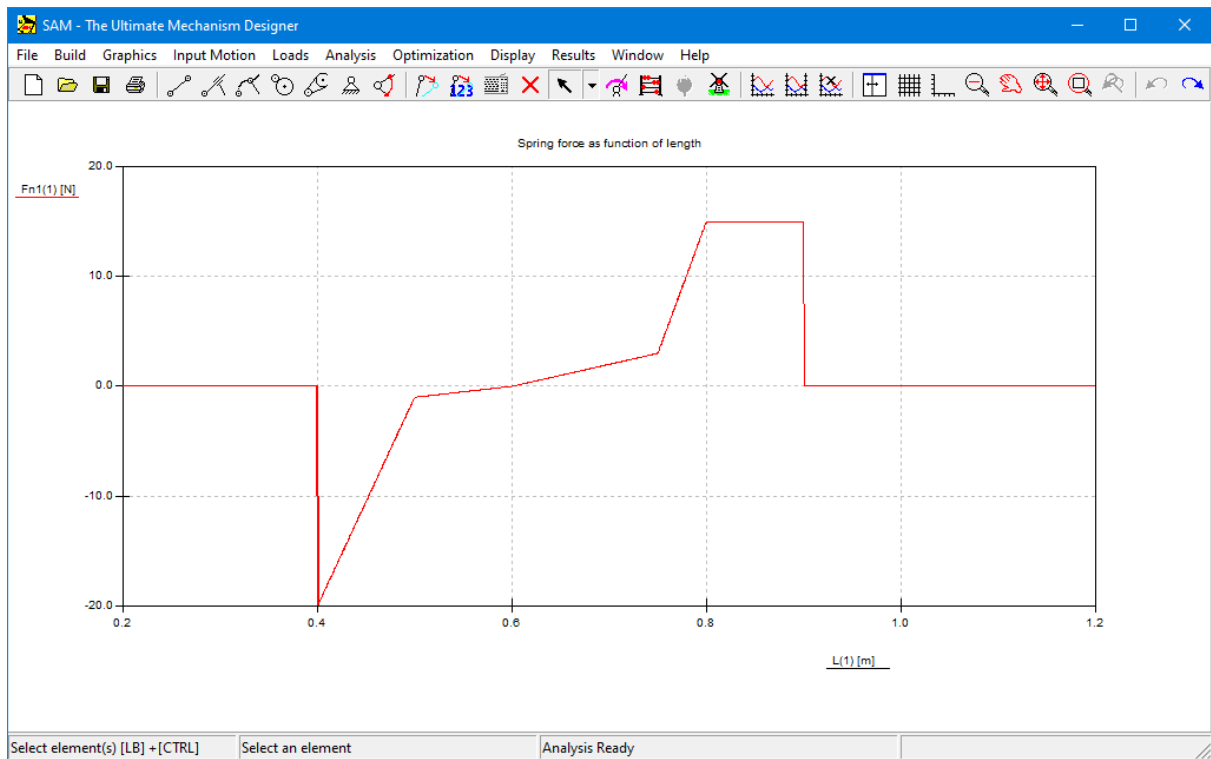
Length 0.794 [m]
Angle 71.677 [deg]

Unloaded Length 0.600 [m]

Points	Elongation [m]	Force [N]
1	0.000	0.000
2	-0.200	-20.000
3	-0.100	-1.000
4	0.150	3.000
5	0.200	15.000
6	0.300	15.000
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		

OK Cancel

Definition of the non-linear elongation-force relation



Spring force as function of length

11.11 Gas spring

A gas spring uses compressed air contained within a closed cylinder sealed by a sliding piston to pneumatically store potential energy and withstand external forces parallel to the direction of movement of the piston. The friction in the sealing means that the extension force differs from the compression force.

Element Properties (Damper 7) X

Properties | Graph Selection | Display | Optimize

Element Nr	7	
Node 1	2	
Node 2	3	
Length	0.649	[m]
Angle	5.240	[rad]

Lo	0.863	[m]
Li	0.428	[m]
Δo	0.005	[m]
Δi	0.005	[m]
F1	100.000	[N]
F2	120.000	[N]
FR	5.000	[N]

OK Cancel

Element dialogue of the new gas spring

The gas spring is defined by the following parameters :

- Lo: maximum length
- Li : minimum length
- Δo : non-useable range
- Δi : non-useable range
- F1: see plot

- F2: see plot
- FR: friction force

Please note that the gas spring properties can not be included in an optimization run. The corresponding tab does exist but shows a message that optimization is not possible.

Please note that (in line with all other elements) the elongation result of the gas spring is calculated relative to the initial position of the mechanism and the dimension of the gas spring in the initial position of the mechanism.

Chapter



12

12 Integrated Analysis Environment

12.1 Keyboard and Mouse

Normally, all user actions, such as building a mechanism, selecting a node or element, starting the analysis and so on, are performed by using the mouse. Only in a few cases it is practical to input data via the keyboard. This section gives an overview of the use of the various mouse buttons and the keyboard.

Left Mouse Button (single click)

A single click of the left mouse button is used for the selection of menu items, dialog boxes, selection of existing nodes or elements in an element or when adding a new element to a mechanism. In order to make life even easier when selecting existing nodes or elements the following facility is provided. In the neighborhood of a node the shape of the cursor changes to a cross, whereas a selected element changes its color. In the software this command is often abbreviated as LB (Left Button).

Left Mouse Button (double click)

When the cursor is located near an element, double clicking opens a dialog box which lets you enter or modify the inertia properties of an element. When adding graphical items to a group, double clicking the left mouse button can be used to finalize the operation (as an alternative to pressing the <Enter> key).

Right Mouse Button (single click)

Depending on the context, the right mouse button can be used to cancel/abort a process or it invokes context sensitive menu options. In the software this command is often abbreviated as RB (Right Button).

<Space Bar>

Every time the program prompts you to make a selection with the mouse, you can press the <space bar> and a selection dialog will appear. In this dialog you can manually enter the required selection. This is especially useful when choosing between two coinciding nodes or overlapping elements. In the process of moving or creating nodes you can use this feature to enter exact nodal coordinates with various options, such as absolute/relative, cartesian/polar or you can even "construct" intersections.

Pressing the key invokes the deletion mode. Any element or graphical item that is selected will be removed.

<ESC>

The <ESC> key can be used to cancel any activity.

<TAB>

The <TAB> key can be used to step through the list of elements while in the element selection mode.

<ENTER>

1. Finalizes an element selection that is initiated by the <TAB> key.
2. Finalizes a group modification (graphics) when adding/removing items.

<Ctrl>+C

Copies the active SAM window to the clipboard.

<Ctrl>+M

Shortcut for Input/Modify. Lets you modify the input motion definition.

<Ctrl>+P

Shortcut for File/Print... . Prior to printing you can enable printing of the graph and/or the mechanism.

<Ctrl>+mouse move

Horizontal or vertical alignment of element.

<F2>

Shortcut for Display/Animation. Start the animation.

<F3>

Shortcut for Build/Element Properties.

<F4>

Shortcut for File/Preferences

<F5>

Shortcut for Optimization/Run.

<F8>

Shortcut to return to Design Wizard

<F9>

Shortcut for Analysis.

<F10>

Shortcut for Display/Options

12.2 Toolbar

The toolbar contains a number of icons that represent short-cuts for menu items.



New Open a new mechanism project.



Open Open an existing mechanism project.



Save Save current mechanism project.



Print Print the current mechanism/graph.



Beam Enter a beam element.



Slider Enter a slider element.



Curved Slider Enter a curved slider element based on polylines or Bézier curves



Gear Enter a gear element.



Belt Enter a belt element.



Rack-Pinion Enter a rack and pinion.



Support Apply support in x and/or y direction.



Fix Rel. Angle Fix relative angle between two elements..



Input Define input (translational and rotational).



Move Move a node.



Exact coordinates Define exact nodal coordinates.



Keyboard entry Make general entry or selection via keyboard.



Delete Start delete action



Select Depending on the chosen modus (drop down menu that is part of the icon) one can either select parts of a mechanism or graphics.



Analysis Start the analysis of a mechanism.



Optimization Start Optimization.



Animation Start Animation.



Graph/Select Select items to be included in graph.



Graph/Toggle Toggle items between y-left and y-right in graph.



Graph/Deselect Remove item from graph.



Layout Modus Toggle between 3 layout modi: 1.mechanism and graph, 2. mechanism only, 3. graph only.



Grid on/off Toggles the background grid on/off.



Ruler on/off Toggles the ruler on/off.



Zoom out Zoom out.



Zoom max Zoom such that mechanism just fits into window.



Zoom Zoom (user can define zoom window).



Pan Move the viewing window



Zoom previous Return to previous zoom settings.



Undo Reverses the last action.



Redo Redo the last (undone) action.

12.3 Menu Commands

12.3.1 File Menu

New

Opens a new mechanism project. When you choose New, you can save changes to the project you have been working on.

Wizard

Lets you choose from a number of mechanism design wizards.

Open

Opens an existing mechanism project.

Save

Saves changes to the document you have been working on. When you choose Save, the project remains open so you can continue working on it.

Save As

Saves a new or existing project. You can name a new project or save an existing project under a new name. The original project file remains unchanged. When you choose Save As, the project remains open so you can continue working on it.

Import DXF

Imports a DXF-file and automatically groups graphical data (see menu Graphics).

Export DXF

Exports the current mechanism to DXF-file which can be imported in any CAD-software for adding details. Note that only the nodes and elements are exported.

Project Notes

Starts text editor in order to include project related notes.

Project Documentation

Generates a complete project documentation in ASCII-format.

Statistics

Displays some characteristics of the current project.

Preferences

Lets you select the unit system and a variety of other settings.

Preferences and project options (Display>Options or Results>Graph Options) are interlinked in case a parameter exists both as a preference and as a project option. Modifying project options does not affect the corresponding preferences field. On the other hand, changes made to the preferences (File>Preferences) are automatically copied to the corresponding options field.

Print

Prints the current mechanism/graph. Prior to printing the user can select whether the graph, the mechanism or both will be printed.

Print Setup

Selects a printer and sets printer options before printing. The available options depend on the printer you select.

Exit

Closes the project you have been working on and quits SAM.

12.3.2 Build Menu

The Build Menu contains all items that are required to create or modify elements and the commands which define the support node of an element.

Available Elements in SAM

Beam, Slider, Curved Slider, Gear, Belt

Extra Elements: Sensor, Spring, Gas Spring, Non-linear Spring, Damper, Friction (both in a rotational and translational version),

Fix Node (X or Y)

After selection of this item you are requested to click a node. Now, move the mouse around the selected node in order to choose between a x-, y- or xy-support. If the desired support condition is shown, press the left button to activate it.

Fix Angle

Fix an angle of an element relative to the world.

Fix relative Angle

Create a rigid fixation between the angles of two elements.

Unfix

Remove fixation of any kind

Move Node

This command lets you click a node and drag it to a new position.

Delete Element

After selection of this menu item, you are prompted to click the element, that you want to delete.

Transform Mechanism

Lets you transform the entire mechanism (optionally including graphical items) in a number of ways:

- Rotate
- Translate
- Scale
- Mirror Horizontal
- Mirror Vertical
- Change Initial Position (any calculated position of the mechanism can be chosen)

In case you only want to transform a certain part of a mechanism, you should first select & group the corresponding elements and then transform that group via the drop

Element Properties

If you want to specify certain element properties, such as inertia or center of gravity, select this menu item first, then click an element. Depending on the selected element type an element property dialog will be opened.

