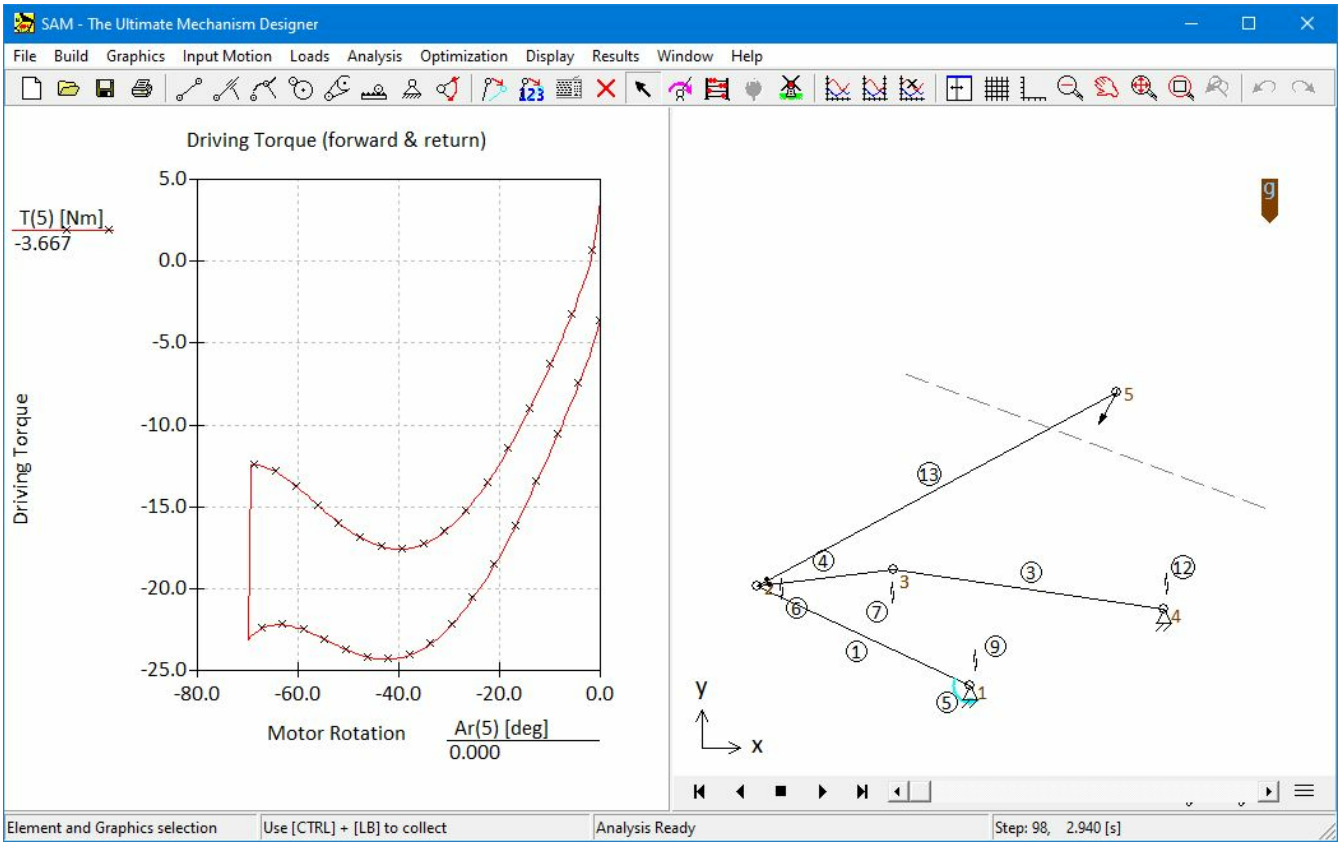
SAM 8.4

The ultimate mechanism designer



© 2024 ARTAS - Engineering Software

Contents

1 Preface 8

1.1 What is SAM ... 8

1.2 What is new in SAM 8.4 ... 8

1.3 About this User's Guide ... 13

1.4 End User License 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, 8.1, 8.2 and 8.3 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

© 2024 ARTAS - Engineering Software

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 80
7.1 Units ... 80
7.2 Number Representation ... 80
7.3 Exact Nodal Coordinates ... 80
7.4 Motion File ... 81
Format ... 82
Importing a motion file ... 82
Exporting a motion file ... 82
7.5 Multiple Input Motions ... 83
7.6 Linear Actuator ... 83
7.7 Inclined Linear Guiding ... 83
7.8 Building Multi-Node Linkages ... 84
7.9 SAM/CAD design process ... 85
7.10 User Defined Results ... 86
Distance ... 88
8 Kinetostatics 94
8.1 Reaction/Bearing Forces ... 94
8.2 External Force/Torque ... 97
9 Optimization 104
9.1 Introduction ... 104
9.2 Optimization Objective ... 105
Objective Performance Number ... 107
Reference or Target Definition ... 107
9.3 Parameter Space ... 108
9.4 Constraints ... 109
9.5 Optimization Options ... 110
Settings Evolutionary Algorithm ... 112
Settings Simplex Algorithm ... 112
10 Examples 114

© 2024 ARTAS - Engineering Software

10.1 Motion Analysis ... 114
Crank Rocker Mechanism ... 114
Quick-Return Mechanism ... 114
Slider Driven Gear Transmission ... 116
Double Crank Mechanism ... 117
Planetary Mechanism (Hypo-Cyclic) ... 117
Curved Slider ... 119
CAM mechanism ... 120
10.2 Force Analysis ... 121
Transfer Mechanism (incl. gravity effects) ... 121
Belt drive with external force ... 123
Height dependent forces on sports car spoiler ... 125
10.3 Optimization ... 126
Driving Torque Minimization ... 126
Optimization of trajectory ... 128
Constrained Optimization ... 130

11 Elements 140
11.1 Beam ... 140
11.2 Slider ... 143
11.3 Curved Slider ... 147
11.4 Gear ... 151
11.5 Rack Pinion ... 155
11.6 Belt ... 160
11.7 Sensor ... 163
11.8 Spring, Damper, Friction ... 164
11.9 Rotational Spring, Damper, Friction ... 170
11.10 Non-linear spring ... 175
11.11 Gas spring ... 177

12 Integrated Analysis Environment 182
12.1 Keyboard and Mouse ... 182
12.2 Toolbar ... 183
12.3 Menu Commands ... 185
File Menu ... 185
Build Menu ... 186
Graphics Menu ... 187
Input Motion Menu ... 188
Input Dialog Box ... 189
Loads Menu ... 191
Force/Torque Dialog Box ... 191
Analysis Menu ... 193
Optimization Menu ... 193
Display Menu ... 194
Results Menu ... 194
Window Menu ... 195
Help Menu ... 195

© 2024 ARTAS - Engineering Software

13 Literature 198

14 Appendix 200

14.1 Location of Files ... 200
14.2 Status/Warning Messages ... 200
14.3 DXF Template File ... 201
14.4 What was new in SAM 8.3 ... 201
14.5 What was new in SAM 8.2 ... 206
14.6 What was new in SAM 8.1 ... 211
14.7 What was new in SAM 8.0 ... 212
14.8 What was new in SAM 7.0 ... 223
14.9 What was new in SAM 6.2 ... 224
14.10 What was new in SAM 6.1 ... 224
14.11 What was new in SAM 6.0 ... 224
14.12 What was new in SAM 5.1 ... 226
14.13 What was new in SAM 5.0 ... 226
14.14 What was new in SAM 4.2 ... 227
14.15 What was new in SAM 4.1 ... 227
14.16 What was new in SAM 4.0 ... 228

15 Glossary of Terms 232

Index 233

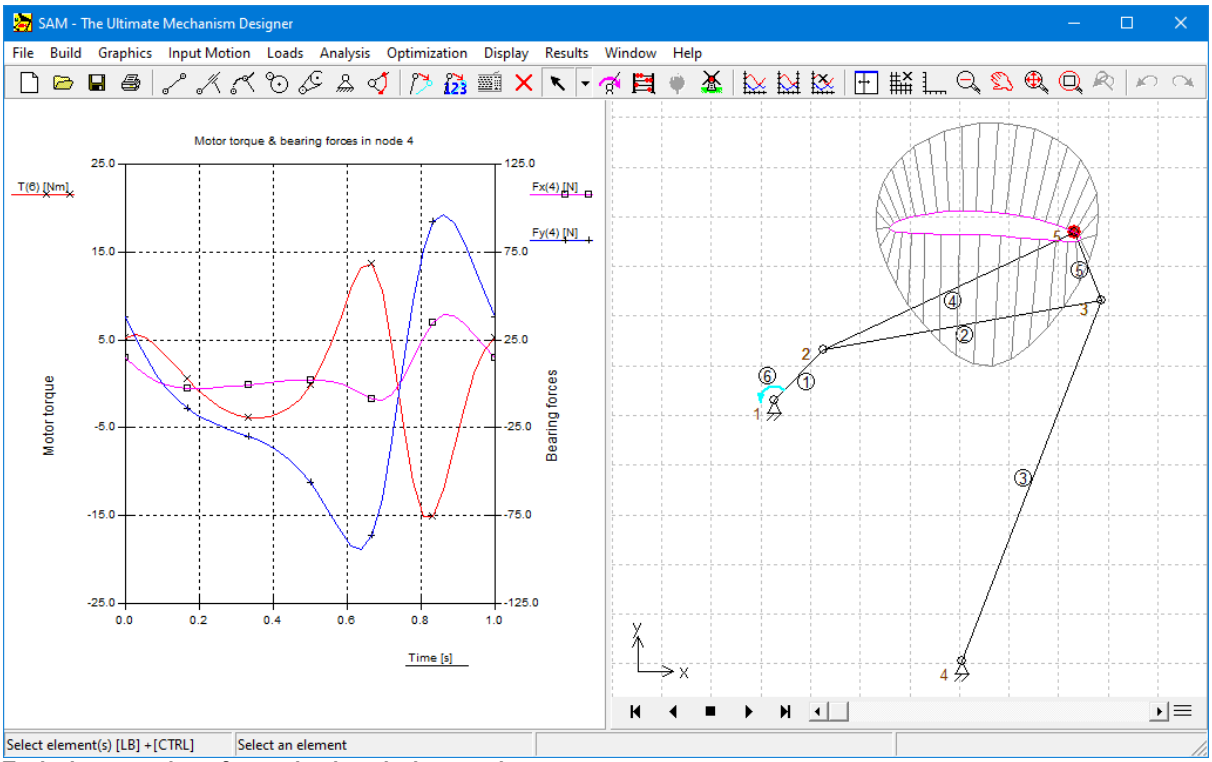
© 2024 ARTAS - Engineering Software

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 8.4

Fixation of elements / bearing forces
In the case of a fixed connection between two or more elements, these elements have one shared bearing. As of SAM 8.4 the combined force that is transmitted via this bearing to the shaft/pin is plotted when selecting the bearing force from a properties dialogue of any of these connected elements. In the past individual forces of each element were plotted, which made it more difficult to interpret the resulting bearing load.

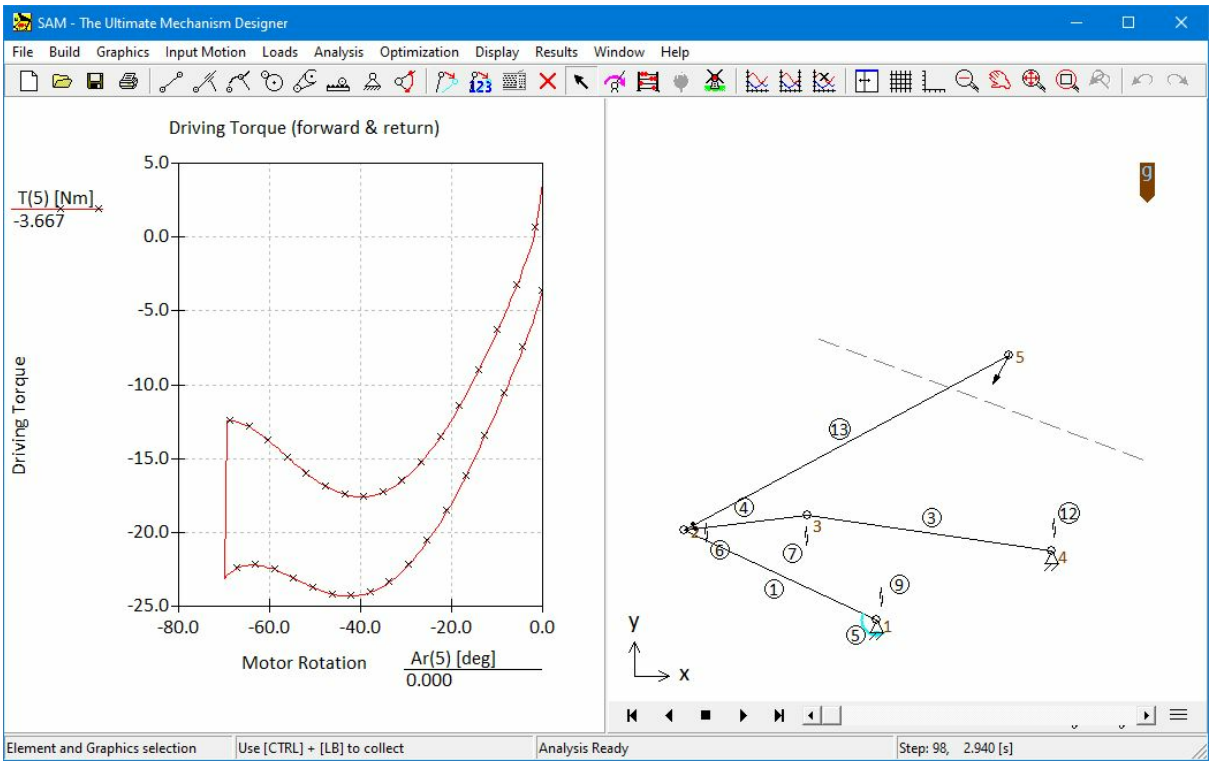
Load dependent friction
- A load dependent friction torque has been added to the already existing constant no-load friction

© 2024 ARTAS - Engineering Software

torque of the rotation friction element.The total friction is now defined by a no-load friction torque, a friction coefficient and a friction radius yielding a friction torque equal to: no-load friction + bearing force * friction coefficient * friction radius. In addition the user has to define which of the involved elements holds the pin.

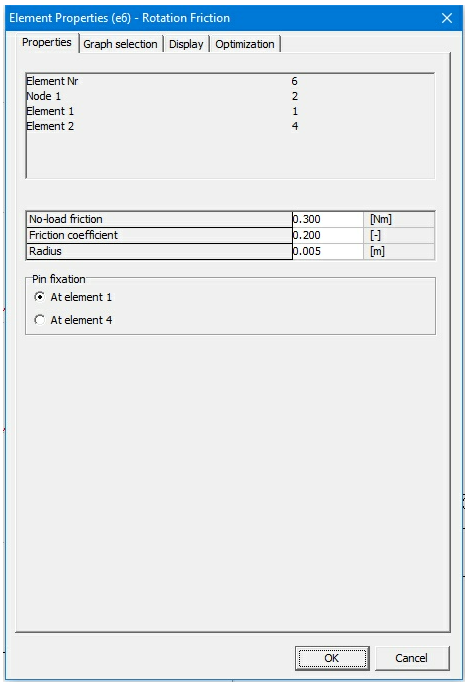
- Friction has also been added to the (curved) slider element via two additional parameters, being the constant no-load friction force and the friction coefficient. The total friction force equals: no-load friction + contact force * friction coefficient.

Example of an motorized automotive engine hood incl. rotation friction elements in all joints



Example of an motorized automotive engine hood incl. normal force dependent rotational friction in all joints, showing the required driving torque (forward & return) as function of the motor rotation.