Node Properties

If you want to specify exact nodal coordinates select this menu item first, then click a node. A dialog will be opened which lets you specify exact nodal coordinates in various manners.

12.3.3 Graphics Menu

The Graphics Menu lets you manipulate graphical data that is added to the mechanism directly or via File/Import DXF.

Point

Draws a point

Line

This command allows you to draw a line (existing points/nodes can be used as starting- or end- point).

Rectangle

This command allows you to draw a rectangle (existing points/nodes can be used as starting- or end-point).

Polyline

This command allows you to draw a polyline (existing points/nodes can be used as starting- or end-point).

Bézier

This command allows you to draw a Bézier curve (existing points/nodes can be used as starting- or end- point).

Circle

Lets you draws a circle (an existing point/node can be used as centre point)

Centerline

Constructs the centreline based on two (existing) points/nodes

Intersection

Constructs all intersection points between any two graphical items (line, centreline or circle). The resulting points can be used for further processing.

Text

Text can be placed at any location (text can be rotated by creating a group that contains this text item and rotating that group).

Text Box

Creates a text box, including optional call-out.

Group

This command lets you group graphical items into one single entity, which can then be moved, rotated, deleted or attached to an element of the mechanism. Clicking an item selects it into the group, clicking it again reverses the selection. Note that all graphical items of an imported DXF-file are automatically grouped and sometimes need to be ungrouped first in order to define new groups.

Ungroup

Reverses a previous Group command. Note that all graphical items of an imported DXF-file are automatically grouped.

Attach

Attach a group of graphical items to any beam or slider element of the mechanism. Note that this option is limited to beam and slider elements. If you want to attach a graphical item to a non-beam element, first connect a beam to that element, then attach the graphical item to the beam.

Detach

Detach the group from the mechanism element. Graphical items that are not attached to an element of the mechanism function as a static background.

Transform Group

Lets you transform a group of graphics items in a number of ways:

- Rotate
- Translate
- Scale
- Mirror Horizontal
- Mirror Vertical

Delete Entity

After selection of this menu item, you are prompted to click the graphics items(s) that you want to delete. Note that the graphical items are really deleted from the SAM file and can only be recovered by re-importing the DXF-file. Only the latest deletion can be "undone".

Delete Group

After selection of this menu item, you are prompted to click the group(s) that you want to delete. Note that the graphical items are really deleted from the SAM file and can only be recovered by re-importing the DXF-file. Only the latest deletion can be "undone".

12.3.4 Input Motion Menu

The input menu lets you define various kinds of input type, such as x-displacement, y-displacement, angle, relative angle and elongation (elongation is used to model a linear actuator, such as a hydraulic or pneumatic cylinder). You can define up to 10 simultaneous input motions and for each you will be prompted to select the appropriate node or element. Next, a dialog box is opened, which lets you define the input motion by combining basic motion laws. Furthermore, an arbitrary motion profile can be imported from a motion file (ASCII-format).

In order to use SAM in a proper way it is essential to remember that all input data is relative to the initial position of the mechanism.

X-displacement

After selection of this menu item you are prompted to click a node and define the input law, that you want to apply to that node in x-direction.

Y-displacement

After selection of this menu item you are prompted to click a node and define the input law, that you want to apply to that node in y-direction.

Angle

Lets you define any angle as input motion. After selection of this menu item you are prompted to select an angle by first clicking a node and - if necessary - by choosing between elements that are attached to that node.

Relative Angle

Lets you define the relative angle between two elements. You will be prompted to select the two elements.

Elongation

Lets you model a linear actuator by specifying the elongation of a beam element. You will be prompted to select the element.

Modify

Lets you modify the input law of an existing input motion.

Delete

Lets you delete an existing input motion.

More:

[Input Dialog Box](#)

12.3.4.1 Input Dialog Box

The heart of the input dialog box is formed by a list box, which contains a sequential list of selected basic motion blocks (the graph shows the equivalent of these data). Initially, this list is empty. It can be filled in the following manner :

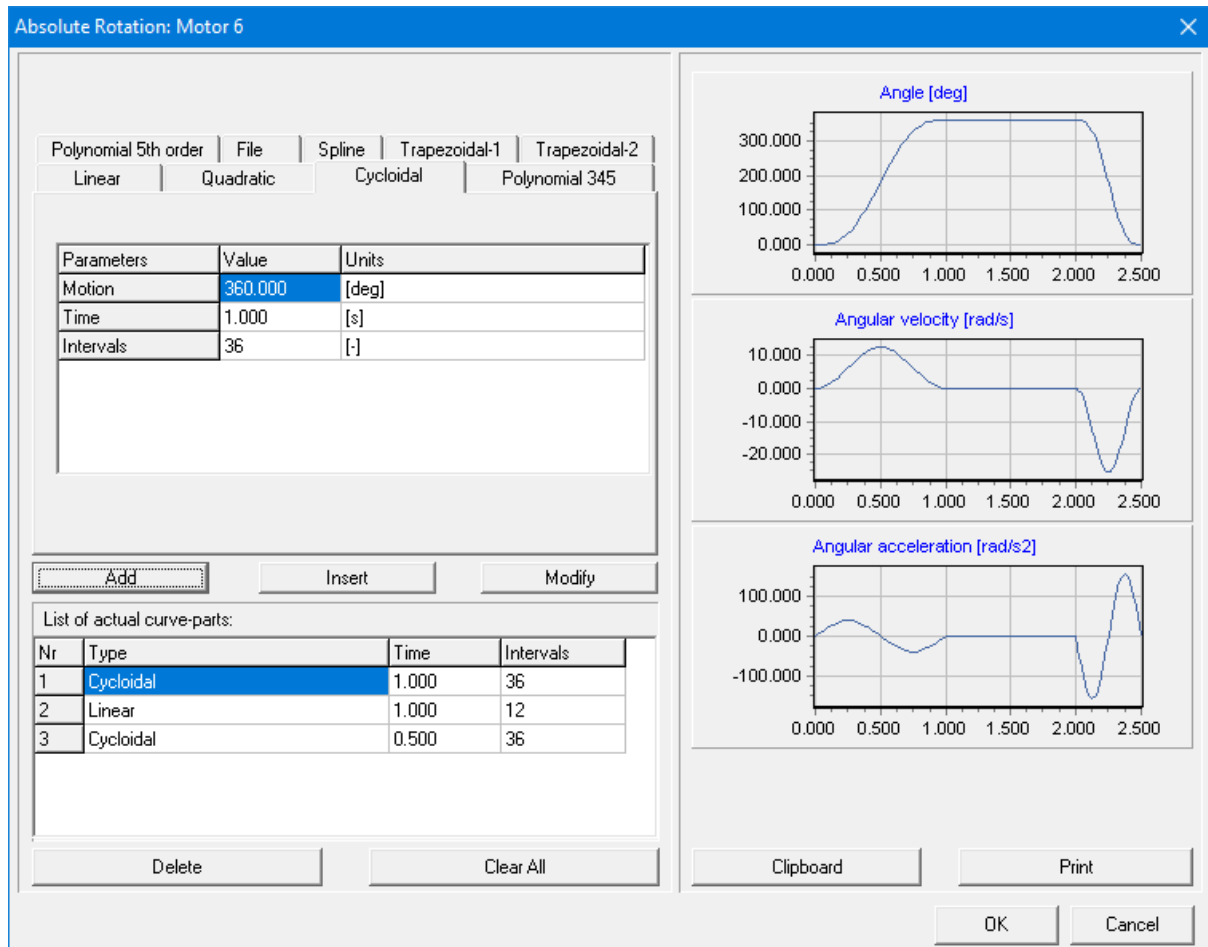
1. Activate one of the basic motion law tabs :

- Linear: motion with constant velocity
- Sinusoidal: sinusoidal motion
- Polynomial : 5th order polynomial motion
- Trapezium: motion profile with piecewise constant acceleration
- Spline : natural or clamped cubic spline
- Motion file: this tab lets you select a motion file

2. Enter the necessary input data

3. Add motion block to list by clicking the "Add" button

By repeating this process, it is very easy to define all sort of input motions by combining basic motion laws. If you want to delete an item from the list, simply activate it by clicking it once and press the "Delete" button.



Input Dialog Box

4. Modify a motion block

Click on a motion input line in the listbox. Then edit the values and press "Modify" to accept the new values.

5. Insert an additional block in between

Select the line and simply press "Insert". This lead to a new entry above the selected row.

If you want to de-activate a line selection, simply click anywhere in the dialog box.

In case a combination of these standard input motion laws is not sufficient to describe a certain input motion you can use a motion file (see chapter Modelling Considerations) that allows you to specify a file, that contains a table of motion data.

12.3.5 Loads Menu

Force

Defines an external force that acts on a node of the mechanism. You will be prompted to click a node and specify the force (value & direction). Forces can be a function of time or any kinematic variable.

Torque

Defines an external torque that acts on an element of the mechanism. You will be prompted to select an element and node and to specify the torque. A torque can be a function of time or any kinematic variable.

Mass

Adds mass to a node. You will be prompted to select a node and to specify the mass. Please see the section on bearing forces to see the difference between adding mass to a node (=shaft) vs. adding mass to the end of a (beam) element.

Inertia

Adds mass moment of inertia to an element. You will be prompted to select an element and a node and to specify the magnitude.

Gravity

Opens a dialog box and allows you to define magnitude and direction of the gravity vector.

Modify

Modifies an existing load item. You will be prompted to select the load, that you want to change.

Delete

Deletes an existing load item. You will be prompted to select the load item, that you want to delete.

[More:](#)

[Force/Torque Dialog Box](#)

12.3.5.1 Force/Torque Dialog Box

Extended force/torque definition

Force value and direction (or torque value) can be entered in a table of any size in which the argument can be time or any kinematics property of a node or an element. This feature can be used to define forces in a local moving coordinate system. It can also be used to model for example wind forces on a sports car spoiler that are a function of the vertical position of the spoiler.

In the dialogue below a force on node 5 is defined (see title of dialogue). It has a constant value of 500N and its angle is always identical to the angle/direction of element 2.