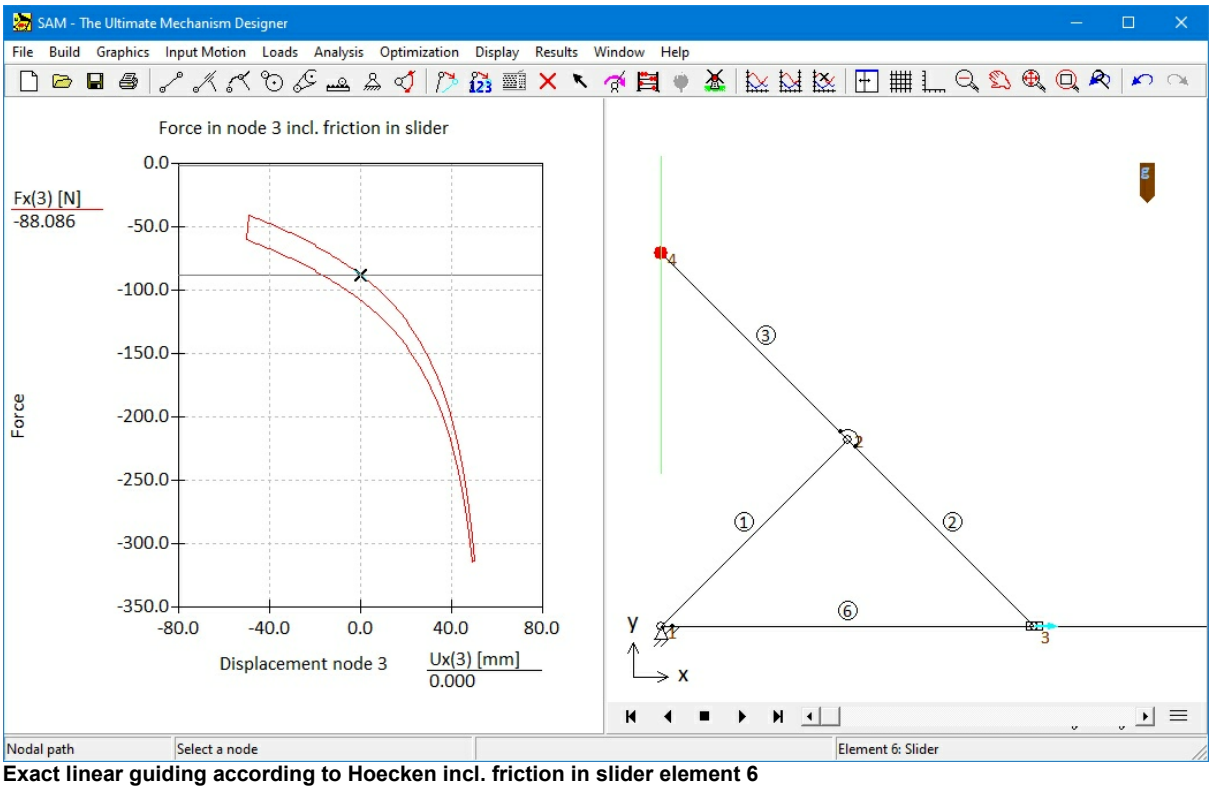
© 2024 ARTAS - Engineering Software



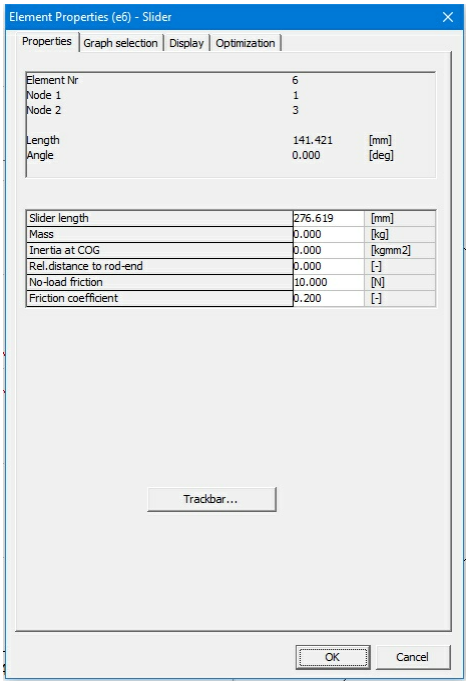
Rotation Friction

© 2024 ARTAS - Engineering Software

Example of a slider element used in a Hoecken mechanism to generate an exact linear motion

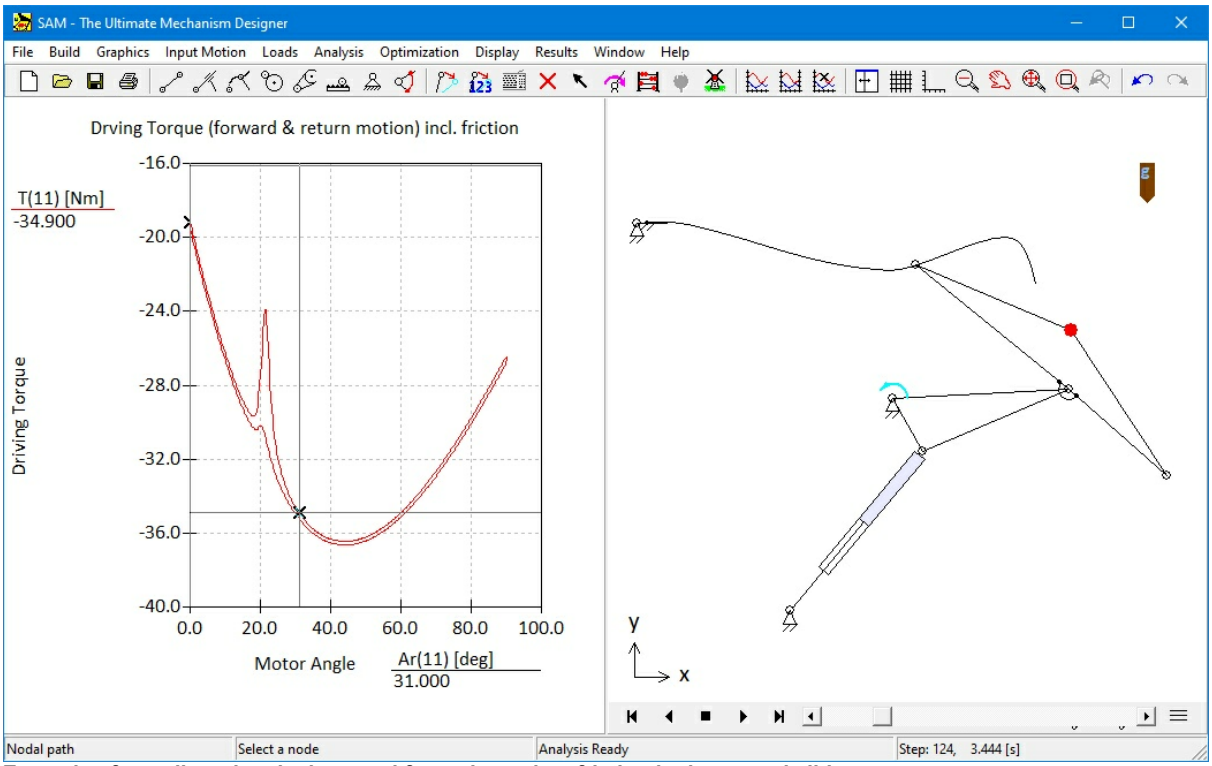


Exact linear guiding according to Hoecken incl. friction in slider element 6

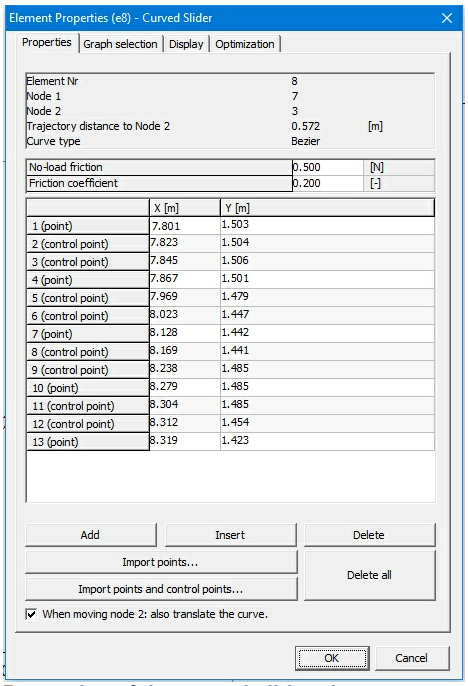


Properties of the slider element

Example of a toolbox riser incl. normal force dependent friction in the curved slider



Example of a toolbox riser incl. normal force dependent friction in the curved slider



Properties of the curved slider element

© 2024 ARTAS - Engineering Software

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 End User License Agreement

Please read the following terms and conditions carefully before using the SAM software. Your use of this software indicates your acceptance of this license agreement. If you do not agree with the terms of this license, you must remove this software from your storage devices and cease to use the product.

Copyright
This software is copyrighted 1991-2024 by ARTAS - Engineering Software. The use and copyright of this software is governed by international copyright treaties. ARTAS - Engineering Software retains full title and rights to this software and documentation and in no way does the license granted diminish the intellectual property rights of ARTAS - Engineering Software. You must not redistribute the registration codes provided on paper, electronically, or in any other form.

Trial / Evaluation
This is not free software. You are hereby licensed to use this software for evaluation purposes without charge for a period of 30 days. Using this software after the evaluation period violates copyright laws and may result in civil and criminal penalties.

License
NODE-LOCKED: The software is licensed on a single, physical and specifically identified computer, and can only run on that specific computer. A Node-Locked license requires that users are physically present and not accessing the machine and using the software from a remote location.

NETWORK/FLOATING: The license (or multiple licenses) resides on a central server of the customer called a license server. The product software is installed on client computers and when the product software is started, it “checks out” a license from the license server. When the product software is terminated on the client, the license is “checked in” and is available for another client to use.

CLASSROOM-KIT: A classroom kit (educational institutes only) consists of 100 floating licenses for the institute, one node-locked license for the professor/teacher plus 100 annual student licenses for distribution to students and use on their personal PC/laptop (the student licenses require internet access during use of the software). Each year the university receives on request new annual student license keys free of charge.

Maintenance
ARTAS - Engineering Software shall have no obligation to provide support or maintenance for the SAM software under this Agreement.

Distribution
This software may be distributed freely in its original unmodified and unregistered form. The distribution has to include all files of its original distribution. Distributors may not charge any money for it. Anyone distributing this software for any kind of remuneration must first contact us for authorization.

Other restrictions
You may not modify, reverse engineer, decompile or disassemble this software in any way, including changing or removing any messages or windows. **Virtualization technology may not be used to circumvent licensing terms**.

Disclaimer
THIS SOFTWARE IS PROVIDED "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. IN NO EVENT WILL ARTAS - ENGINEERING SOFTWARE BE LIABLE TO YOU FOR ANY DAMAGES, INCLUDING INCIDENTAL OR CONSEQUENTIAL DAMAGES, ARISING OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. YOU ACKNOWLEDGE THAT YOU HAVE READ THIS LICENSE, UNDERSTAND IT, AND AGREE TO BE BOUND BY ITS TERMS.

© 2024 ARTAS - Engineering Software

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

© 2024 ARTAS - Engineering Software

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:

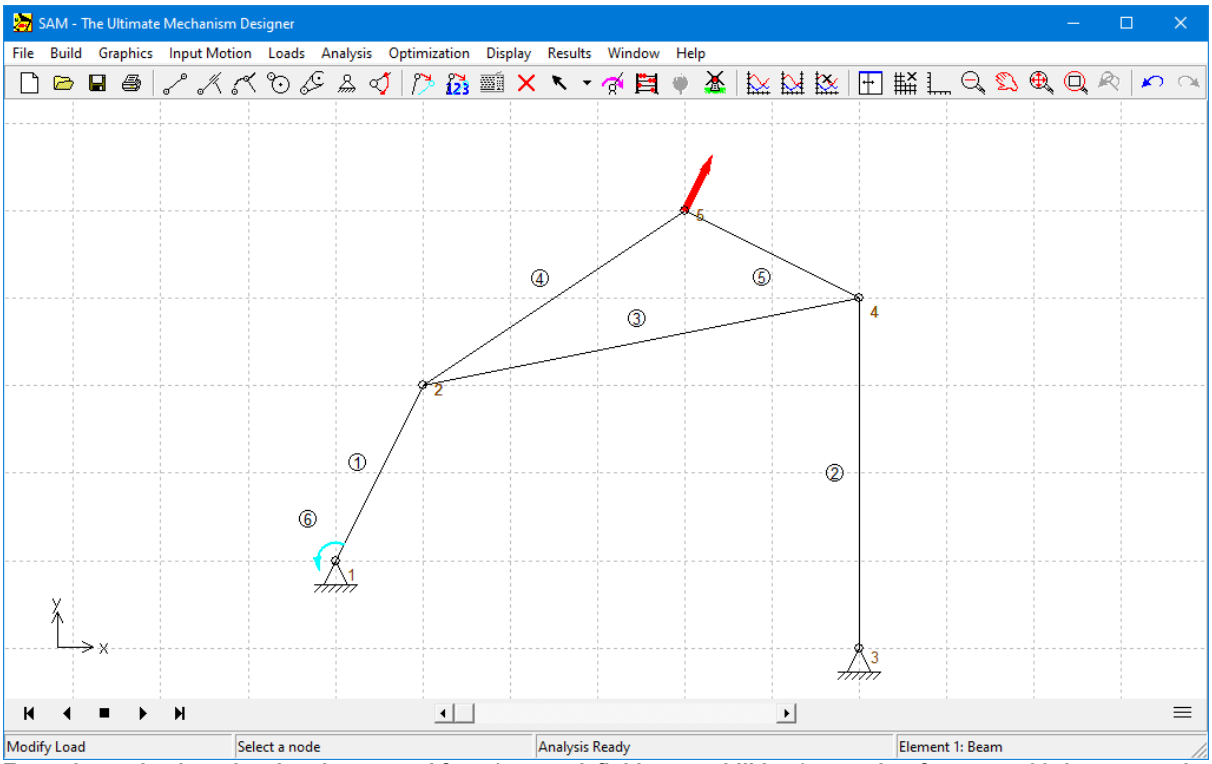
- constant velocity
- polynomial
- cyclical motion
- 2.order velocity profile
- cubic splines

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

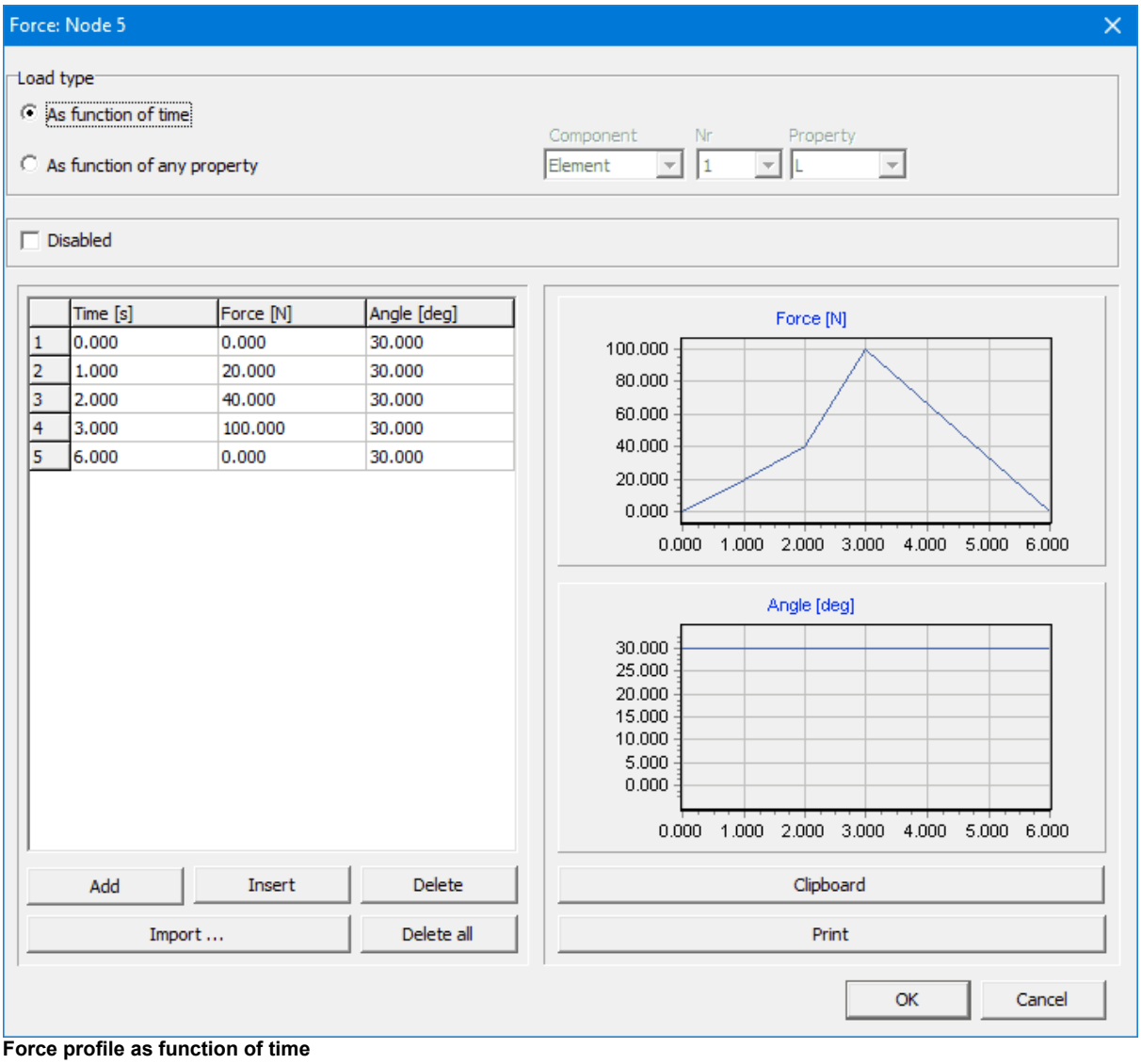
External forces or torques can be applied as function of time or as function of any motion results that is calculated by SAM. To illustrate this feature the following example mechanism is used.

© 2024 ARTAS - Engineering Software



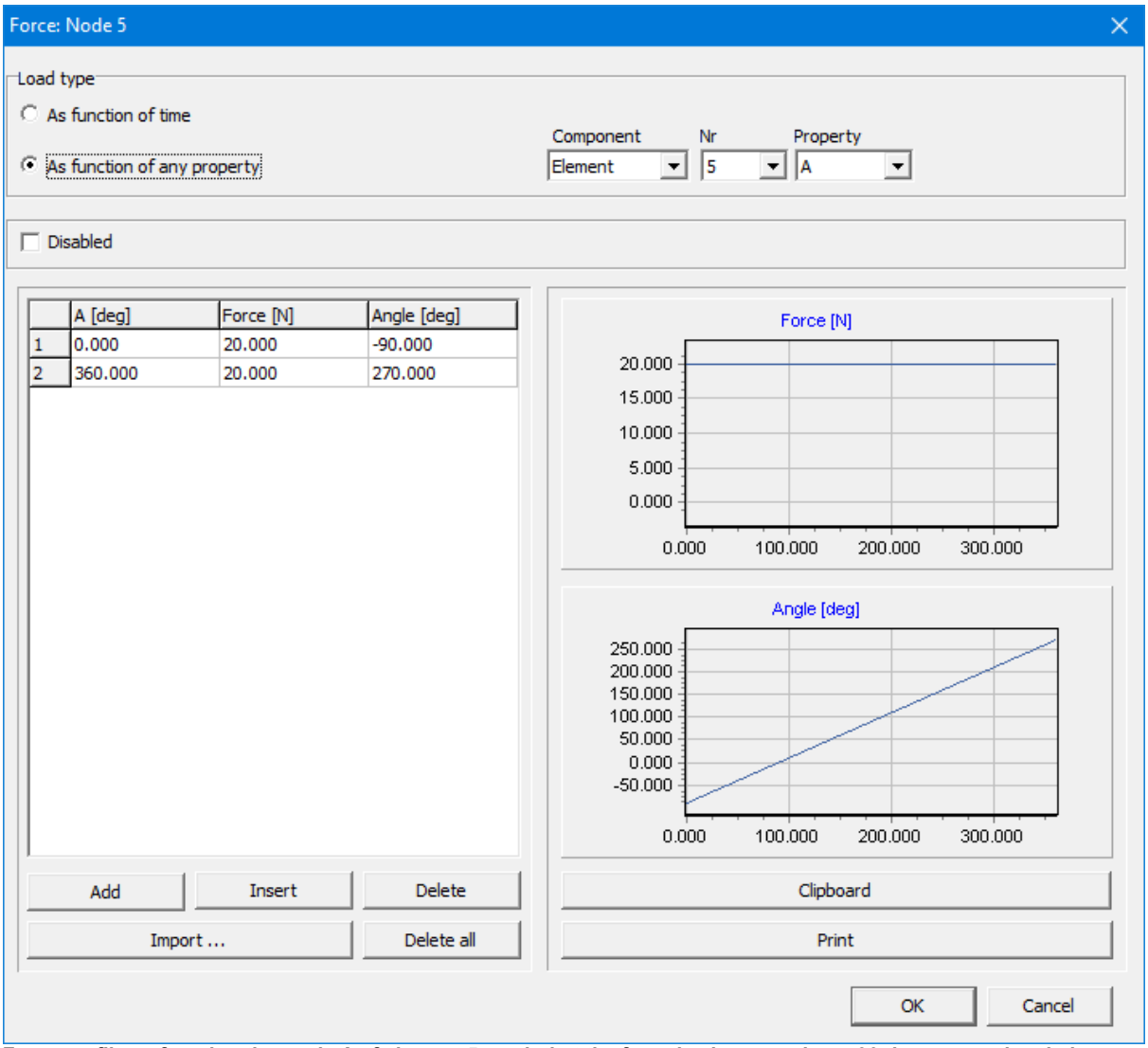
Example mechanism showing the external force/torque definition capabilities (currently a force at a -90 degrees angle relative to the angle of element 5 (node 4 to node 5) is shown).

© 2024 ARTAS - Engineering Software



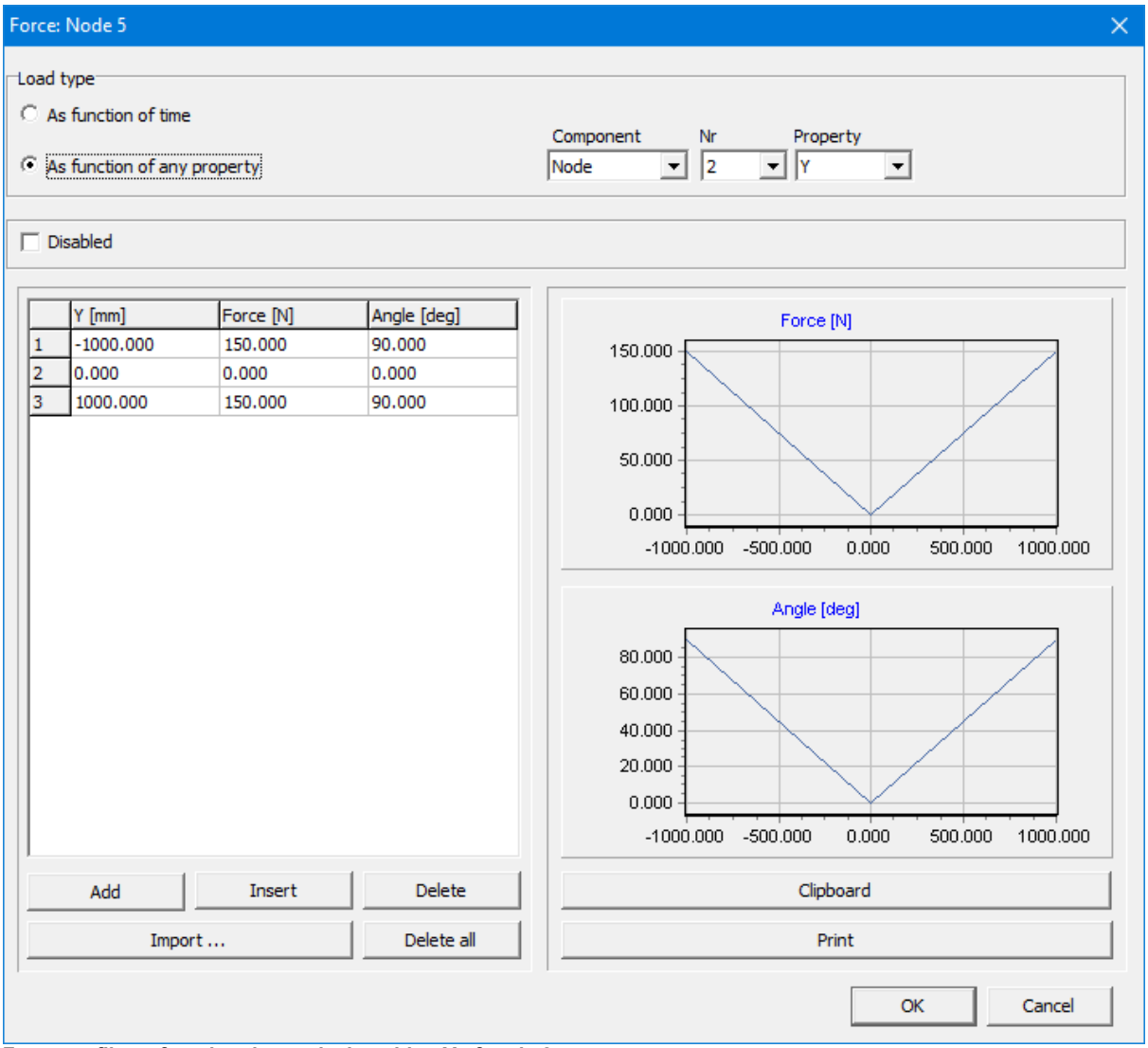
Force profile as function of time

© 2024 ARTAS - Engineering Software



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)

© 2024 ARTAS - Engineering Software



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

© 2024 ARTAS - Engineering Software

- 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 and to define penalty functions for constraint optimization.

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, which might include the distance function to consider for example collision between objects.

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 gnerally 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 of 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

© 2024 ARTAS - Engineering Software

- 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, ...

© 2024 ARTAS - Engineering Software

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, ...

2023: SAM 8.3 Dynamic points, distance between nodes, points, elements and shapes, collision detection, ...

2024: SAM 8.4 Force dependent friction has been added to the rotation friction element and the (curved) slider element. ...

Future plans for SAM include :

- Tools to further improve the modelling, analysis and optimization work flow
- 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).

© 2024 ARTAS - Engineering Software

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, 11 or higher
- At least 50Mb hard disk space (incl. manual).

3.2 Installing SAM

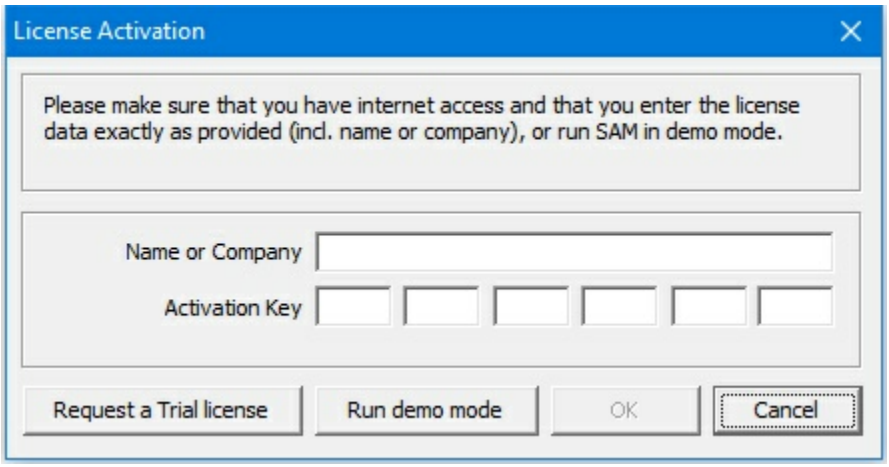
SAM comes in a compressed format. The installation program will expand the files and install each at the proper location on the hard disk.

- Run "sam84us_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

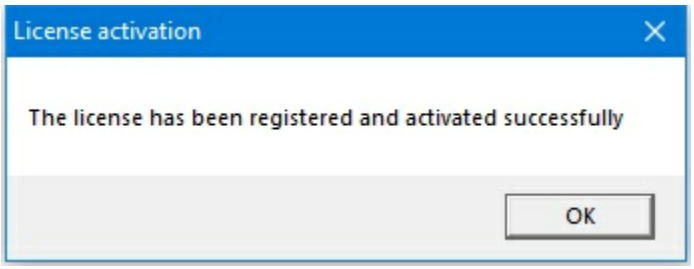
The first time that you start-up SAM after the installation you will be requested to fill in your specific license data that has been supplied with the original software. Please make sure that you have internet access during the activation process and that you type in the name/company and license data 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.



License Information Dialog

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

© 2024 ARTAS - Engineering Software



Floating License
Please contact your IT-manager.

3.3 Use of SAM 3.0, 4.x, 5.x, 6.x, 7.0, 8.0, 8.1, 8.2 and 8.3 project files

The new SAM 8.4 project file format/syntax differs from older versions of the software. Old SAM 5.x, SAM 6.x, SAM 7.0, SAM 8.0, SAM 8.1, 8.2 and SAM 8.3 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 8.4 project format and **can not** open 8.4 project files.

SAM 3.x and 4.x projects were saved as a binary file, while from version 5.0 onward 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.

© 2024 ARTAS - Engineering Software

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

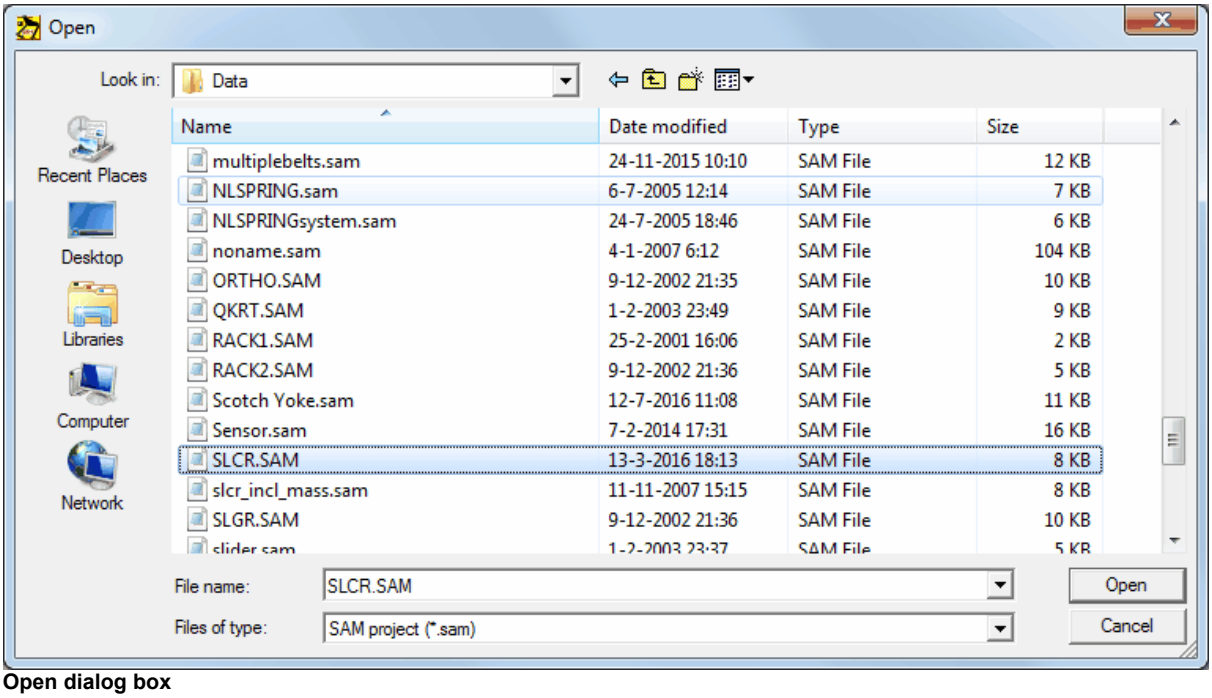
In order to illustrate the various steps we will look at the example of a slider-crank mechanism.

Motion Analysis

Choose Open from the file menu or click 

The Open dialog will appear.

© 2024 ARTAS - Engineering Software

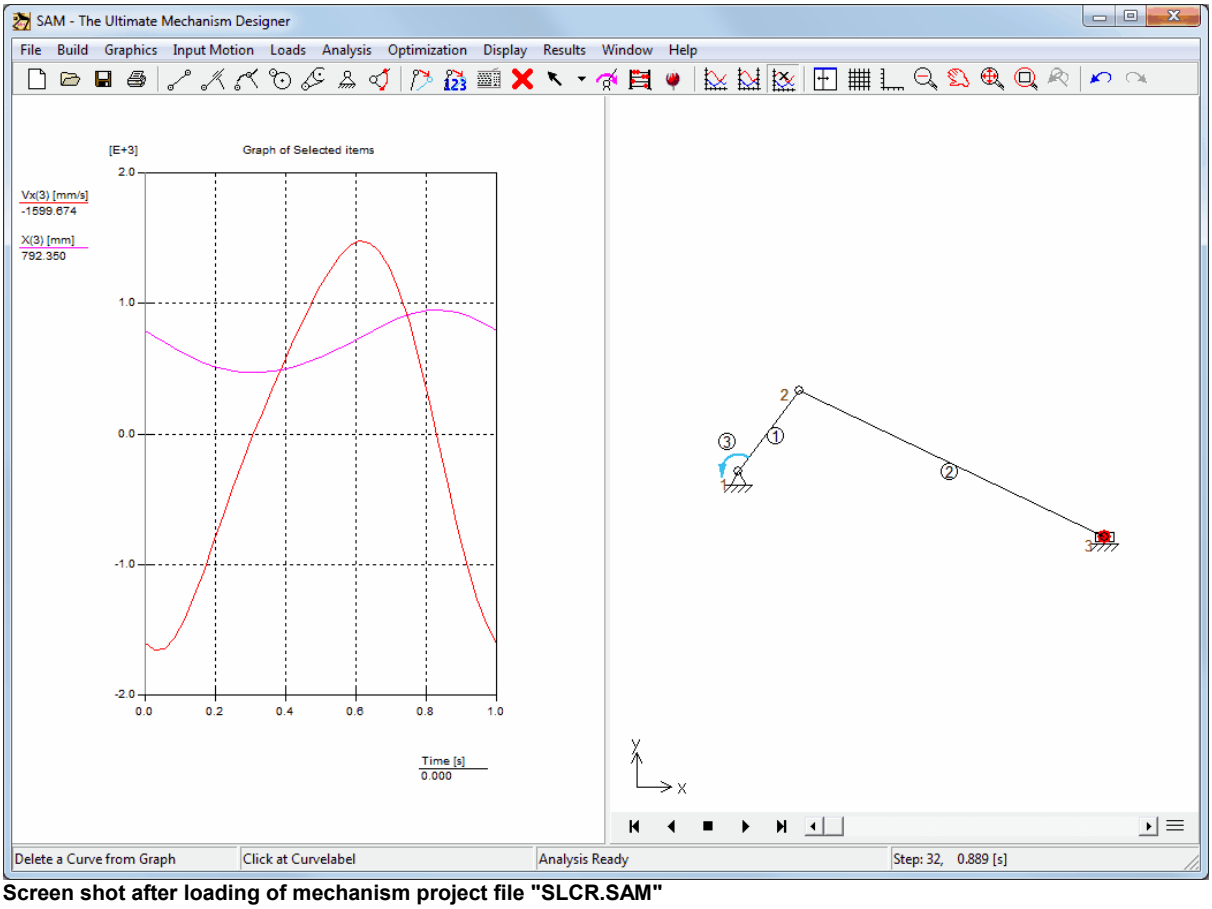


Open dialog box

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 :

© 2024 ARTAS - Engineering Software



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.