Force: Node 5

Load type

☐ As function of time

☒ As function of any property

Component: Element Nr: 2 Property: A

☐ Disabled

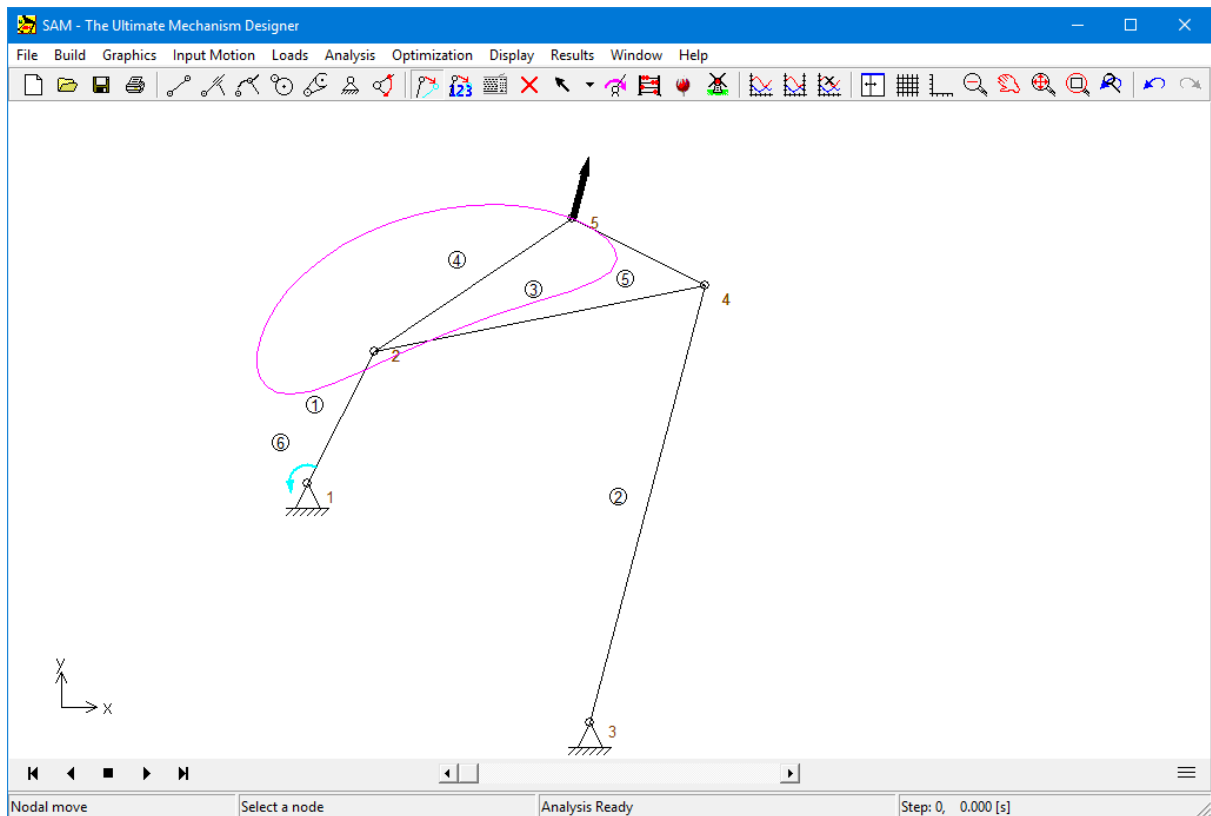
	A [rad]	Force [N]	Angle [rad]
1	-100.000	500.000	-100.000
2	100.000	500.000	100.000

Force [N]

Angle [rad]

Buttons: Add, Insert, Delete, Import ..., Delete all, Clipboard, Print, OK, Cancel

Force dialogue (example: force of 500N on node 5 with a direction identical to the angle of element 2)



Example: force of 500N on node 5 with a direction identical to the angle of element 2)

12.3.6 Analysis Menu

This menu command starts the analysis process if the automatic analysis after each modification is disabled (see: File > Preferences > Analysis).

First a dialog will be shown from which you can select the output results that you need for post processing. You can either select individual items or select entire groups of variables or even select all data. This selection procedure has been introduced in order to speed up analysis and because one is normally only interested in a limited number of variables.

You can use <F9> to start an analysis without invoking the options dialog. This is especially useful when doing identical analyses of the same mechanism with only slight parameter changes.

12.3.7 Optimization Menu

Objective

Lets you define the type of optimization target.

Parameters

Defines the parameters that can be varied during the automated search for an optimal solution.

Run

Starts the optimization.

Options

Sets the various optimization options.

12.3.8 Display Menu

Animation

Starts animation of the mechanism. It is stopped by pressing <ESC> or the right mouse button. The Build/Set_Initial_Position menu lets you perform a "manual" animation by dragging the mechanism.

Create Movie

Starts an animation which is recorded in GIF- or AVI-format.

Screen to Clipboard Ctrl+C

Sends the contents of the mechanism window to the clipboard.

Path

Lets you (de)select nodes for the display of the motion path.

Hodograph

Lets you (de)select nodes for the display of the velocity hodograph.

Fixed Centrode

Lets you (de)select linkages for the displays of the fixed centrode.

Moving Centrode

Lets you (de)select linkages for the displays of the moving centrode.

Evolute

Lets you (de)select nodes for the display of the evolute.

Options

Opens a dialog box, which lets you modify the display and various line settings.

Preferences and project options (Display>Options or Results>Graph Options) are interlinked in case a parameter exists both as a preference and as a project option. Modifying project options does not affect the corresponding preferences field. On the other hand, changes made to the preferences (File>Preferences) are automatically copied to the corresponding options field.

12.3.9 Results Menu

Select

Lets you select node or element data, that you want to export or view in a graph. You will be prompted to click a node or element and make a selection.

Clear Selection

Removes all items from the selection list, so you can start all over again.

User Defined ...

Via a sophisticated formula parser you can generate user defined results by combining standard simulation results.

X-axis Definition

Time is used as default variable for the x-axis of a plot, however any result can be used as x-axis.

Export

Lets you export the selected data to an ASCII-file. This can either be a standard data file or a special Motion File, which can be selected in the input motion definition.

Copy to Clipboard Ctrl+C

Copies the graph windows to the clipboard.

Remove Curve

Lets you de-select an item of the graph. You will be prompted to click on the label of one of the items of the graph.

Toggle Curve

Items of a graph can be scaled according to the scaling of the left or the right vertical axis of the graph. The command Toggle Curve lets you switch between the left and right side. You will be prompted to select an item by clicking on its label.

Graph Options

Lets you modify the way in which the data is displayed in a graph.

Preferences and project options (Display>Options or Results>Graph Options) are interlinked in case a parameter exists both as a preference and as a project option. Modifying project options does not affect the corresponding preferences field. On the other hand, changes made to the preferences (File>Preferences) are automatically copied to the corresponding options field.

12.3.10 Window Menu

Mechanism and Graph

SAM displays the mechanism and the graph window.

Mechanism

SAM displays the mechanism window only.

Graph

SAM displays the graph window only.

Preset Window Sizes

Switches to the preset values which are defined under File > Preferences > Windows.

12.3.11 Help Menu

Contents

Starts the SAM help system.

Search for Help on ...

Find help on a specific SAM topic.

How to Use Help...

Opens the Windows help file, that explains the standard help functionality.

Updates/Upgrades...

Lets you check whether updates (minor changes/fixes) or even upgrades (new functionality) are available.

About SAM

Displays information about SAM.

Chapter



13

13 Literature

Werff, K. v.d. (1975); "Kinematics of coplanar mechanisms by digital computation"; Fourth World Congress on the theory of machines and mechanisms, Newcastle upon Tyne, 1975

Werff, K. v.d. (1977); "Dynamic analysis of planar mechanisms with rigid links"; Comp. Meths. Appl. Mech. Eng. 11 (1977) 1-18.

Werff, K. v.d. (1977); "Kinematic and dynamic analysis of mechanisms, a finite element approach"; Dissertation (Delft Univ. Press, 1977).

Werff, K. v.d. (1979); "A finite element approach to kinematics and dynamics of mechanisms"; Fifth World Congress on the theory of machines and mechanisms, 1979

Werff, K. v.d. and H. Rankers (1979); "Getriebetyp-unabhängige Methode der Analyse der Kinematik und Dynamik der Räder-Kurbelgetriebe"; VDI-Berichte Nr. 321, 1979

Klein Breteler, A.J. (1987); "Kinematic Optimization of mechanisms, a finite element approach"; Dissertation (Delft Univ. Press); 1987

Klein Breteler, A.J. (1992); "Kinematische Getriebe-Analyse, eine iterative Methode mit Hilfe Finiter Elemente"; VDI-Fortschrittberichte Nr. 211 (Reihe 1: Konstruktionstechnik/Maschinenelement); 1992

Schwab, A.L., and Meijaard, J.P. (1999); "The belt, gear, bearing and hinge as special finite elements for kinematic and dynamic analysis of mechanisms and machines. Proceedings of the Tenth World Congress on the Theory of Machines and Mechanisms, IFToMM, June 20--24, 1999, Oulu, Finland, Oulu University Press, pp. 1375--1386.

Schwab, A.L., and Meijaard, J.P. (2000); "Dynamics of flexible multibody systems having rolling contact: application of the wheel element to the dynamics of road vehicles"; Proceedings of the 16th IAVSD Symposium, August 30-September 3, 1999, Pretoria, South Africa, Supplement to Vehicle System Dynamics 33, pp.338-349.

Chapter



14

14 Appendix

14.1 Location of Files

PROGRAM => C:\ProgramFiles(x86)\SAMXX\

- SamXX.exe
- SamXX.chm
- SamDLL32.dll
- ReadMe.txt

SETTINGS => { user } \AppData\Local\Artas\SamXX\

- SamXX.ini

The INI file is created the first time you use SAM. If you then want to move the INI file to a different directory, you can use the following syntax:

{ full path SAM Executable } \SAMXX.EXE -ini"Your Sam INI directory\SAMXX.INI"

PROJECTS => (initially) { public } \Documents\Sam\

User's choice: any writable storage location

- ***.sam
- ***.txt
- ***.dxf

BACKUP & TEMPORARY FILES => { user } \AppData\Local\Temp\SAMXX\

Use of backup & temporary files depends on user and ini-settings

- ***.dat
- ***.out
- ***.log
- SessionID__Bak00.~sam, SessionID__Bak01.~sam, ... SessionID__Bak09.~sam

EXAMPLES => { public } \Documents\Sam\Examples\

- ***.sam

DXF TEMPLATE => (initially) { all users } \Artas\SamXX\

User's choice: any writable storage location

- DxfTempl.dxf

LICENSE FILES => { all users } \Artas\SamXX\

- License.txt
- LicenseKey.txt

14.2 Status/Warning Messages

The following messages can occur during creation of a mechanism or its analysis:

Mechanism Underdetermined

Some sort of constraint is missing :

- You forgot to define a support.
- You did not connect the angle between two beams.

- You intended to connect a beam to a belt/gear but forgot to connect the angles.
- Missing input motion.
- Two nodes were intended to coincide (= one node) but are created as separate nodes.

Mechanism Overdetermined

Too many constraints such that mechanism can not move:

Somewhere you have fixed an angle or position that should be able to move.

Convergence Problem --> Analysis stopped !

Mechanism can not reach the desired position :

You defined a motion profile leading to a position that the mechanism can not reach. The results prior to that situation are valid and can be used for animation of the mechanism and postprocessing of the results.

No input defined

Missing input motion :

Most probably, you forgot to define any input motion at all (up to three separate input motions can be defined).