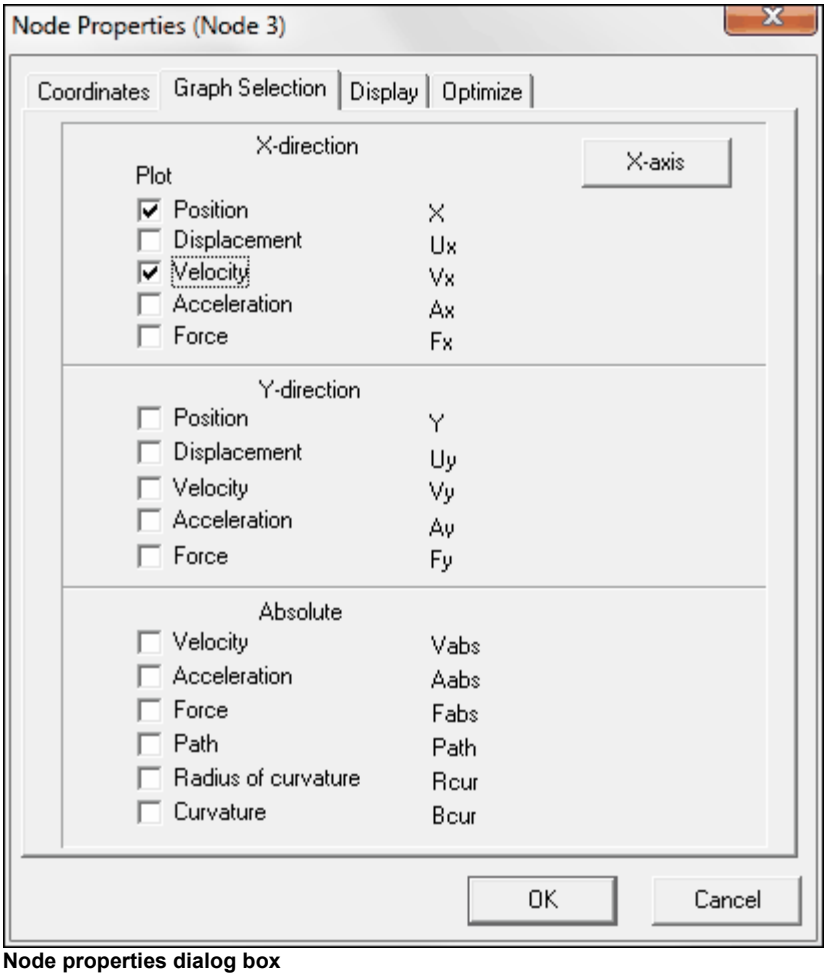
Press any of the Animation keys underneath the mechanism

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.

© 2024 ARTAS - Engineering Software

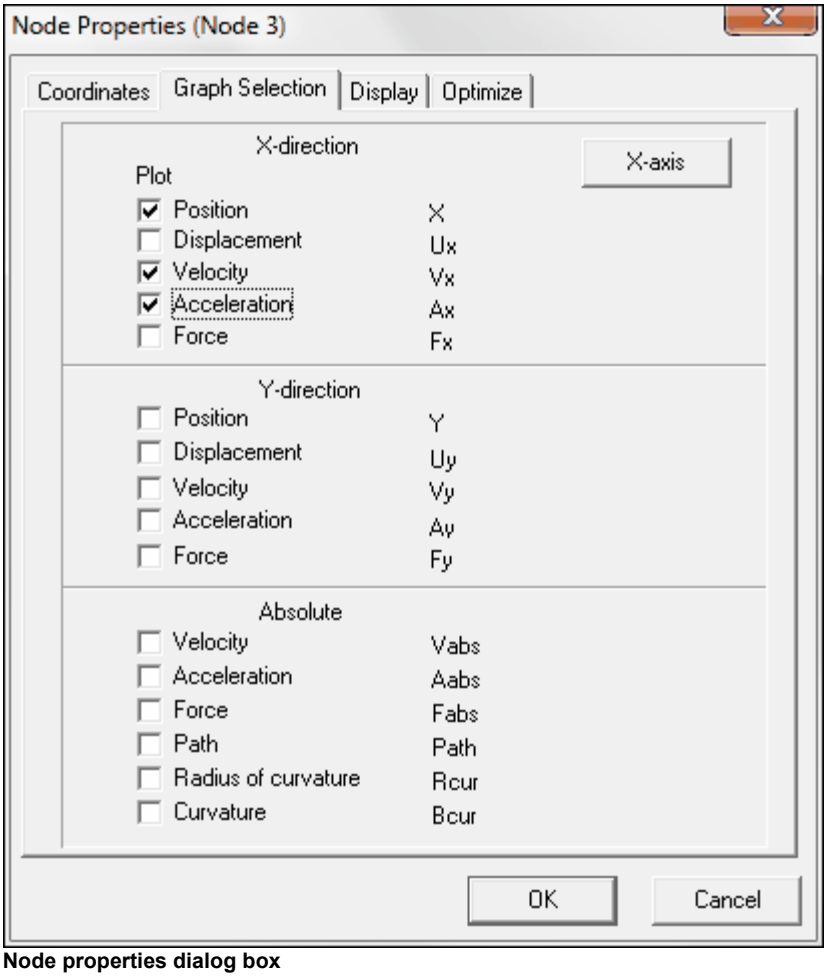


Node properties dialog box

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 Ax and press OK.

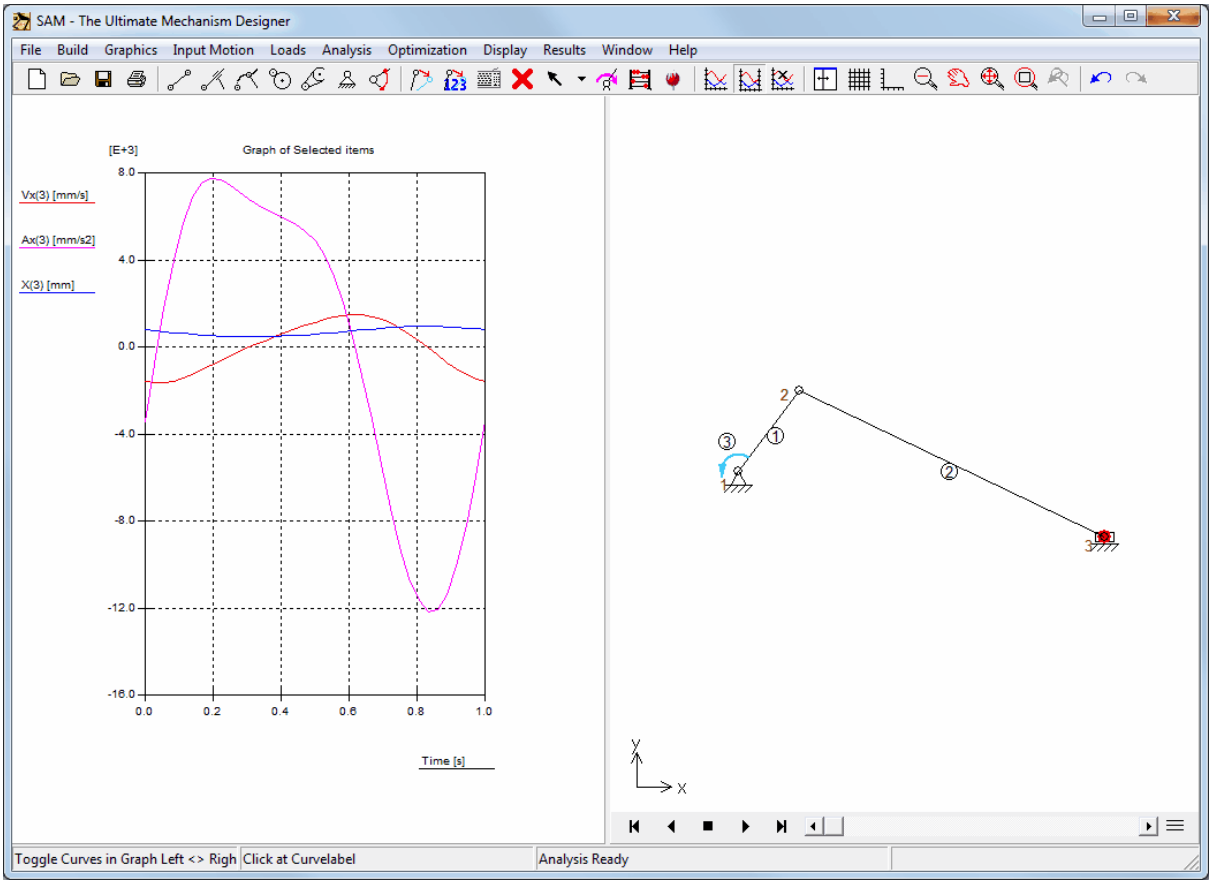
© 2024 ARTAS - Engineering Software



Node properties dialog box

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

© 2024 ARTAS - Engineering Software



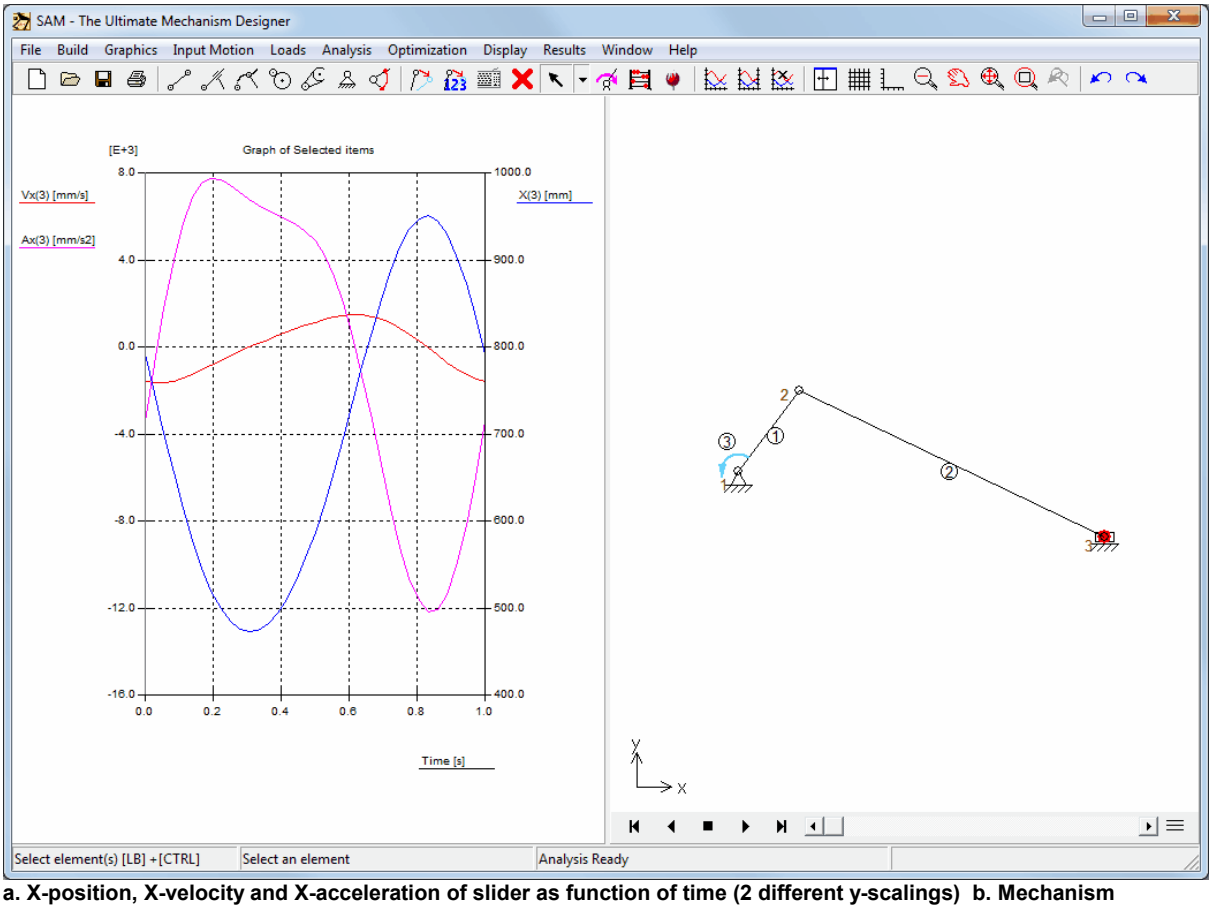
a. X-position, X-velocity and X-acceleration of slider as function of timeb. Mechanism

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).

© 2024 ARTAS - Engineering Software

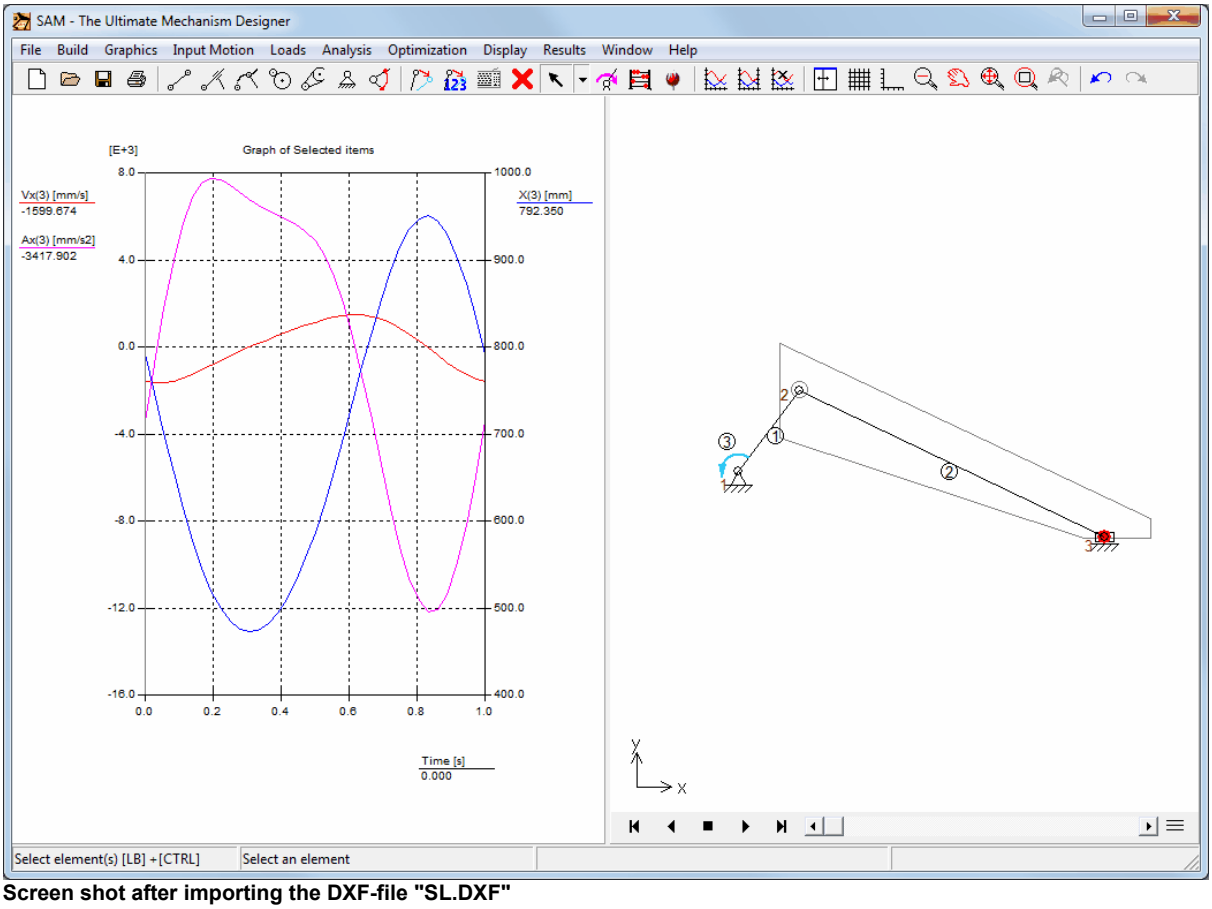


a. X-position, X-velocity and X-acceleration of slider as function of time (2 different y-scalings) b. Mechanism

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:*

© 2024 ARTAS - Engineering Software



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 

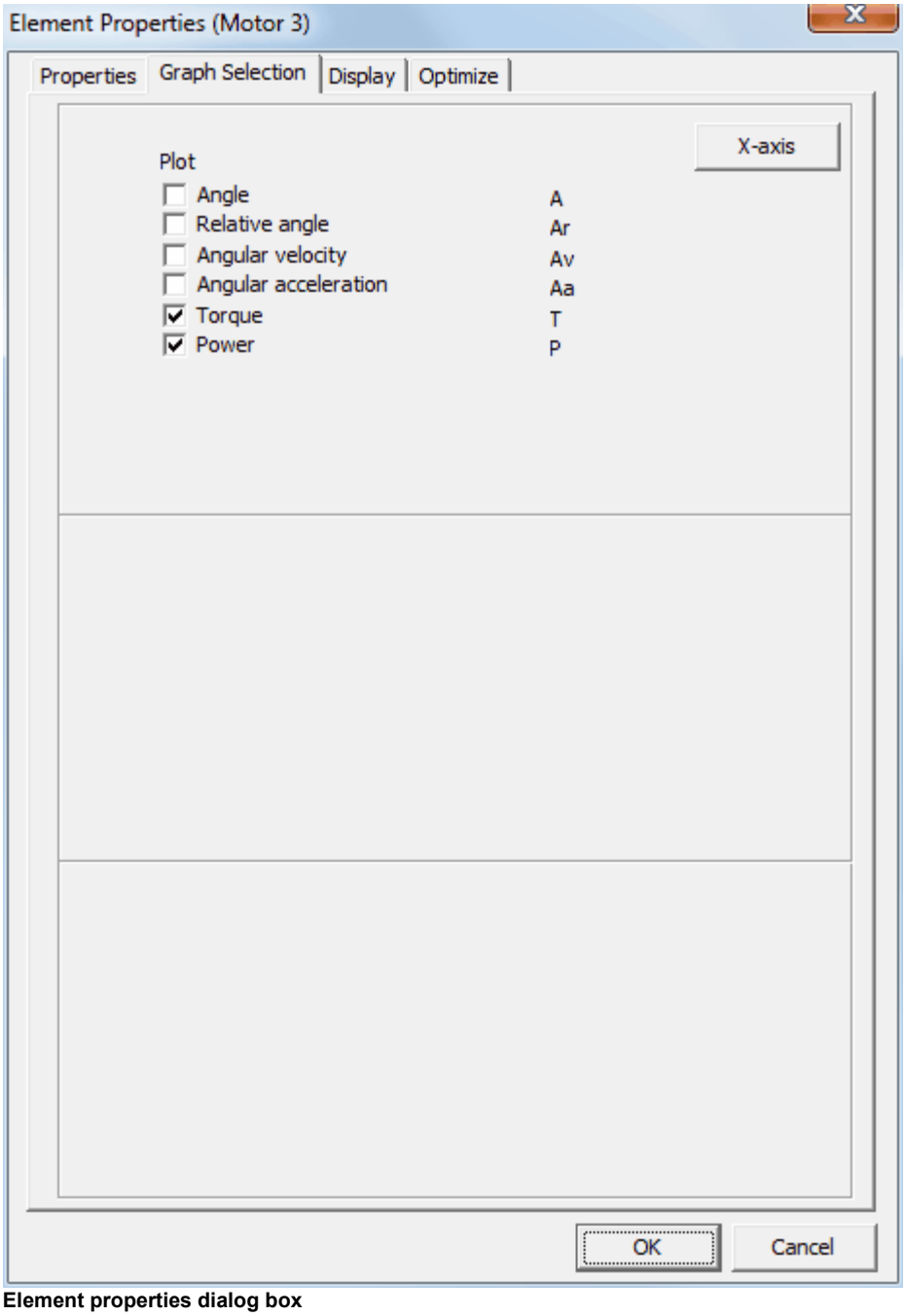
You will now see an animation in which the graphical items move together with element 2.

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

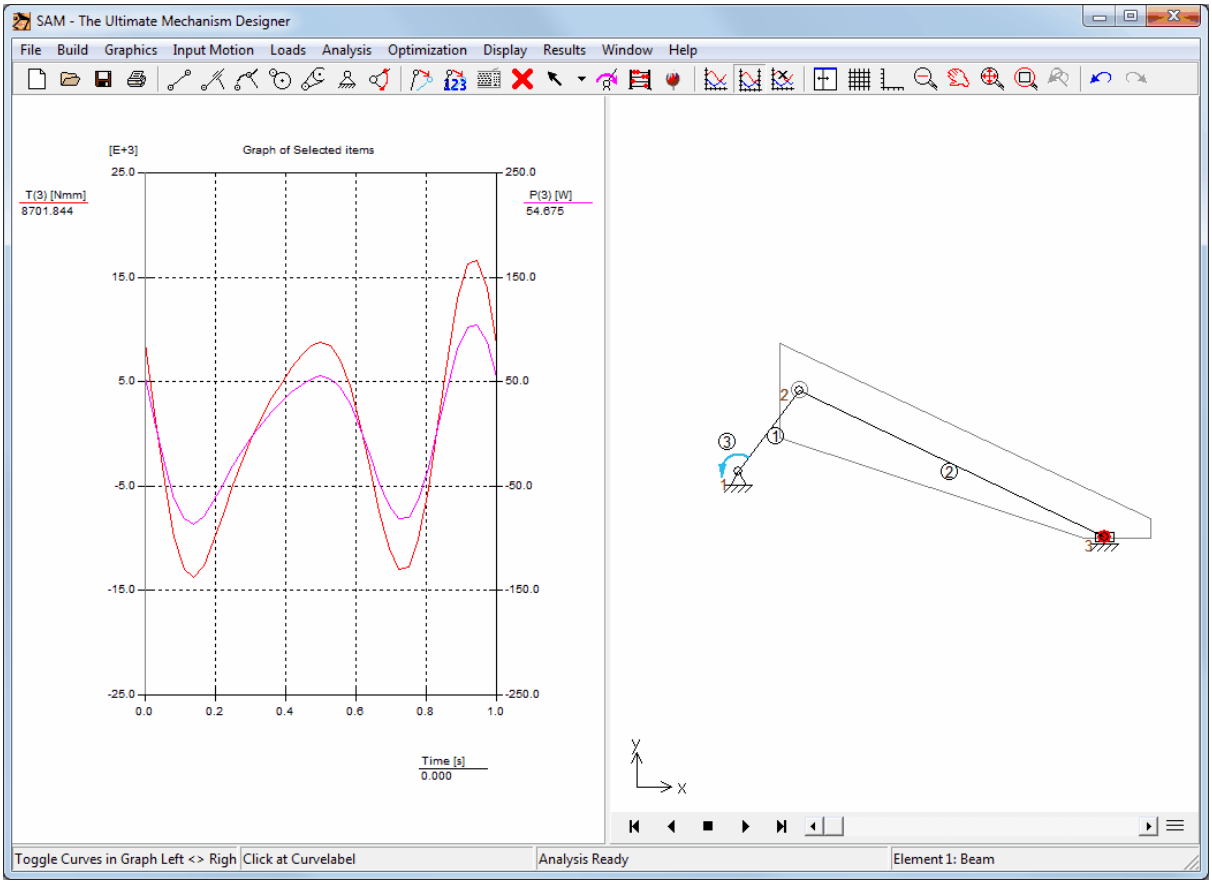
© 2024 ARTAS - Engineering Software



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.

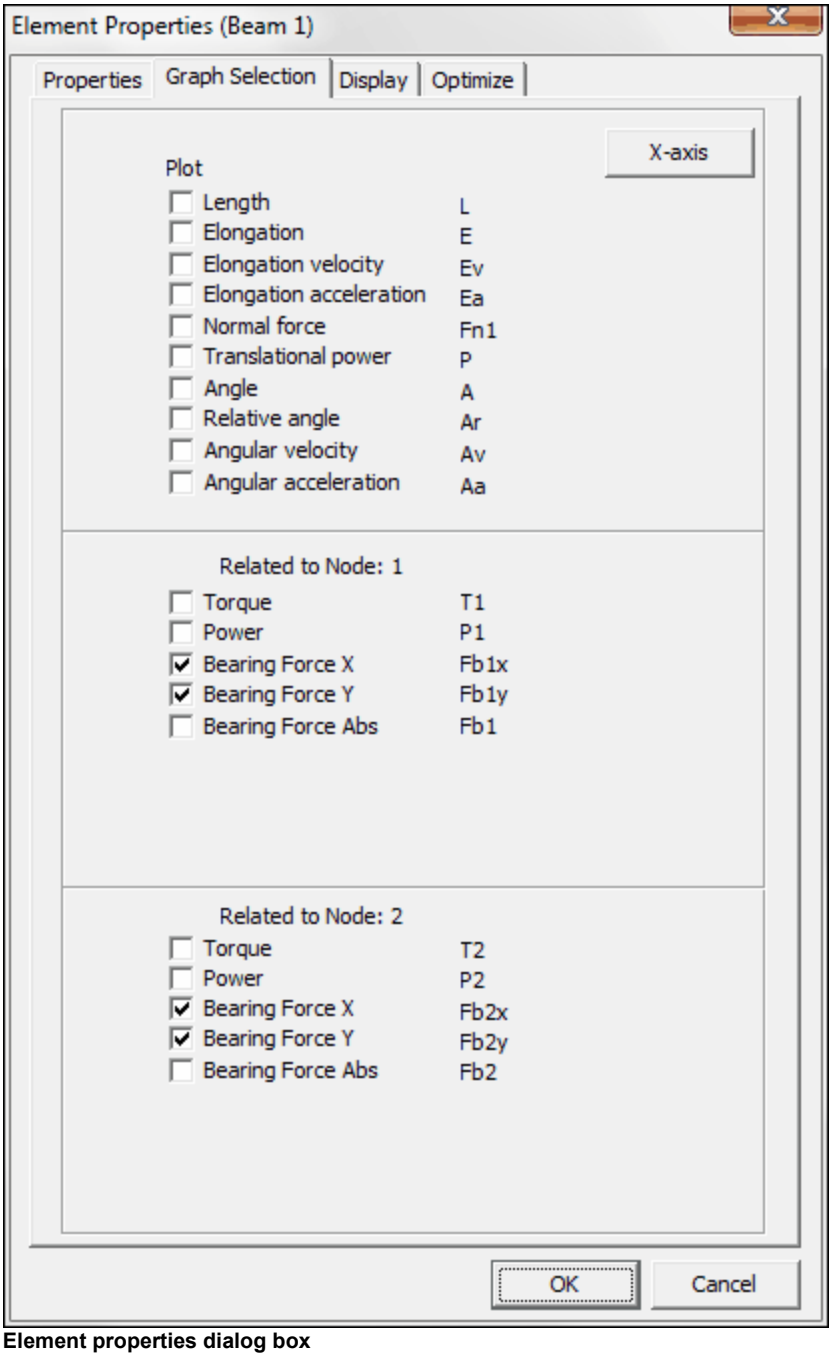
© 2024 ARTAS - Engineering Software



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

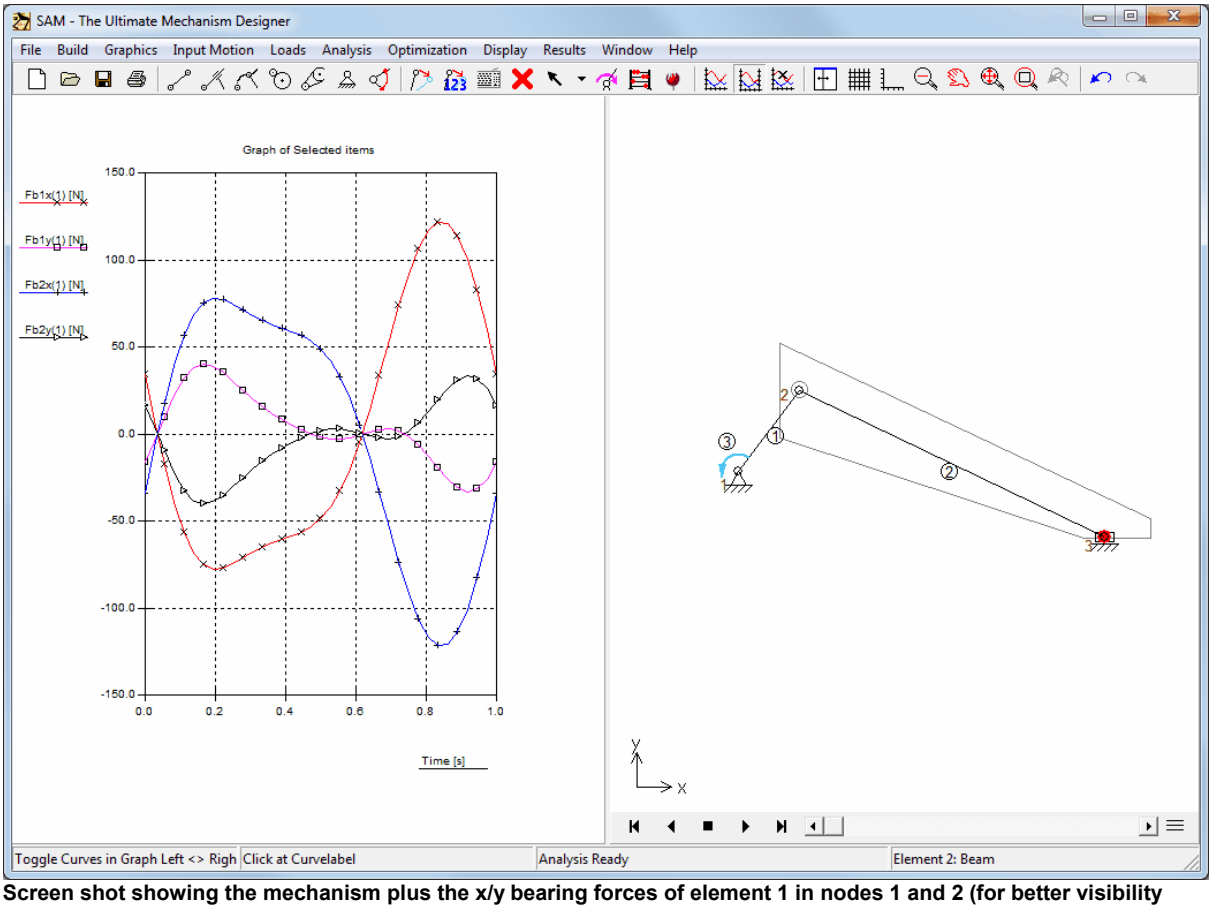
Next we will look at the x/y bearing forces of element 1 at the nodes 1 and two. In order to do so we need to make the following selection of results:

© 2024 ARTAS - Engineering Software



Element properties dialog box

© 2024 ARTAS - Engineering Software



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.