14.3 DXF Template File

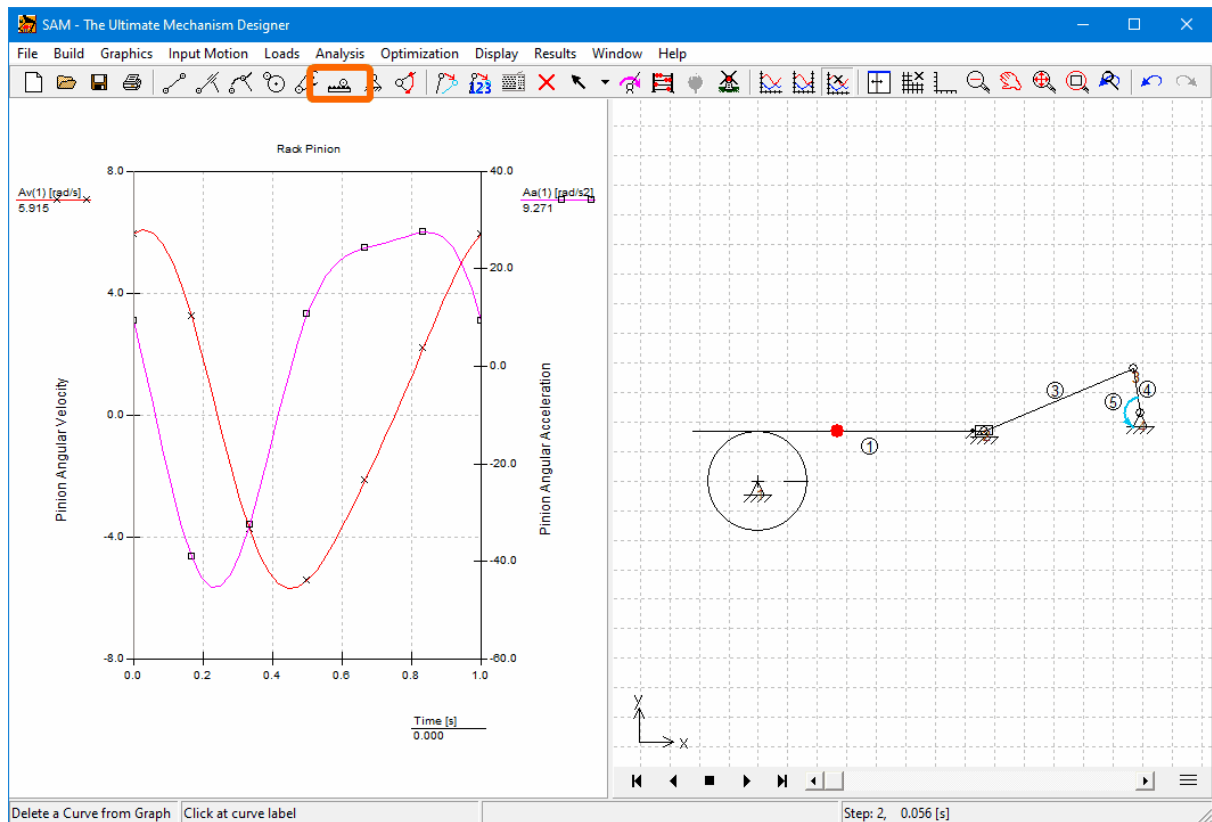
In order to achieve maximum adaptation to your CAD working method, SAM uses a DXF template file each time that a mechanism is exported in DXF-format. This template file performs all settings, such as line type, color and so on. We have provided a default template file DXFTEMPL.DXF but if you want SAM to use different setting when exporting the mechanism to a DXF file you can easily make modifications. All you need to do in your CAD-program is to save a drawing containing exactly one point, one single line, one circle and one arc, using the settings that you prefer, to a DXF file and make this file the DXF template file by changing one line in SAM70.INI. If you do not want to change the SAM50.INI you can simply save the line drawing as DXFTEMPL.DXF.

A template file is a DXF-file that contains exactly one point, one line, one circle, one arc and one text entry (these entities must be created in exactly that order !)

14.4 What was new in SAM 8.1

Dedicated rack-pinion element

In the past one had to mimic a rack-pinion by using either a belt drive or a gear pair. In the new version of SAM, a dedicated rack-pinion element including pressure angle and mass/inertia properties has been added.



Mechanism including a rack-pinion element

Automatic Synchronization of Multiple Inputs

Each input motion generates its own vector of discrete time stamps at which the motion/force of the mechanism needs to be evaluated. In the past, the vectors generated by multiple input motions needed to be identical within a certain allowable deviation threshold, otherwise a warning message was generated. In the new approach, a global vector of discrete time stamps is generated by merging all unique time stamps of each individual time vector. Time stamps that are close enough to satisfy the threshold requirement are treated as one.

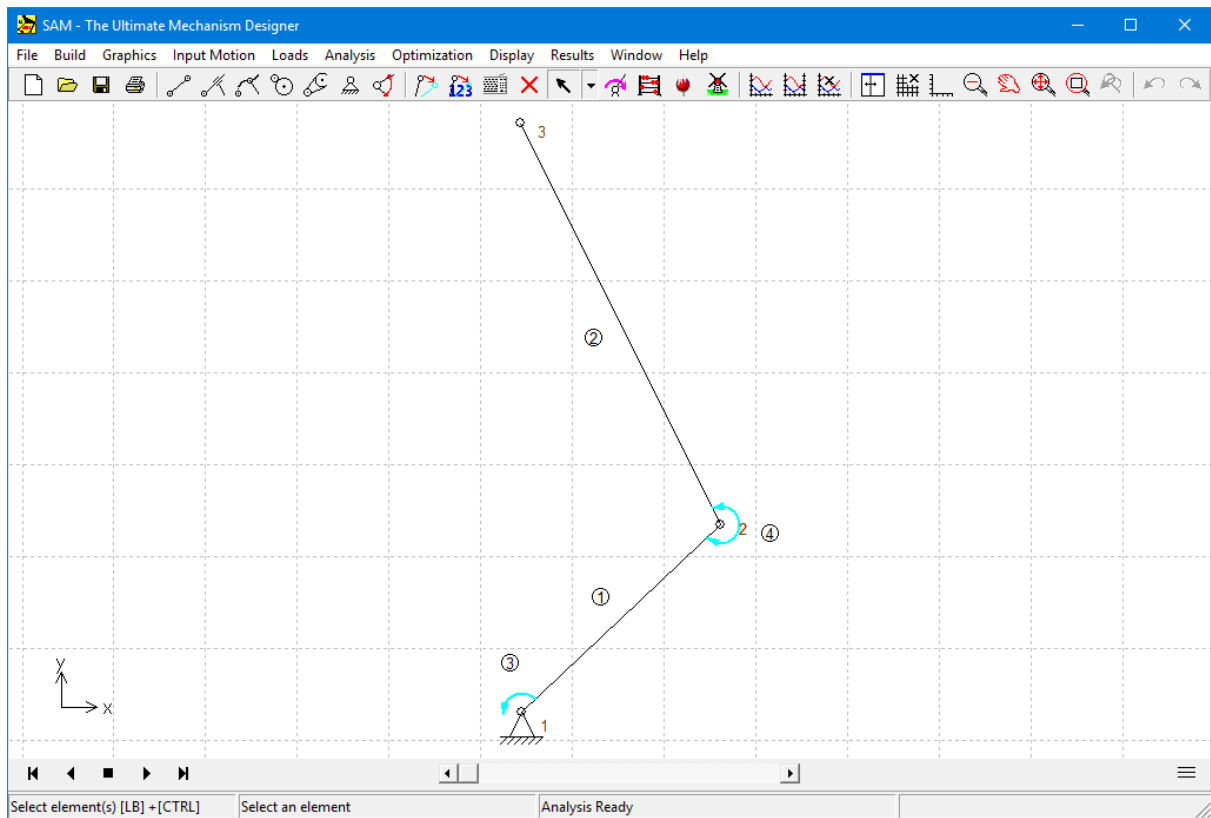
Increased analysis/optimization speed

By optimizing the analysis process a huge increase of analysis/optimization speed has been realized. This manifests itself especially during an optimization, requiring typically thousands of analysis runs, in which optimization speed increases of up to a factor 4 have been realized.

14.5 What was new in SAM 8.0

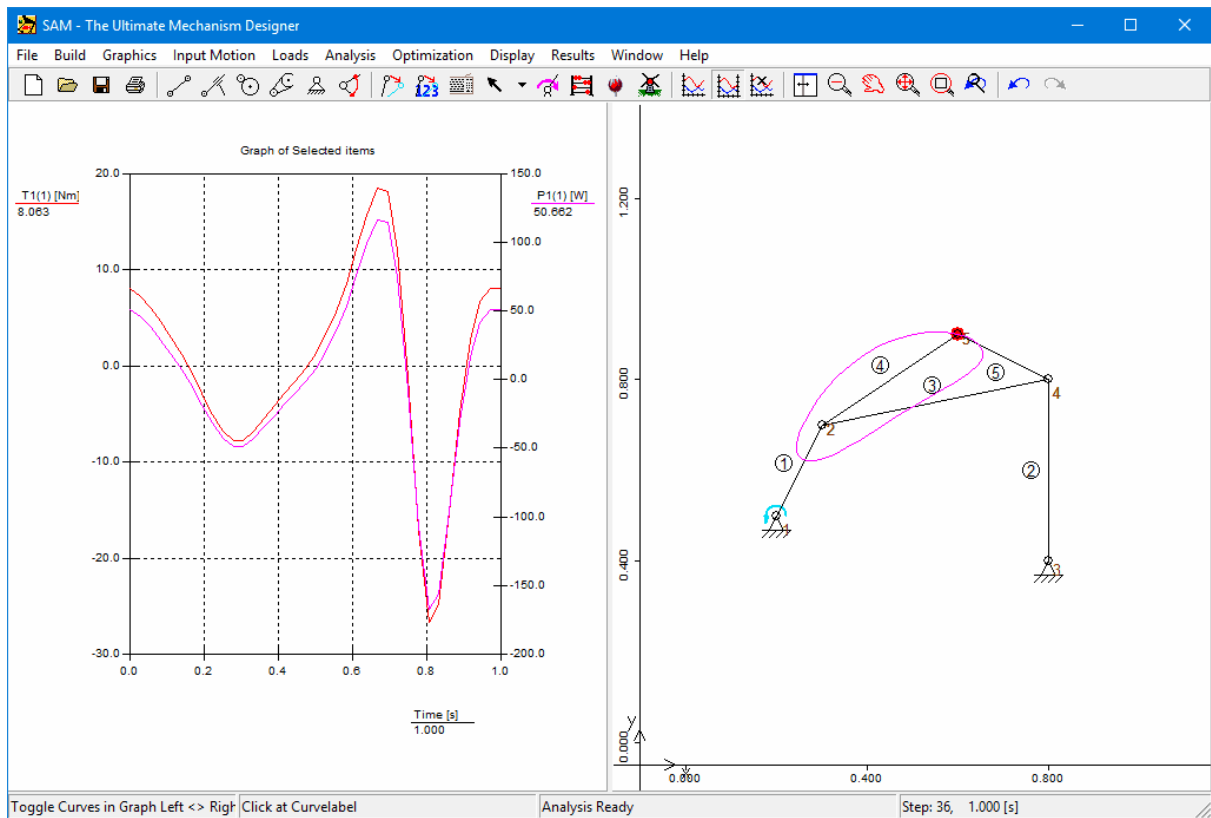
Introduction motor element

A dedicated motor element has been introduced, which will automatically be added to the system as soon as the corresponding (relative) angle input motion is created. The main benefit of this approach is the simplified calculation of the torque/power of a motor that is located in a moving joint.