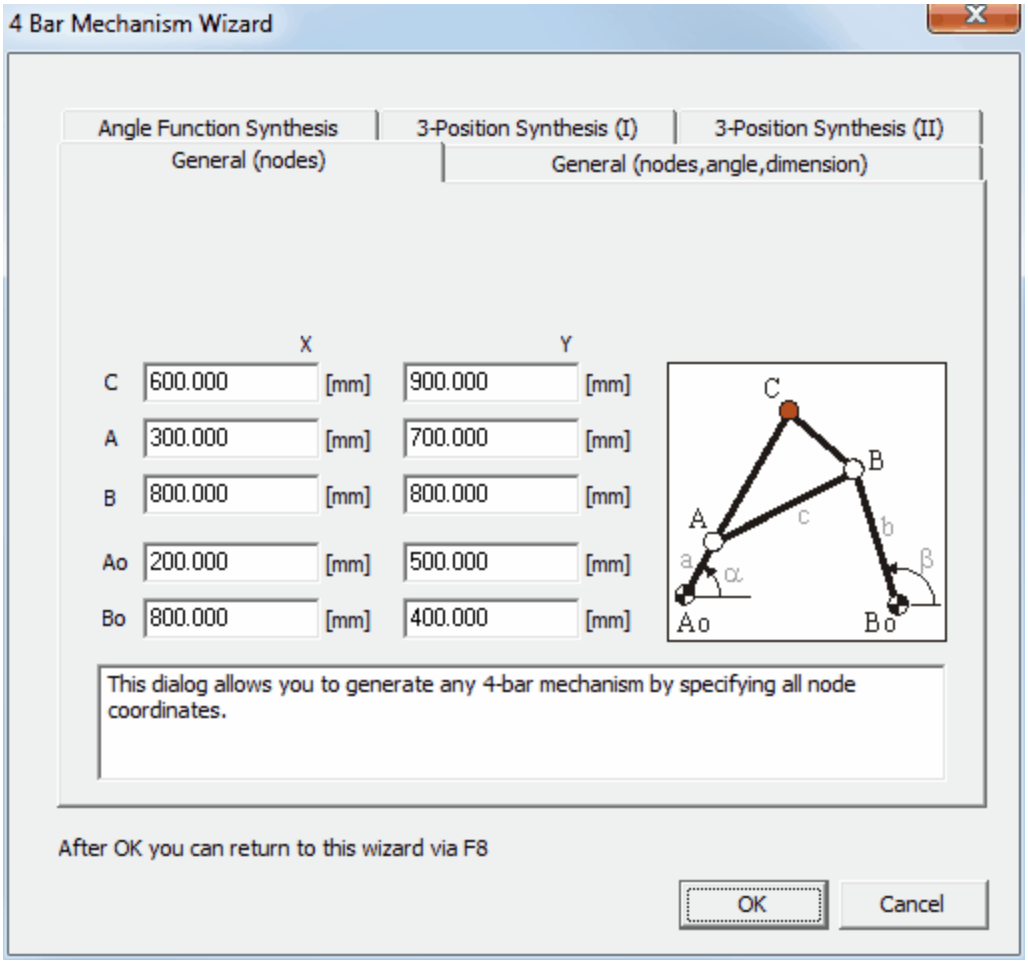
4.2 Using the Design Wizard to create a mechanism

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 fullfilled by a 4-bar mechanism.

© 2024 ARTAS - Engineering Software

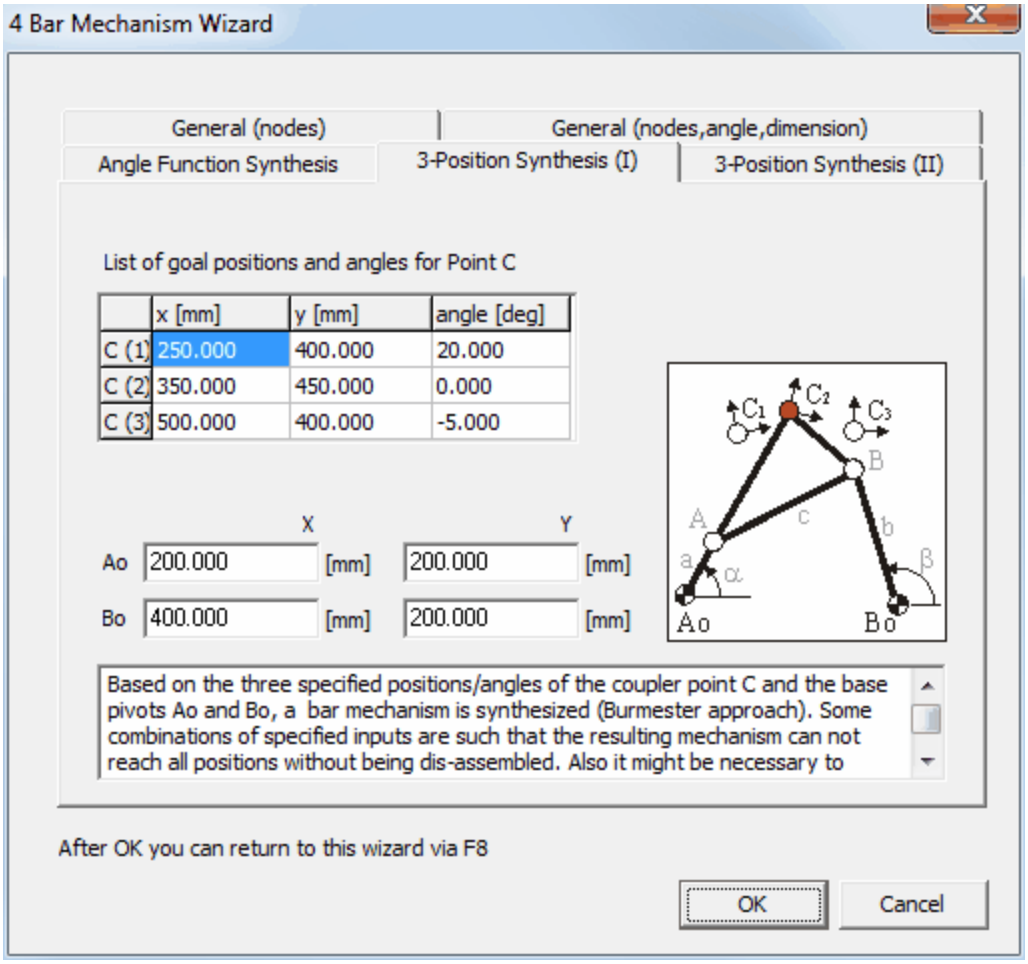


Design Wizard "4-Bar Mechanism"

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.

© 2024 ARTAS - Engineering Software



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

Press OK to accept the current values

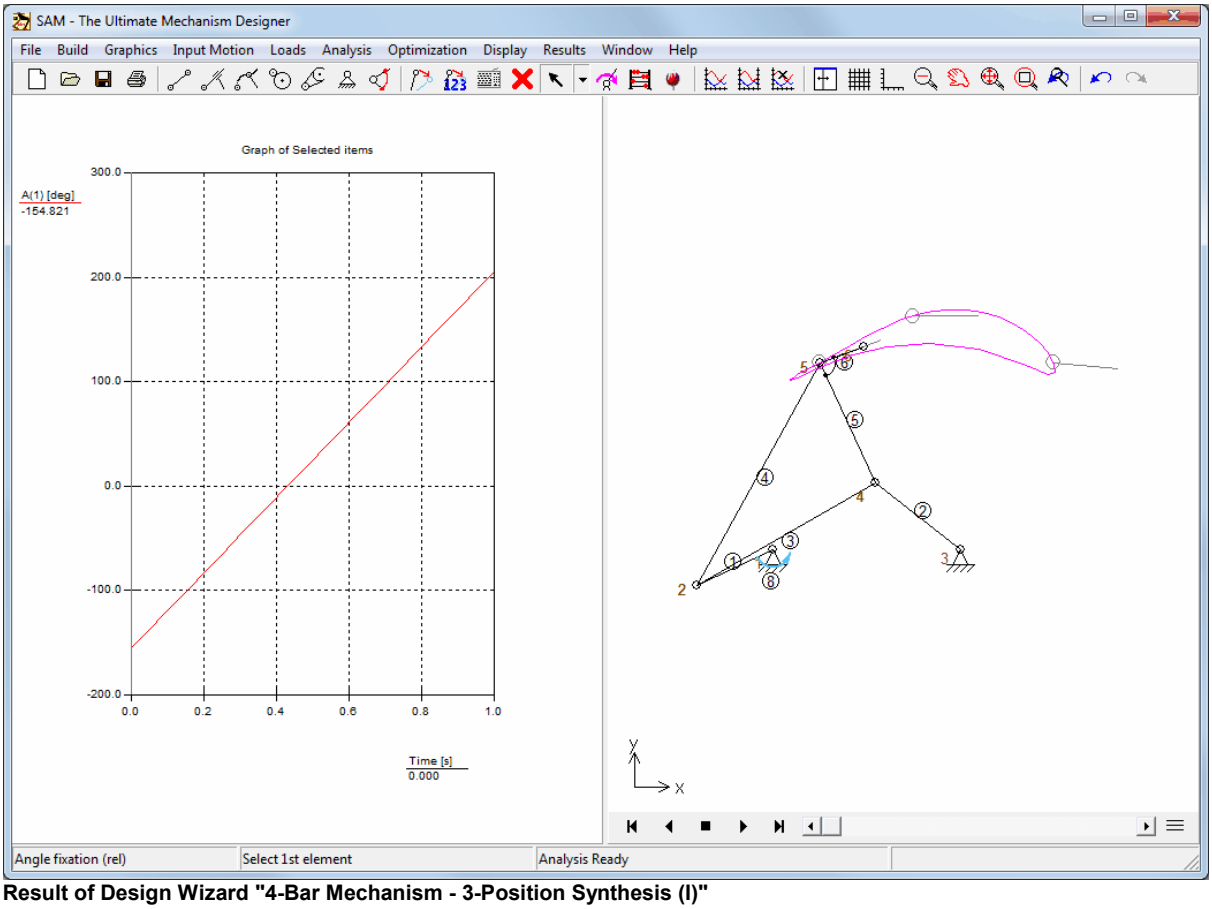
Based on the three specified positions/angles of the coupler point C and the base pivots Ao and Bo, a 4-bar mechanism is synthesized, analyzed (based on a certain input motion) and the path of the coupler point (+ the 3 specified goal positions/angles) is displayed in the mechanism window. Initially,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

© 2024 ARTAS - Engineering Software



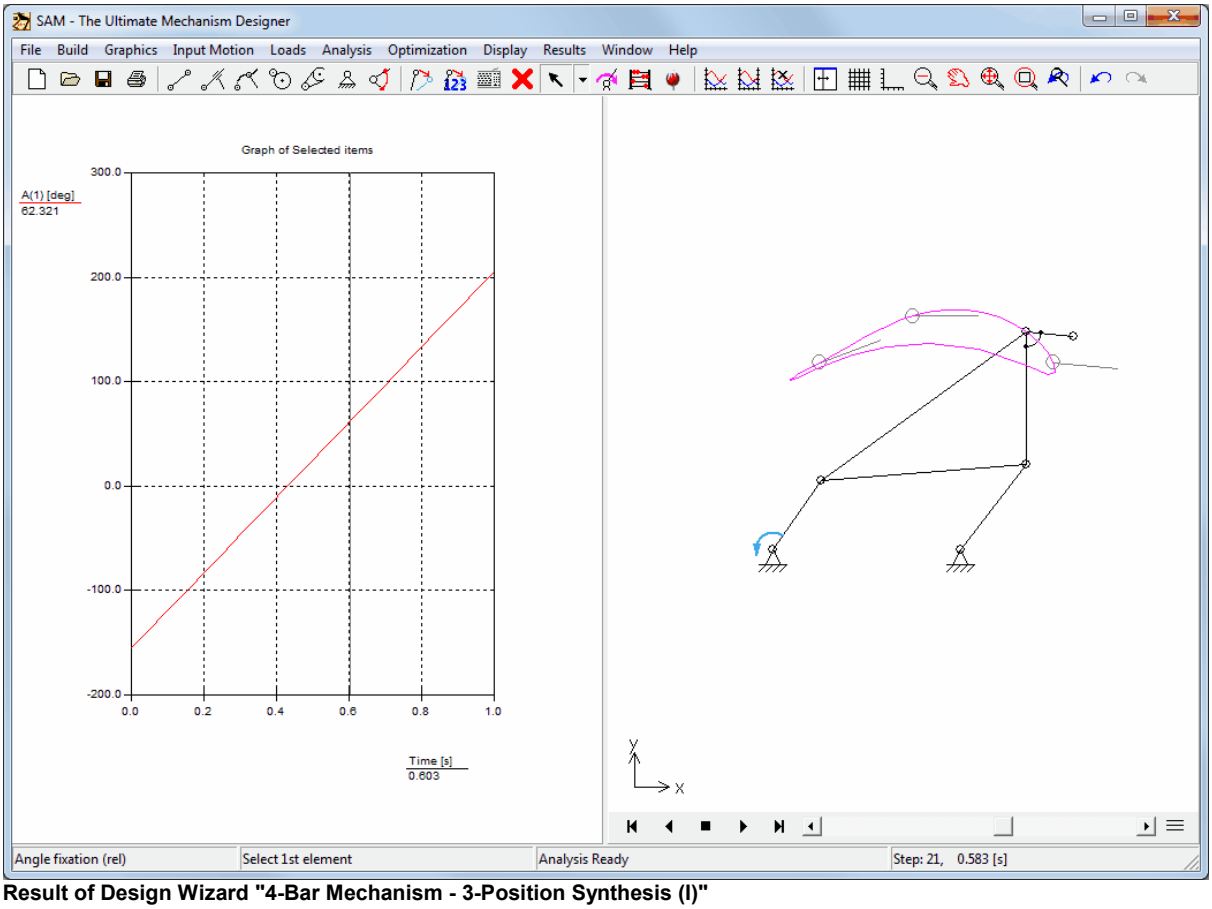
Result of Design Wizard "4-Bar Mechanism - 3-Position Synthesis (I)"

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.

© 2024 ARTAS - Engineering Software



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 Ao or Bo.

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 .

© 2024 ARTAS - Engineering Software

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/Analyis) an analyis 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 

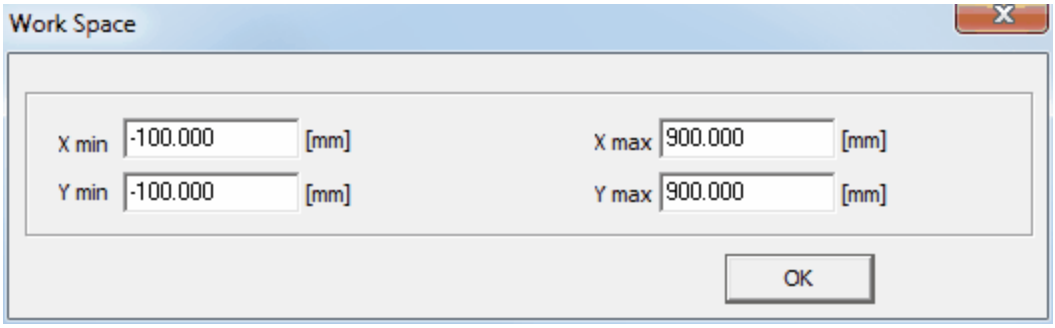
Choose Save from the File menu to save the project.

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.

© 2024 ARTAS - Engineering Software



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.

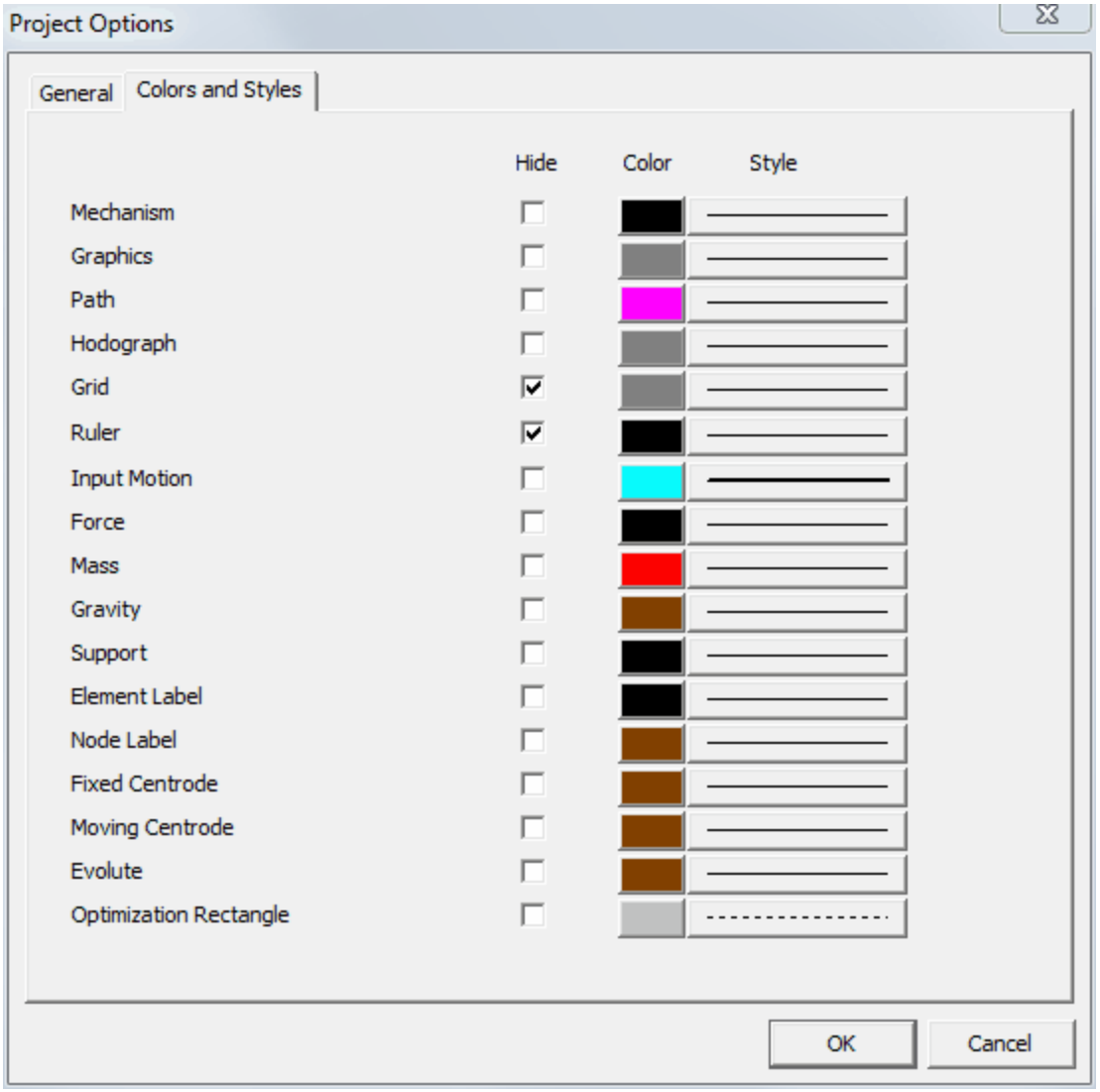
Maximize the mechanism-window by pressing the corresponding button.

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).

© 2024 ARTAS - Engineering Software

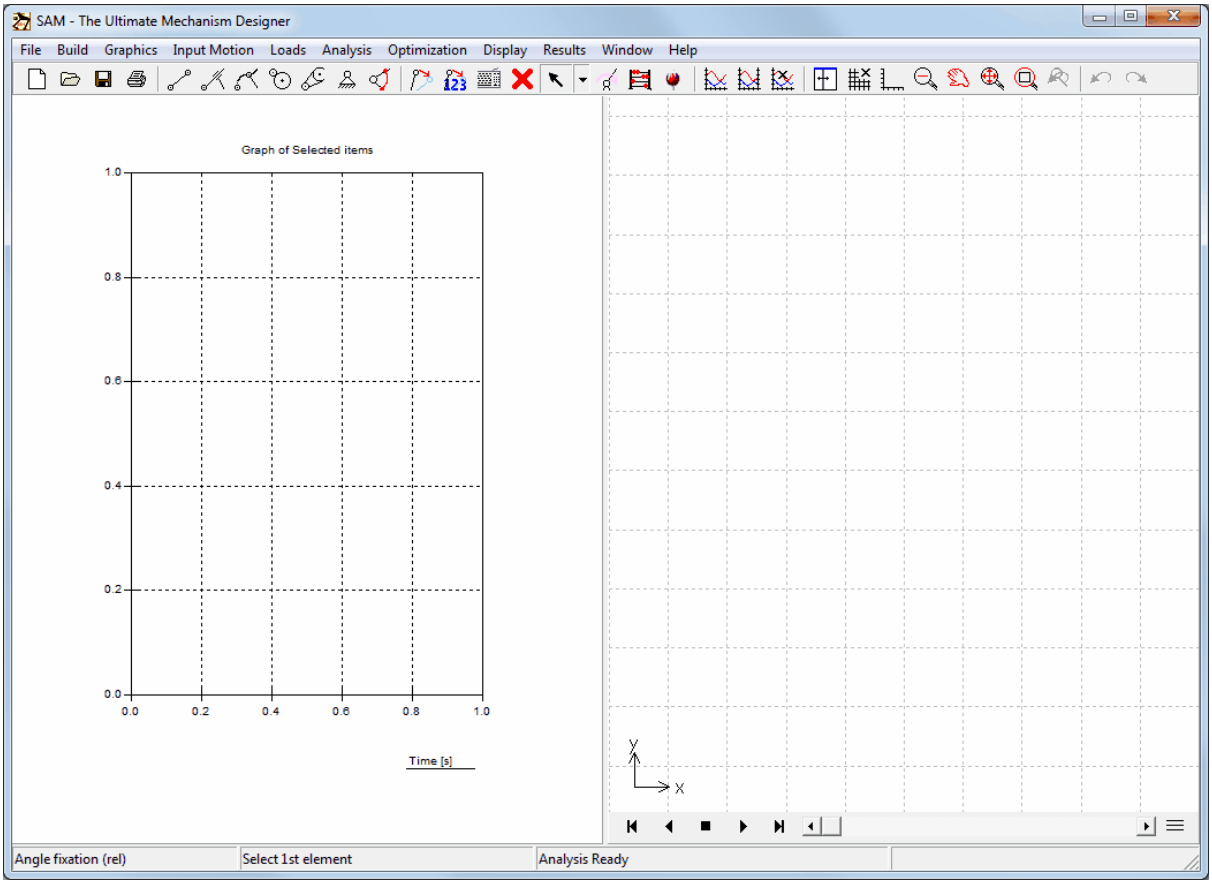


Display Options dialog box

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 :

© 2024 ARTAS - Engineering Software



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.

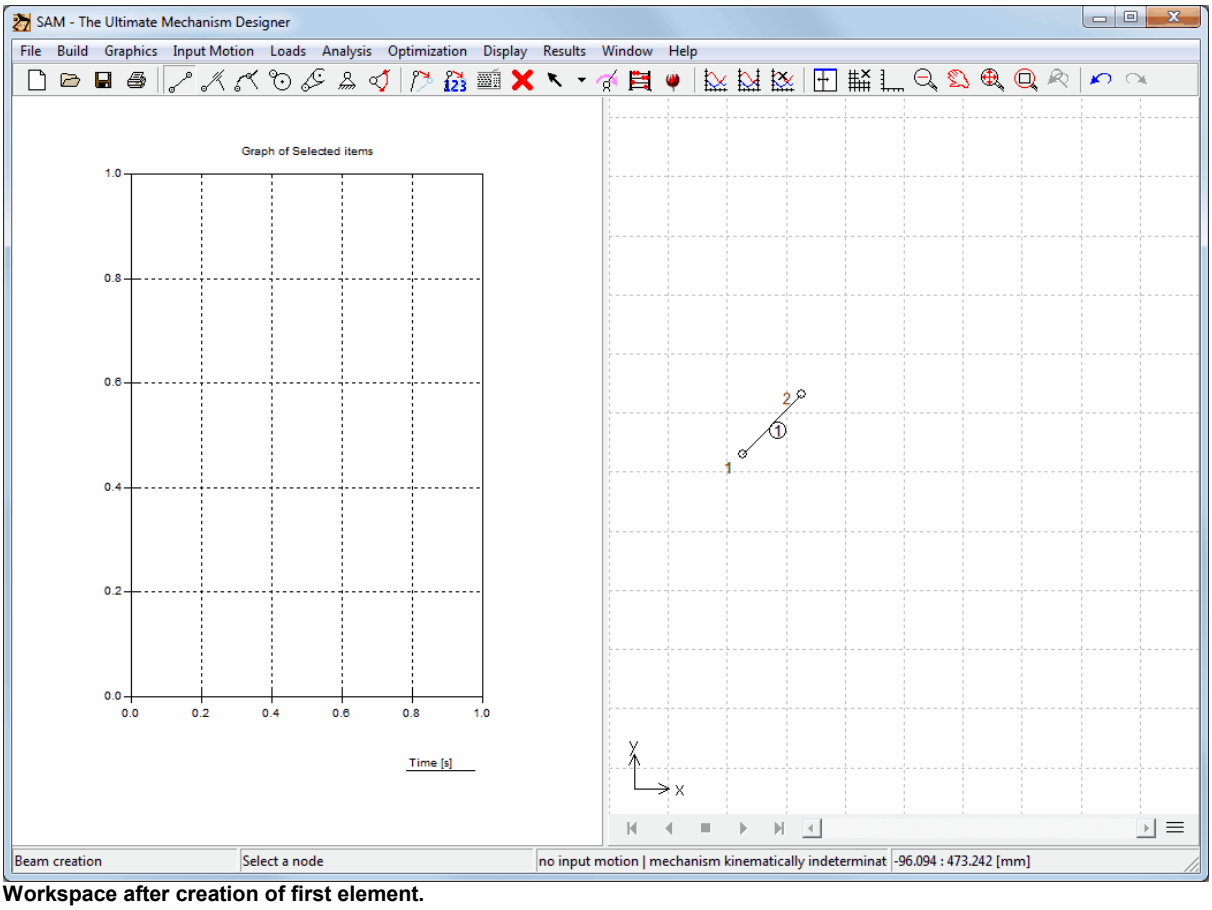
Move the mouse to the end point of the element and press the left button to finish creating a beam element.

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

© 2024 ARTAS - Engineering Software

the Build menu and click the element that you want to delete.

Your screen should now look similar to this :



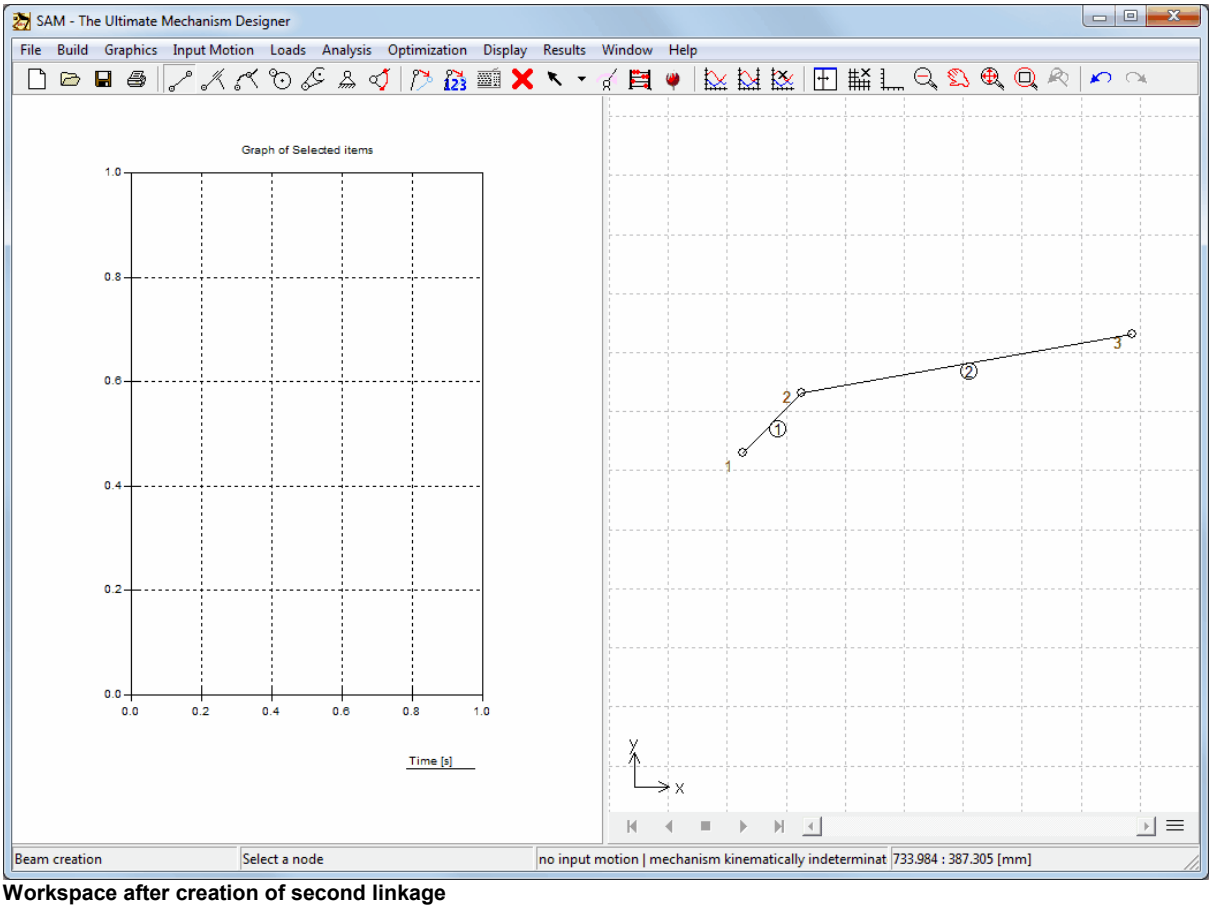
Workspace after creation of first element.

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.

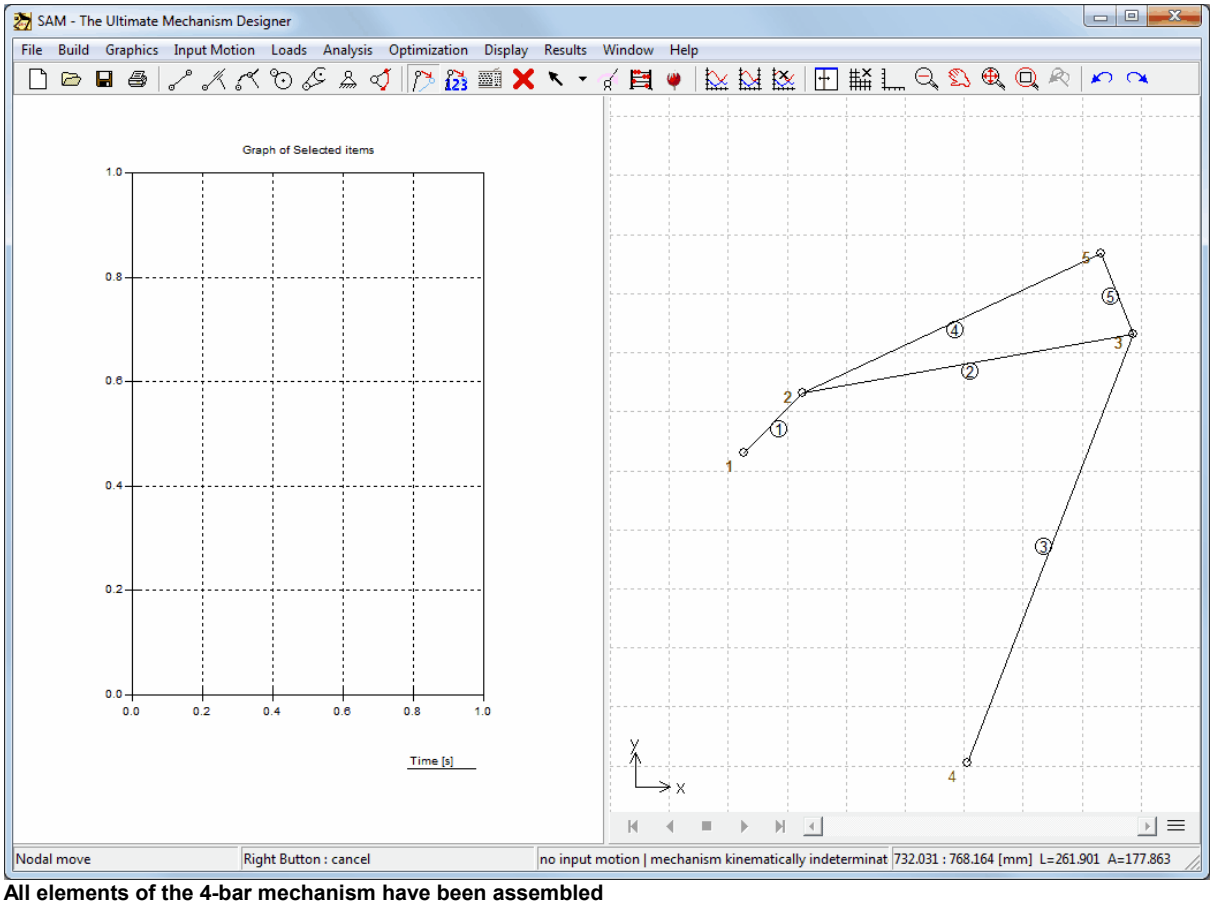
© 2024 ARTAS - Engineering Software



Workspace after creation of second linkage

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

© 2024 ARTAS - Engineering Software



All elements of the 4-bar mechanism have been assembled

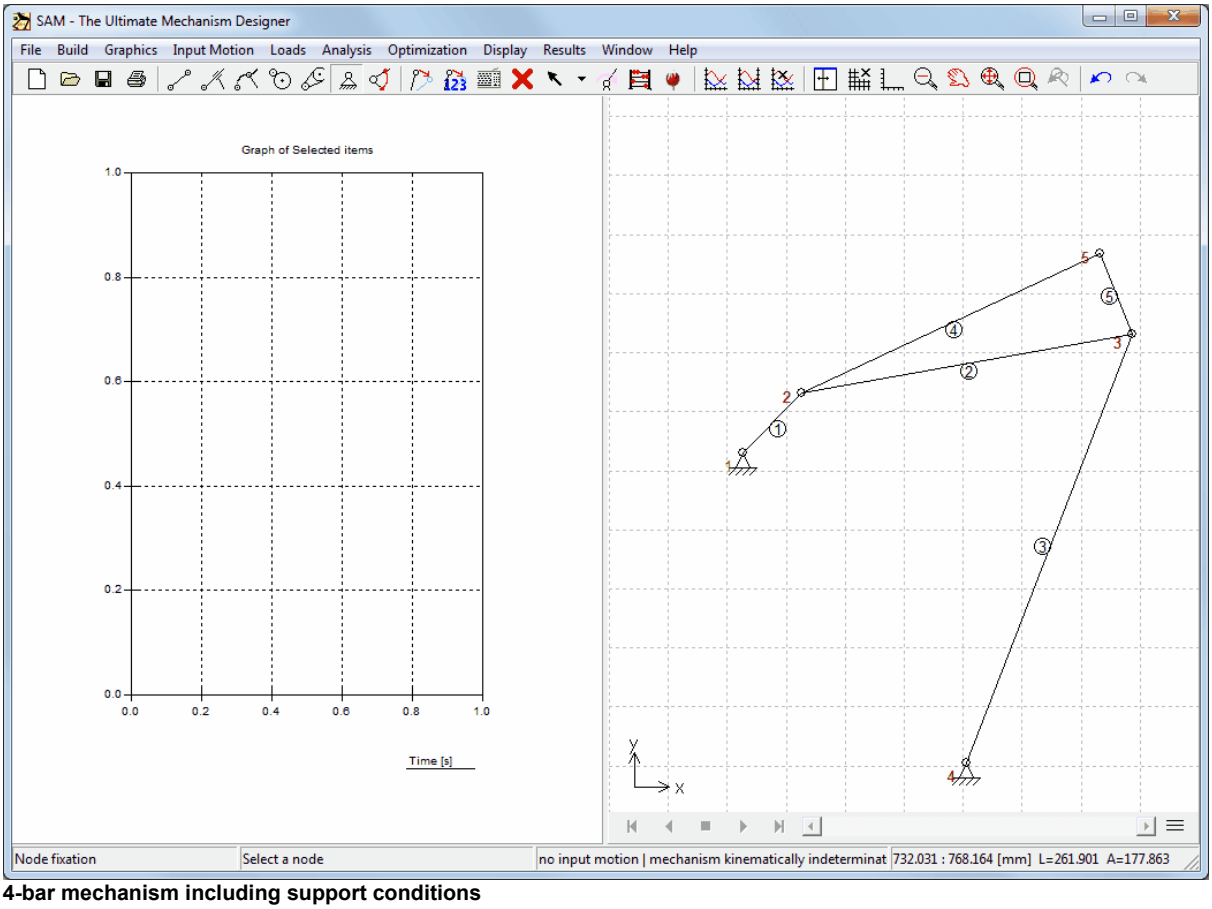
Select "Fix Node (x and/or y)" in the Build menu or click . Next, click on node 1 and move around the node until the xy-support condition is displayed. Then press the left mouse button again.