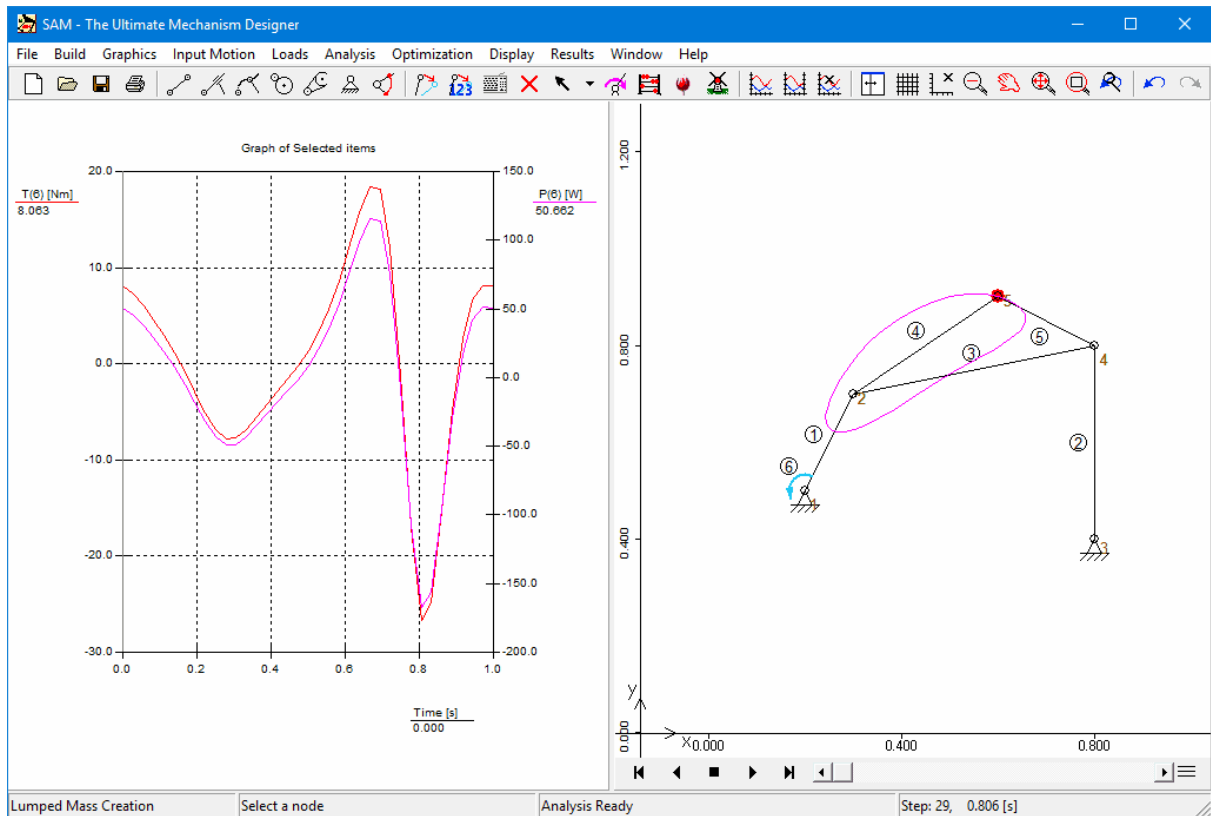


Example robot arm with two motor elements (3) and (4)

IMPORTANT NOTE: Due to the introduction of this new motor element the power required to drive a system has been removed from the crank (beam) element and is only available via the output results of the motor element. See the comparison between SAM 7.0 and SAM 8.0 in the following two screenshots



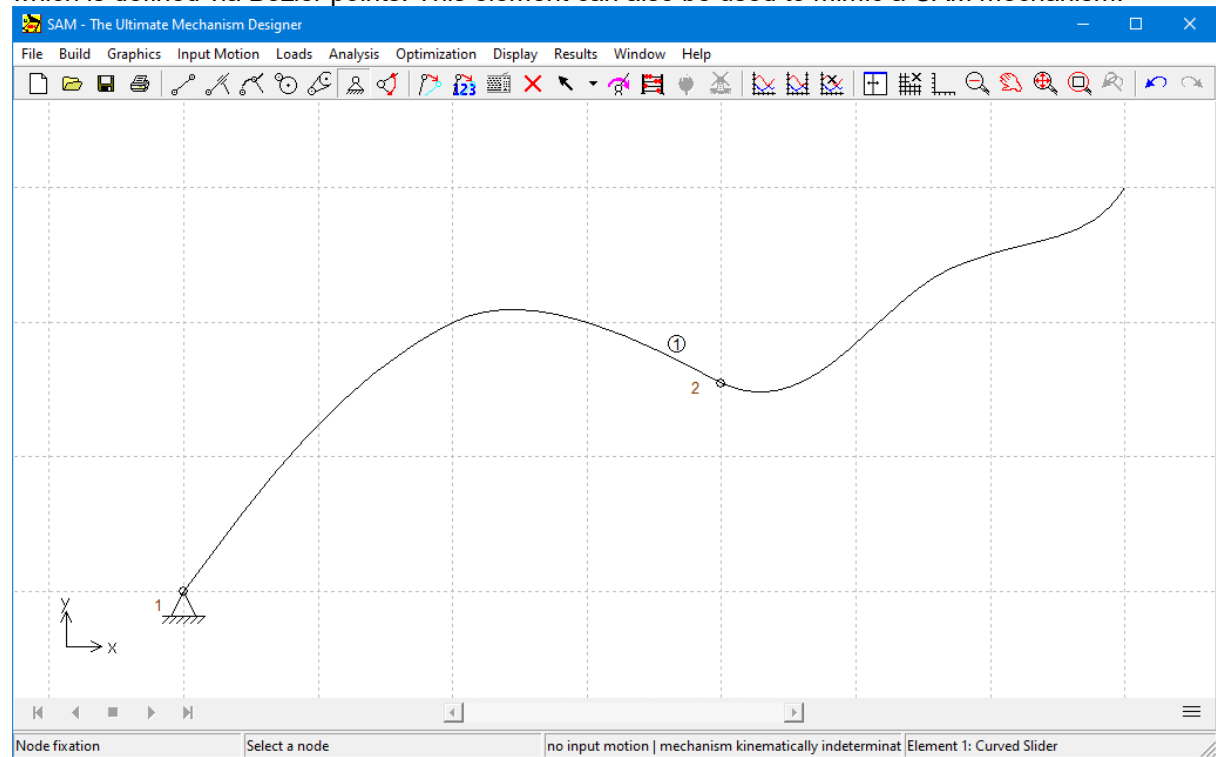
SAM 7.0 results: Torque (left) & Power (right) in element 1 at node 1



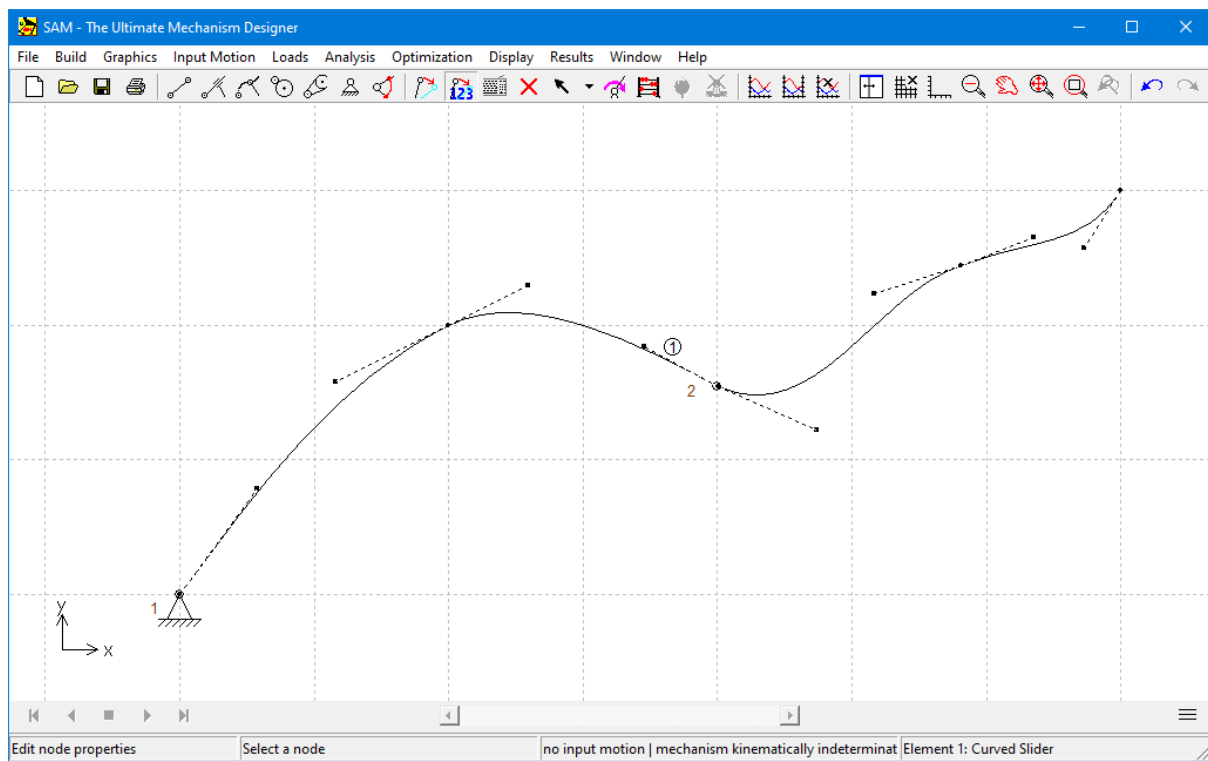
SAM 8.0 results: Torque (left) & Power (right) in element 6 (=motor).

Curved Slider

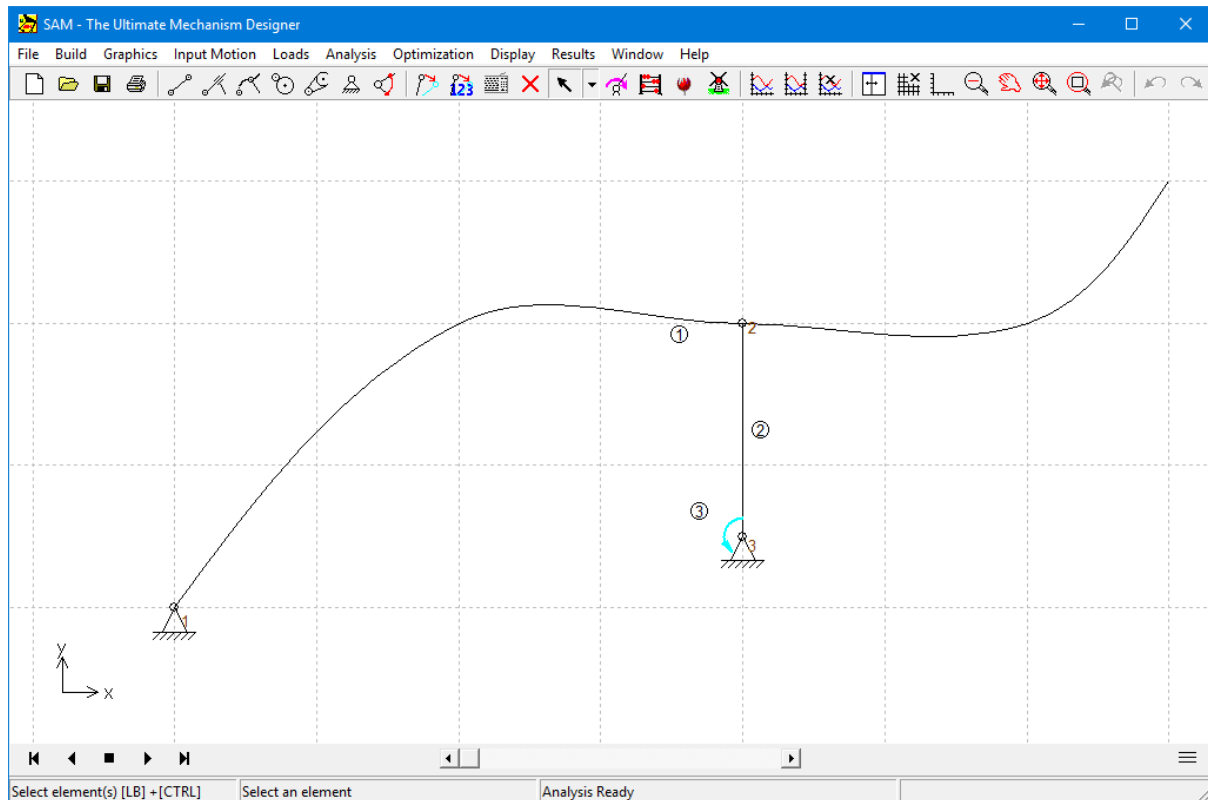
A new slider element has been introduced which consists of two nodes and an arbitrary curved shape, which is defined via Bezier points. This element can also be used to mimic a CAM mechanism.



Curved slider element



The shape of a curved slider element is defined by Bézier segments



Example of a curved slider element (1) combined with a crank and motor

Extended force/torque definition

Force value and direction (or torque value) can be entered in a table of any size in which the argument can be time or any kinematics property of a node or an element. This feature can be used to define forces in a local moving coordinate system. It can also be used to model for example wind forces on a sports car spoiler that are a function of the vertical position of the spoiler. In the dialogue below a force on node 5 is defined (see title of dialogue). It has a constant value of 500N and its angle is always identical to the angle/direction of element 2.

Force: Node 5

Load type

☐ As function of time

☒ As function of any property

Component: Element Nr: 2 Property: A

☐ Disabled

	A [rad]	Force [N]	Angle [rad]
1	-100.000	500.000	-100.000
2	100.000	500.000	100.000

Force [N]

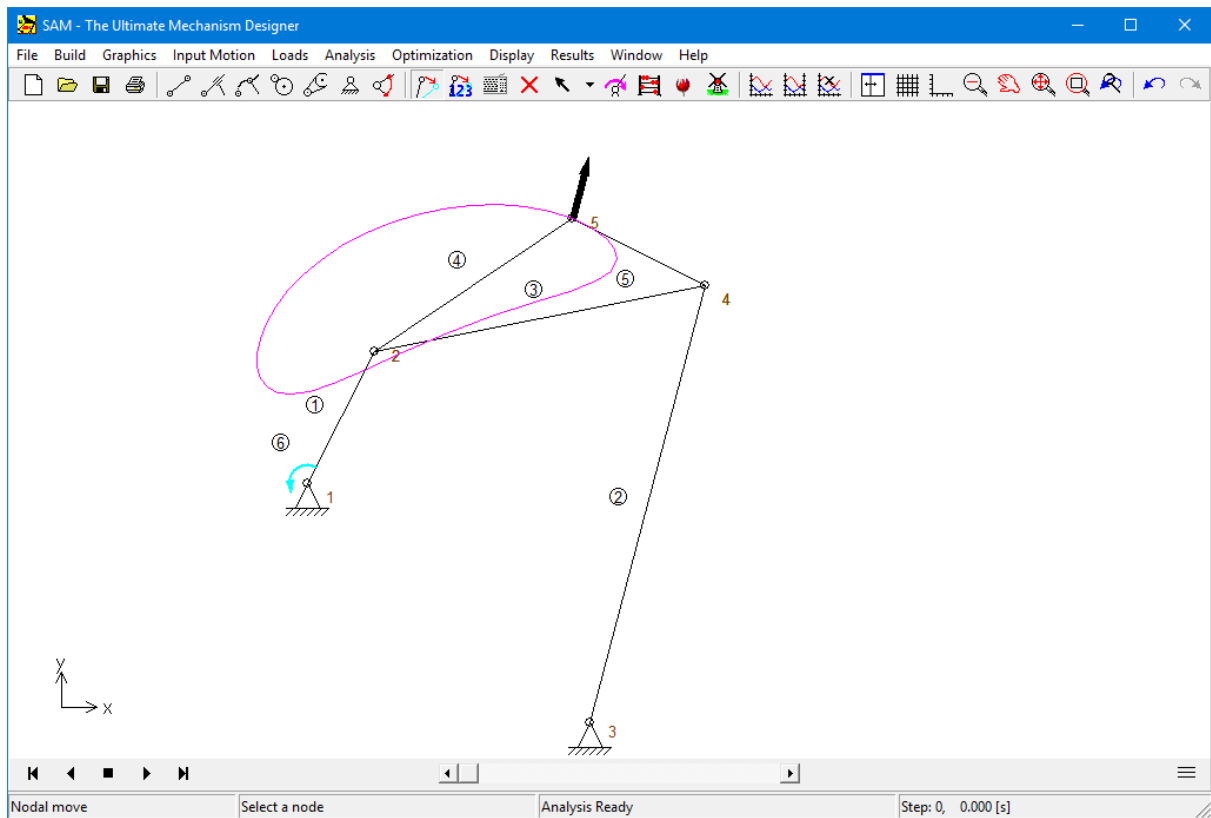
Angle [rad]

Clipboard

Print

OK Cancel

Force dialogue (example: force of 500N on node 5 with a direction identical to the angle of element 2)



Example: force of 500N on node 5 with a direction identical to the angle of element 2)

New Gas Spring element

A gas spring uses compressed air contained within a closed cylinder sealed by a sliding piston to pneumatically store potential energy and withstand external forces parallel to the direction of movement of the piston. The friction in the sealing means that the extension force differs from the compression force.

Element Properties (Damper 7) ✕

Properties | Graph Selection | Display | Optimize

Element Nr	7	
Node 1	2	
Node 2	3	
Length	0.649	[m]
Angle	5.240	[rad]

Lo	0.863	[m]
Li	0.428	[m]
Δo	0.005	[m]
Δi	0.005	[m]
F1	100.000	[N]
F2	120.000	[N]
FR	5.000	[N]

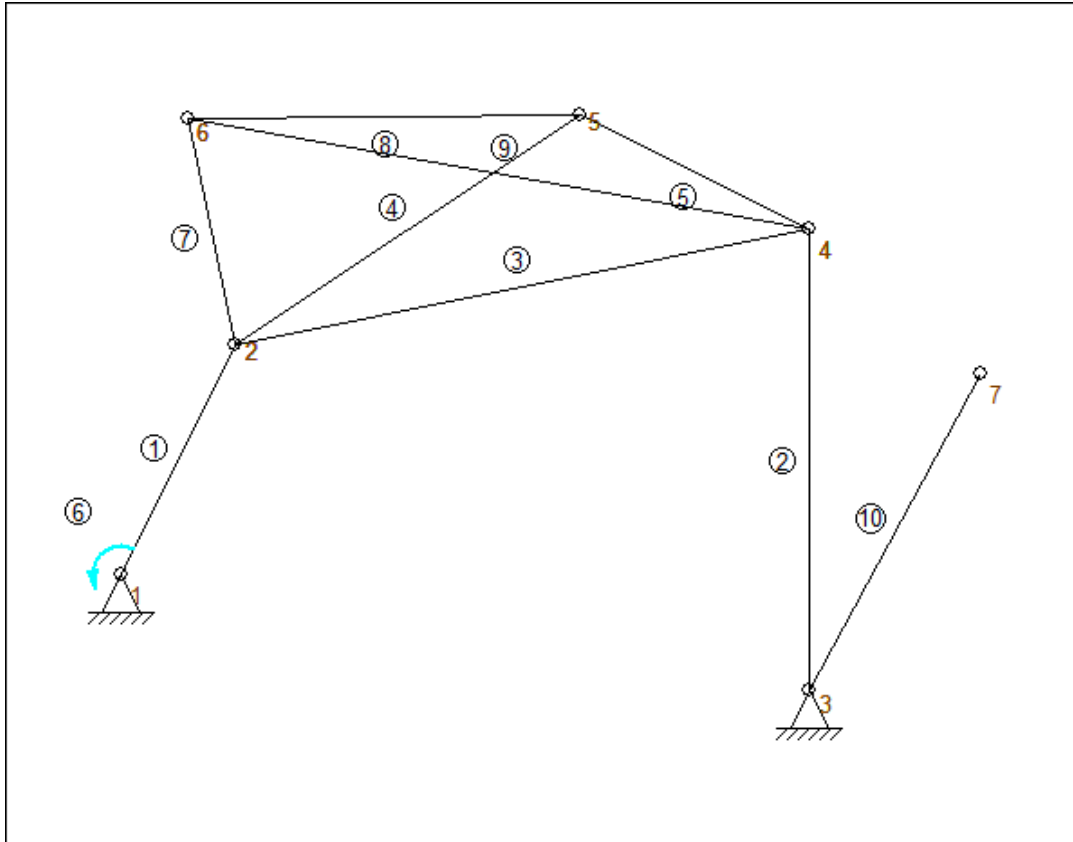
The diagram illustrates the mechanical configuration of a gas spring element. The top part shows a horizontal assembly with a force F, x applied at the left end. The total length is Lo , and the internal length is Li . Displacements Δo and Δi are indicated. The bottom part is a graph with Force (F) on the vertical axis and Displacement (x) on the horizontal axis. It shows three parallel lines representing different force states: $F1$ (bottom), $F2$ (middle), and $F2+FR$ (top). The graph also shows the relationship between $F1$ and $F2$, and between $F2$ and $F2+FR$.

OK Cancel

Element dialogue of the new gas spring

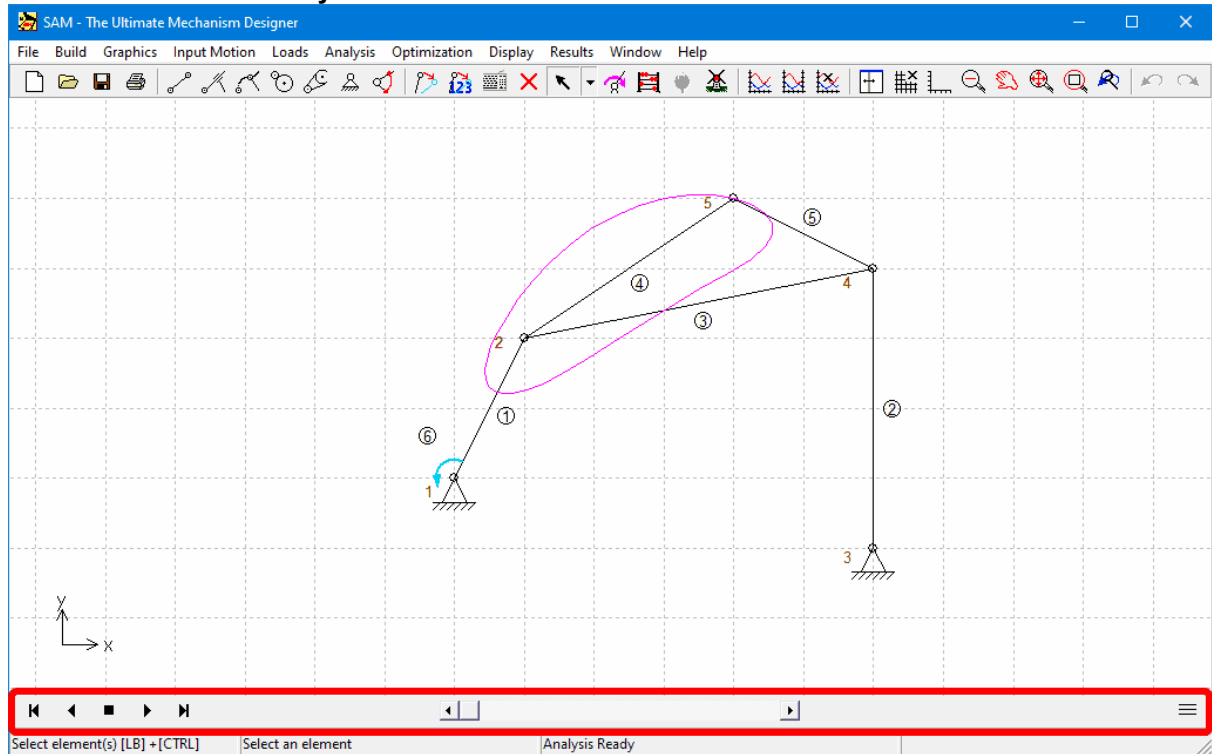
Mobility analysis based on Singular Value Decomposition (SVD)

A SVD analysis is introduced to analyse whether the mechanism is kinematical indeterminate and/or statically overdetermined. In contrast to the previous analysis based on the theory of Grübler, the SVD is able to give the right diagnosis in the case the mechanism is simultaneously kinematical indeterminate (because on global mechanism scale a constraint is missing) and statically overdetermined (because locally there is one constraint too many).



Mechanism that is both kinematical indeterminate (element 10 is free to rotate) and statically overdetermined (only one diagonal allowed in the quadrilateral formed by nodes 2-6-5-4)

Advanced Animation Player



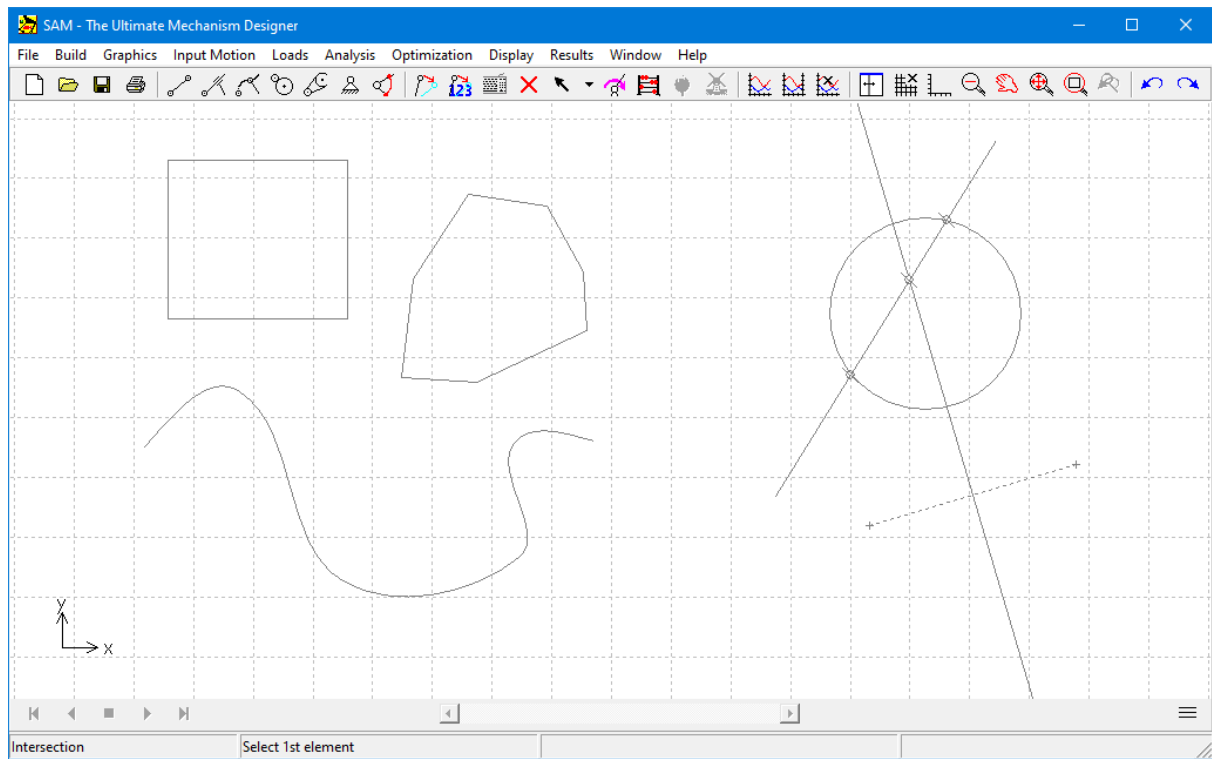
Screenshot highlighting the new animation player

(Semi-) dynamic construction aid

The existing basic construction tool has been made dynamic. This means that center lines and intersections of basis shapes like lines and circles are automatically modified when the underlying shapes are edited.

Additional graphic shapes

- Polyline
- Bezier
- Rectangle



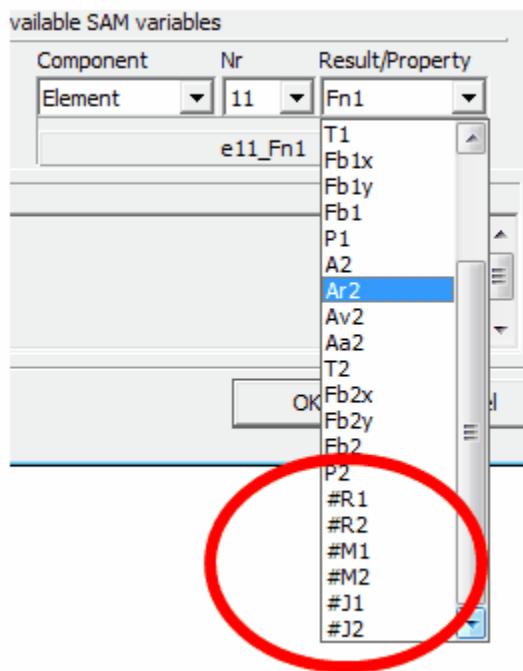
Screen capture of drawing features

New functions available in formula editor (User Defined Results)

- rms
- first
- last
- intf (not just total integral value *int* but also intermediate results)

Element properties available in formula editor (User Defined Results)

Next to element related results one can now also include element properties, such as mass or radius of a pulley, in the equations. These properties are indicated by #property and included at the end of the drop-down list of items that can be selected.



Screenshot showing the selectable items of a belt drive including the element properties that can be selected as of SAM 8.0

14.6 What was new in SAM 7.0

User Defined Results *(only in SAM Professional)*

A sophisticated formula parser offers the possibility to combine simulation results in multiple ways to derive user defined results.

Constrained Optimization *(only in SAM Professional)*

The feature of User Defined Results opens the path to perform constrained optimization by adding the constraint condition as a penalty function to the original function one is optimizing.

Easy Calculation of Bearing Forces in Moving Joints

The calculation of bearing forces in joints is now equally easy for fixed and for moving joints. In previous versions of SAM the calculation of bearing forces in moving joints required manual steps, which have been eliminated.

New Transformation & Grouping Commands

New transformation (mirror, translation, rotation, ..) and grouping commands have been added that allow the user to easily modify elements/mechanisms but also graphical items.

New Licensing System

The new licensing system supports node-locked licenses and floating licenses. It is a two-step approach in which customers first receive an activation ticket. During the activation process, which requires internet access to send information to our central license manager, the final license will be generated and send to the customer.

Various Smaller Enhancements

Display of a scale; export selection & reordering of a subset of results; when exporting a mechanism as DXF file also the displayed trajectories/path(s) are exported, ...

14.7 What was new in SAM 6.2

Unicode Support

The main feature of SAM 6.2 is the support of strings based on Unicode, which facilitates the translation of SAM into languages requiring Unicode representation.

14.8 What was new in SAM 6.1

Project Notes

Text editor functionality has been added such that the user can include and store project notes directly in your SAM project file.

Input Motion

Next to the Natural Cubic Spline (with second derivative equal to zero at the first point and last point of a series of points) the Clamped Cubic Spline is also supported. The latter allows the definition of the first derivatives in the first point and last point of a series of points.

Optimization

After completion of the global exploration of the solution space it is now possible to execute an automated local optimization of ALL solutions found during the global exploration.

Display

- All elements can have individual color/line-type settings.
- A cross-hair (incl. horizontal/vertical mouse snap) simplifies the modeling of mechanisms.

Pan/Zoom

Pan and Zoom are supported by the scroll wheel in a similar way as done in the Google Earth application.

Graph/Mechanism Window

The two windows can be dimensioned individually. Furthermore, a new icon is introduced to easily switch between the three display modi: 1. mechanism, 2. graph, 3. mechanism & graph.

Automated Updates

Checking the presence of update and installing updates has been simplified tremendously. If requested, the checking & installation can even be done automatically..

Videos in GIF Format

Next to the existing AVI format, SAM now also supports the much preferred (Animated) GIF format.

Additional Checks

Many extra automated checks have been introduced to minimize the risk of input or problem definition flaws that lead to incorrect analyses.

Bug Fixes

A number of bugs and inconsistencies have been resolved.

14.9 What was new in SAM 6.0

Optimization (Professional version of SAM only)

Taking the initial design/topology as a starting point one can for example further improve the quality in which the trajectory of a coupler point equals the target trajectory by changing the geometry of the mechanism within pre-defined ranges. Or one can minimize the peak or RMS value of the driving torque of a mechanism by adding a compensating mass and let SAM determine the optimal value of the mass and its position within the allowable range. Just as in the case of the trajectory optimization one can also specify a reference function and minimize the difference between the actual and the

reference function. When designing for example fitness equipment one is generally seeking a predefined force as function of displacement.

The goal for optimization can be the minimization or maximization of a variety of properties (peak, RMS, average, ...) of the difference between the actual and the target behaviour of a mechanism, such as :

- trajectory of a node (with or without prescribed timing)
- any motion or force quantity (as function of time or another quantity)

SAM seeks the optimum by modifying the following properties within user-defined ranges :

- geometry of mechanism
- element properties, such mass, spring constant, transmission ratio, ...

The optimization process in SAM is based on a two step approach, consisting of :

- Exploration of the design space
- Optimization of a specific solution

First, the entire parameter space is explored globally using a combination of a pure Monte-Carlo technique and a so-called Evolutionary Algorithm, which is a optimization technique derived from Genetic Optimization. The top list of such a global exploration are shown in the Explore list box, which displays the value of the optimization function and the corresponding parameters. The individual with the best property is listed at the top.

Next, the designer can select one of the results from the Explore window and start a local optimization. This local search can be either based on a Simplex technique or on a Evolutionary Algorithm with a smaller parameter range centered around the selected solution.

The combination of a global exploration strategy and a local optimization strategy (with the designer in the loop for selecting the mechanism that is further optimized) is believed to give the best trade-off between speed and coverage of the design space.

Next to this user controlled approach a fully automated approach can also be selected.

Context-Sensitive Right Mouse Button

In previous versions of SAM the user interaction model was based on "Select action from menu or toolbar, then select item to apply action to", for example Delete Element followed by a selection of the element to be deleted. Many programs, such as Microsoft Office, use a different approach which is based on "Select an item and press right mouse key to see which actions can be applied to the selected item".

In order to satisfy both experienced SAM users and novice users which are used to the Microsoft Office approach, SAM 6.0 supports both user interaction approaches.

News Alert

If requested, users that are linked up to the internet are automatically informed about any news related to SAM as soon as they start-up SAM.

License Levels: Professional, Standard and Light

Three different license levels have been introduced to better tune the performance and price to the need of the customers. These three levels are:

- Professional: Kinematics, Kinetostatics (force analysis) and Optimization

- Standard: Kinematics and Kinetostatics (force analysis)
- Light: Kinematics

14.10 What was new in SAM 5.1

Input Motion

The optimized analysis kernel now supports up to 10 simultaneous independent input motions (the maximum used to be 3).

The 5. order polynome motion profile with arbitrary velocity and acceleration at the interval boundaries has been added to the list of standard motion laws. This allows the modelling of very complex input motions.

In the unusual situation that the default analysis parameters (max. number of iterations and iteration threshold) do not suffice, they can now be manually edited (see File/Preferences/Analysis).

Non-Linear Spring

A new element "non-linear spring" has been added to the list of existing element types. It allows the modelling of a spring with a non-linear force-elongation characteristics by defining this relation in a table.

Display

Via the toggle "Real-time Animation" (see File/Preferences/Display) the animation of the mechanism on the screen is linked to the actual speed.

The symbols drawn in the xy-plot to identify individual curves used to be linked to each point of the curve. In case of small intervals and many time steps this made it hard to distinguish between the various curves because all were overloaded with symbols. As of SAM 5.1 it is possible to define an increment between the points that a symbol is assigned to (see Results/Options).

14.11 What was new in SAM 5.0

Optimization of the analysis-kernel:

- unlimited complexity of the mechanism (no restrictions on # of nodes/elements)
- unlimited number of motion steps
- significant shorter analysis time

"Real Time" analysis

Analysis and update of animation, paths, hodograph and all plots while the user is modifying the mechanism (for example via dragging of a node with the mouse).

New options regarding the definition of motion/load profiles, including:

- cubic spline
- 2.order velocity profile
- table

"Design Wizard" for the generation of 4-bar mechanisms:

- general purpose
- (angle)function generation (Freudenstein equations)
- three position synthesis

"Design Wizard" for approximated straight line coupler motion:

- Watt
- Chebyshev
- Roberts
- Evans
- Hoecken

"Design Wizard" for exact straight line coupler motion:

Various alternatives incl. planetary gears, beltdrives are available

Generation of video file (AVI-format)

14.12 What was new in SAM 4.2

Force analysis extended to mechanisms containing belts and gear

In contrast to previous versions of SAM it is now possible to perform a force analysis of a mechanism that includes a gear or a belt (without pre-tension). In the analysis the non-linearity caused by the fact that the internal force of a belt must be positive (tension) and the internal force of a gear transmission must be negative (pressure) is taken into account.

Simple CAD construction tools required for graphical mechanism synthesis

Points, lines, circles, centrelines and intersections can be constructed directly in SAM. This enables the graphical mechanism synthesis, such as the well-known Burmester approach. Also, it is possible to add text to the drawing of the mechanism.

Display of the fixed/moving centrode and of the evolute

The fixed or moving centrode of any linkage and the evolute of any node can be displayed.

Various display options

In the dialog Display/Options the following options have been added:

- Comment that is included with each screenshot (Ctrl-C)
- Inversion of the direction of the velocity hodograph

14.13 What was new in SAM 4.1

Project Documentation

Complete project documentation in an ASCII-file --> File/Project Documentation.

Selectable Engineering Units and Number Representations

You can choose your own mix of engineering units (metric, english, custom) and also the way that reals are represented (floating/fixed point, number of digits) --> File/Preferences. Please note that importing or exporting data is always done in the currently active unit system.

Screenshots with filename, date/time and your company name

Screenshots can now be generated by pressing Ctrl-C. If desired, the name of the current file, date/time and the name of your company is automatically integrated in the screenshot. .

Easy screenshot generation of motion/force profile

An extra button has been created in the motion and force dialogs to send the motion/force profile directly to the clipboard.

Start of Analysis without Options Dialog

The function key <F9> starts an analysis without invoking the options dialog. This is especially useful when doing identical analyses of the same mechanism with only slight parameter changes.

New Display Options

The menu Display/Options now contains additional items :

- Animation Delay
- Scaling of Hodograph
- Zoom Extents with/without considering full motion

Export Results

The menu Results/Export contains new options for formatting the export file.

Dynamic link between animation and graph

If you have displayed the mechanism and a graph next to each other (Windows/Tiled) and the graph cursor is ON, you can move this cursor from left to right and the mechanism will move accordingly.

14.14 What was new in SAM 4.0

CAD-Interface via Import/Export of DXF-data

Using graphics components (i.e. machine or product parts) as a base for realistic representation and design support.

For instance, importing the drawing of a machine frame in SAM helps the designer to take the outer limits of his design into consideration. Creating a complete CAD-based mechanism, linking (attaching) it to the schematic mechanism drawing in SAM and animating the results gives a very realistic view of the designed mechanism. This feature can be very practical to avoid collision issues.

Once the conceptual design of a mechanism is ready, the schematic mechanism drawing can be exported to any CAD package via DXF-format to finish the complete product design.

Extended display options

- All entities of SAM (i.e Element, Grid, Force, Support etc.) have their own colour and line styles (solid, dashed etc.). All these can be individually tuned by the user.
- Users can choose to have a black background (as in many CAD-programs).
- The grid setting can be lines, points of off.

Zoom facilities

Various zoom-buttons have been added, such as Zoom Rectangle Zoom Extents, Zoom Minus and Zoom Previous (note: the zoom functions are 'Transperent', which means that they can be used within any process, without influence. Even during animation).

The 'Auto Zoom' button is skipped, due to the new Zoom facilities.

Export of results

Any items calculated by SAM (i.e nodal position, force, elongation, angle etc.) can be exported in plain ASCII text format. the user can setup a format accepted by standard Spreadsheet packages. Moreover the user can export data in the SAM Input Motion File format (*.CUR). So, motion data calculated as the output of one analysis can be used as input for another project.

Transformations

All the mechanism (if desired including the Graphics) can be rotated and scaled.

Expanded UNDO functionality

The user can undo any previous action (one level).

Input motions and loads dialog

All selections in the input motion definition and in the force/torque definition can be easily modified. Furthermore, the corresponding graph can be printed now.

Motion file

A motion file can be edited directly from the Input Motion Definition Dialog, using the editor of your choice. Each time a "Save" operation is performed in the editor, the actual motion plot in the Input Motion Definition Dialog is updated.

Manual animation

The user can drag the mechanism manually through all animation steps forwards and backwards. If desired, a new initial or reference position of the mechanism can be defined.

Keyboard entry of nodal coordinates

A speedbutton is added to directly edit nodal coordinates via the keyboard.

Norm or vector-length

In the previous version of SAM, only individual x- or y-components of analysis results were available. In version 4.0 also the norm or vector-length ($\sqrt{x^2+y^2}$) can be plotted.

Chapter



15

15 Glossary of Terms

velocity hodograph

Locus of the arrowhead of the velocity vectors (rotated 90 degree) of a moving point.

path

Line that a moving point describes in the fixed reference system.

kinetostatics

Quasi-static force analysis based on the kinematic motion of the mechanism. In each position the force equilibrium due to inertia forces (acceleration), external forces and spring, damper and friction forces is calculated. In the kinetostatics analysis, the forces result from the specified motion, whereas in a real dynamic analysis the motion that results from the specified forces are calculated via integration procedures.

Group

A group is a selection of graphical items that is treated as one entity.

Evolute

Locus traced on the fixed coordinate system by the instantaneous centre of motion curvature of a node.

Centrode

Locus of the instantaneous centre of velocity of a linkage (described either in the fixed coordinate system or in the moving linkage coordinate system).

Moving Centrode

Locus traced on the moving (linkage related) coordinate system by the instantaneous centre of velocity of a linkage.

Fixed Centrode

Locus traced on the fixed coordinate system by the instantaneous centre of velocity of a linkage.

Index

- 3 -

3-Position Synthesis (I) 62, 64
3-Position Synthesis (II) 64

- 4 -

4-Bar Mechanism 58, 60

- A -

A quick example 30, 83, 150, 172, 175, 177, 208
About this User's Guide 12, 13, 23, 24, 26, 30, 78, 118, 172, 173, 177, 206, 208
Analysis Menu 183
Analysis Problems 190
Analysis Results 21
Approximated Straight Line Mechanism 69, 70, 71, 72, 73, 74, 206

- B -

Beam 132
Bearing Forces 88
Belt 150
Belt drive with external force 115
Belt-Coupled Dyad 69
Building a new mechanism (without Design Wizard) 45, 46, 176, 178, 179
Building Multi-Node Linkages 82

- C -

CAD Interface 21, 30, 78, 83, 175, 177, 191, 208
closed Bezier curve 138
Constrained Optimization 22, 122
Crank Rocker Mechanism 106

- D -

Design 16, 60, 69, 206
Display Menu 184, 207

Double Crank Mechanism 109
Driving Torque Minimization 113, 118
DXF Template File 191

- E -

European Academic Software Award 24
Exact Nodal Coordinates 78, 172, 173, 176
Exact Straight Line Mechanism 64
Example of a 4-bar mechanism (motion analysis) 46, 58, 78, 96, 100, 102, 132, 172, 173, 176, 208
Exporting a motion file 80

- F -

force analysis 88
force in a local coordinate system 90
Force/Torque Dialog Box 181
Forces in Moving Nodes 88
Format 80

- G -

Gear 141
Geared Double-Crank Mechanism 66
General (nodes) 58
Graphics Menu 177

- H -

Help Menu 185
History
 Present and Future 23, 78, 80, 99, 106, 207
Hypo-Cyclic Planetary Gear Mechanism 67

- I -

Import points 138
Import points and control points 138
Importing a motion file 80
Inclined Linear Guiding 81
Input Dialog Box 179, 206
Input Motion 17, 41, 45, 46, 62, 64, 79, 80, 118, 172, 178, 179, 180, 181, 184, 190, 191, 206, 208
Installing SAM 26
Introduction 96

- K -

Keyboard and Mouse 172

- L -

LB 172

Linear Actuator 81, 132, 178

Loads Menu 180

- M -

Maximize(cost function+penalty) 96

Modelling 17, 23, 46, 60, 69, 81, 82, 115, 135, 141, 150, 154, 160, 173, 176, 177, 178, 190, 206, 207

- N -

Node-Locked License 26

Non-linear spring 165, 167

- O -

Objective Performance Number 96, 98

Optimization Menu 183

Optimization Objective 96, 120

Optimization of trajectory 120

Optimization Options 102, 183

options 175

Overview - Capabilities - Optimization 22, 23

- P -

Penalty Function 96

Planetary Gear-Coupled Dyad 68

Planetary Mechanism (Hypo-Cyclic) 109

Post-Processing 22, 30, 41, 45, 46, 96, 99, 106, 113, 118, 120, 184, 206, 207

Preference 175

project options 175

Purchase Agreement 13, 30, 83, 160, 172, 177

- Q -

Quick-Return Mechanism 106

- R -

RB 172

Reference or Target Definition 99

Running a sample project 30, 45, 46, 83, 175, 183, 184

- S -

Settings Evolutionary Algorithm 103

Settings Simplex Algorithm 104

Slider 135

Slider Driven Gear Transmission 108

Symmetric Slider-Crank Mechanism 65

- T -

The Design Process 16, 17, 30, 41, 46, 58, 59, 60, 62, 64, 65, 81, 82, 108, 118, 120, 173, 176, 177

Transfer Mechanism (incl. gravity effects) 113

trial license 26

- U -

Units 78, 79, 80, 178, 179, 184, 208

User Defined Results 84

Using Graphics 83

Using the Design Wizard to create a mechanism 41, 58, 62

- W -

What is SAM ? 8, 12, 17, 21, 22, 23, 41, 45, 46, 67, 68, 78, 82, 83, 88, 96, 99, 100, 102, 103, 108, 109, 118, 120, 141, 154, 160, 172, 173, 175, 176, 177, 178, 183, 184, 190, 204, 206, 207

What was new in SAM 4.0 ? 208

What was new in SAM 4.1 ? 207

What was new in SAM 4.2 ? 207

What was new in SAM 5.0 ? 206

What was new in SAM 5.1 ? 206

What's new in SAM 6.0 ? 12, 17, 22, 23, 30, 45, 96, 100, 102, 103, 104, 113, 115, 118, 141, 150, 154, 160, 165, 167, 173, 176, 185, 204, 206, 207

Window Menu 185