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.

© 2024 ARTAS - Engineering Software

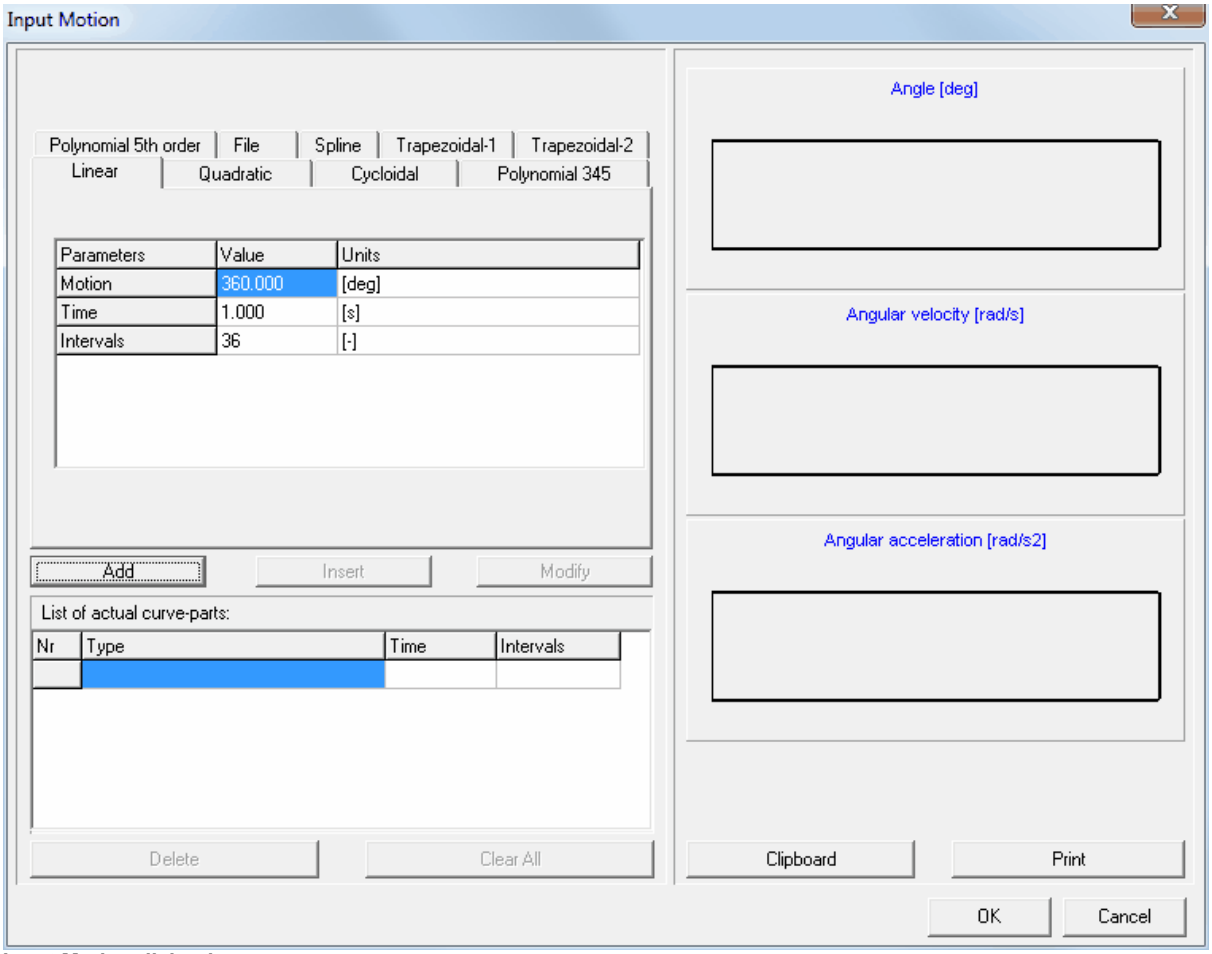


4-bar mechanism including support conditions

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.

© 2024 ARTAS - Engineering Software

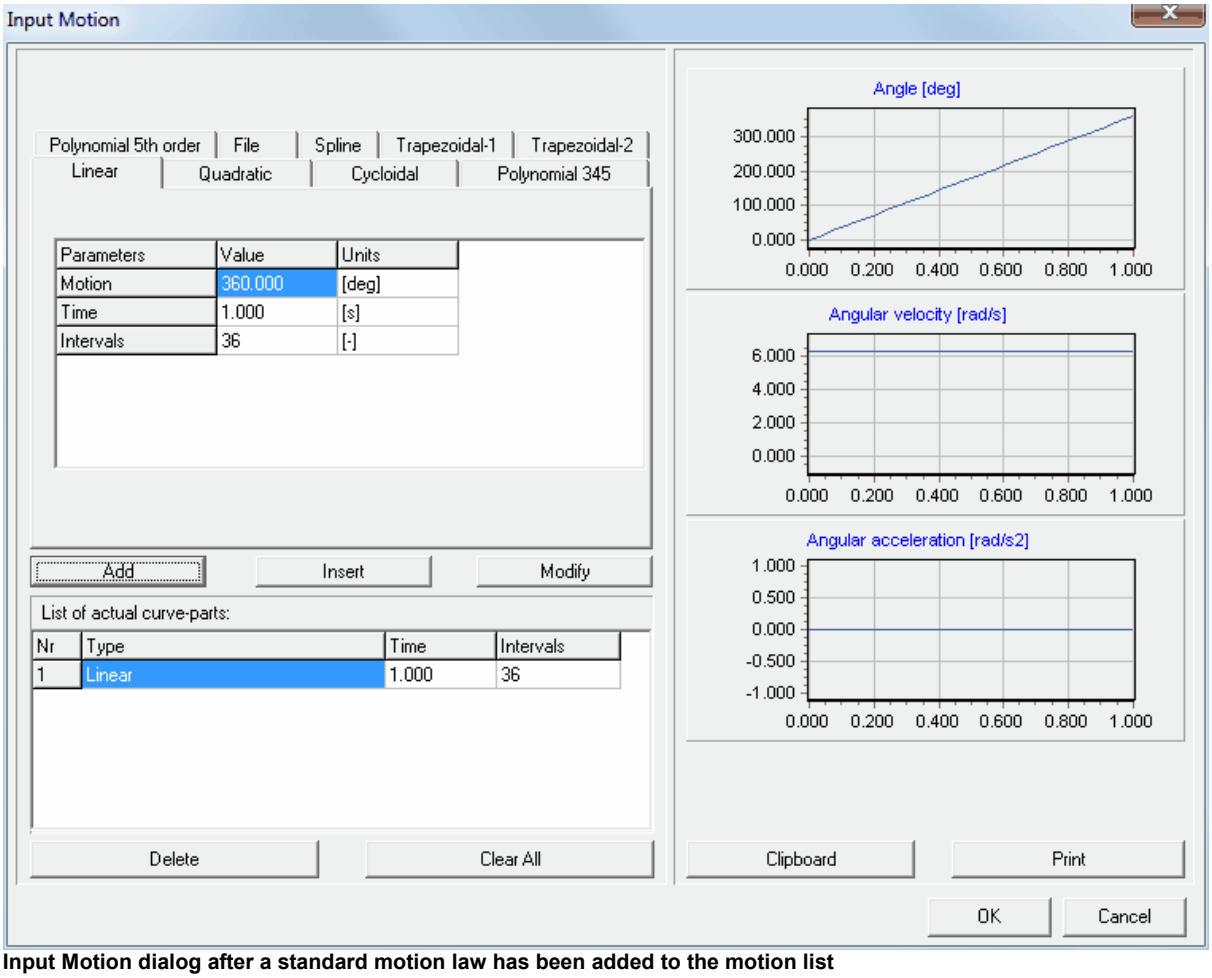


Input Motion dialog box

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.

© 2024 ARTAS - Engineering Software



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.

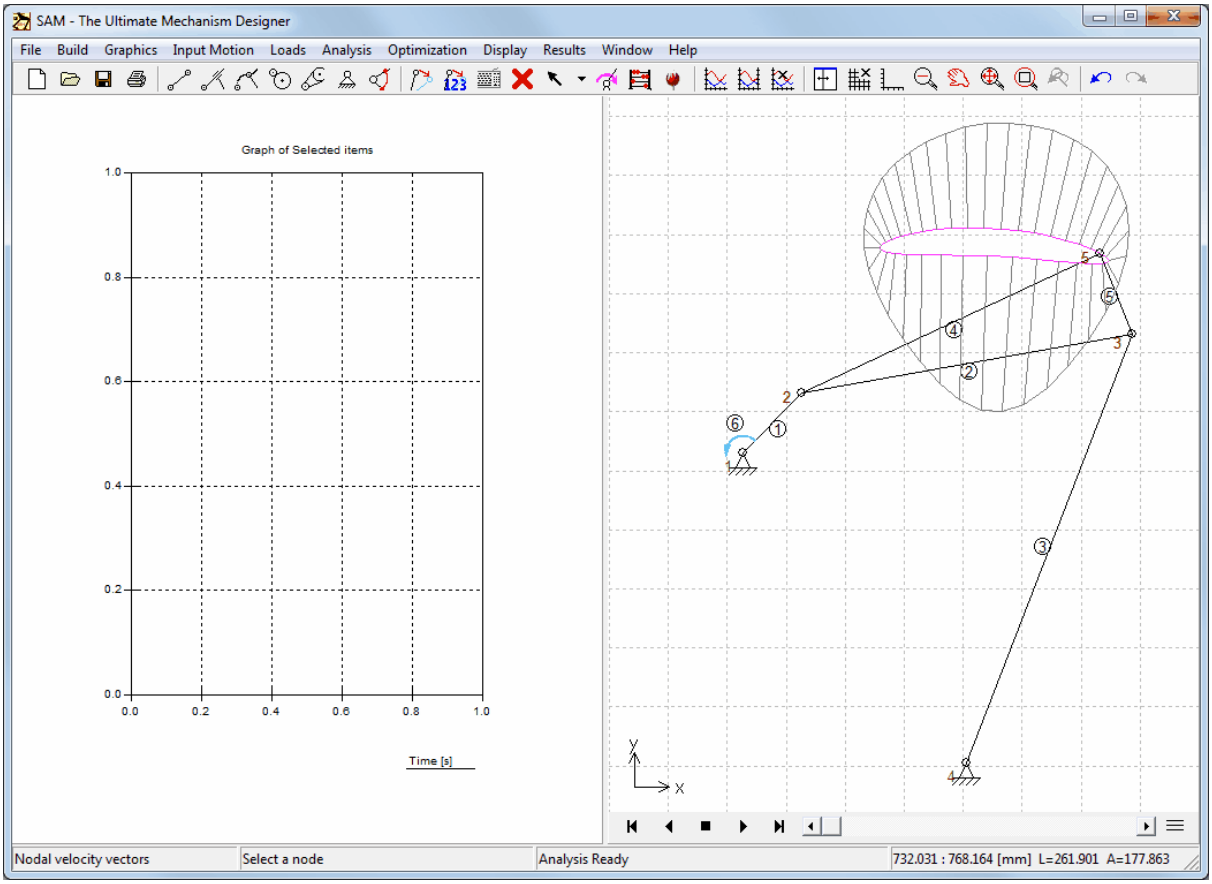
Select Path **in the Display menu and click the coupler point (node 5)**

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.

© 2024 ARTAS - Engineering Software



Path and velocity hodograph of coupler point

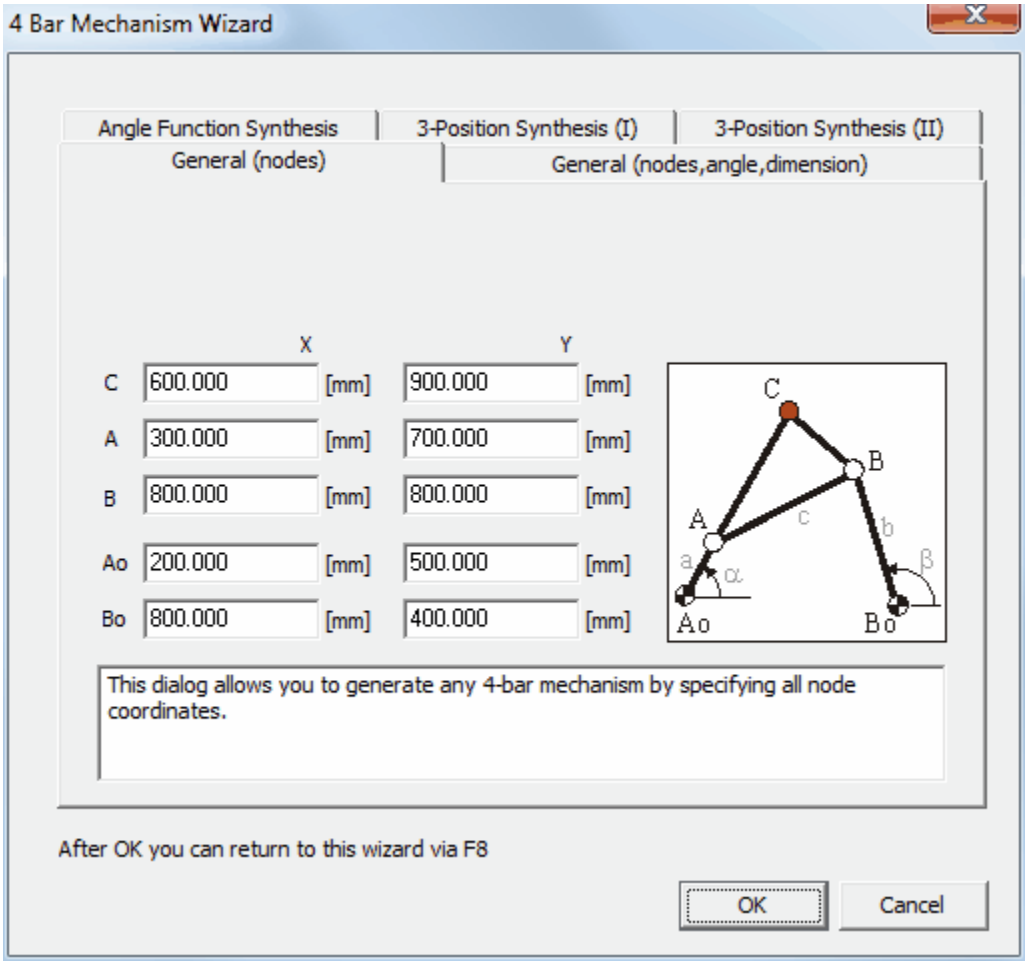
Chapter 5

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)

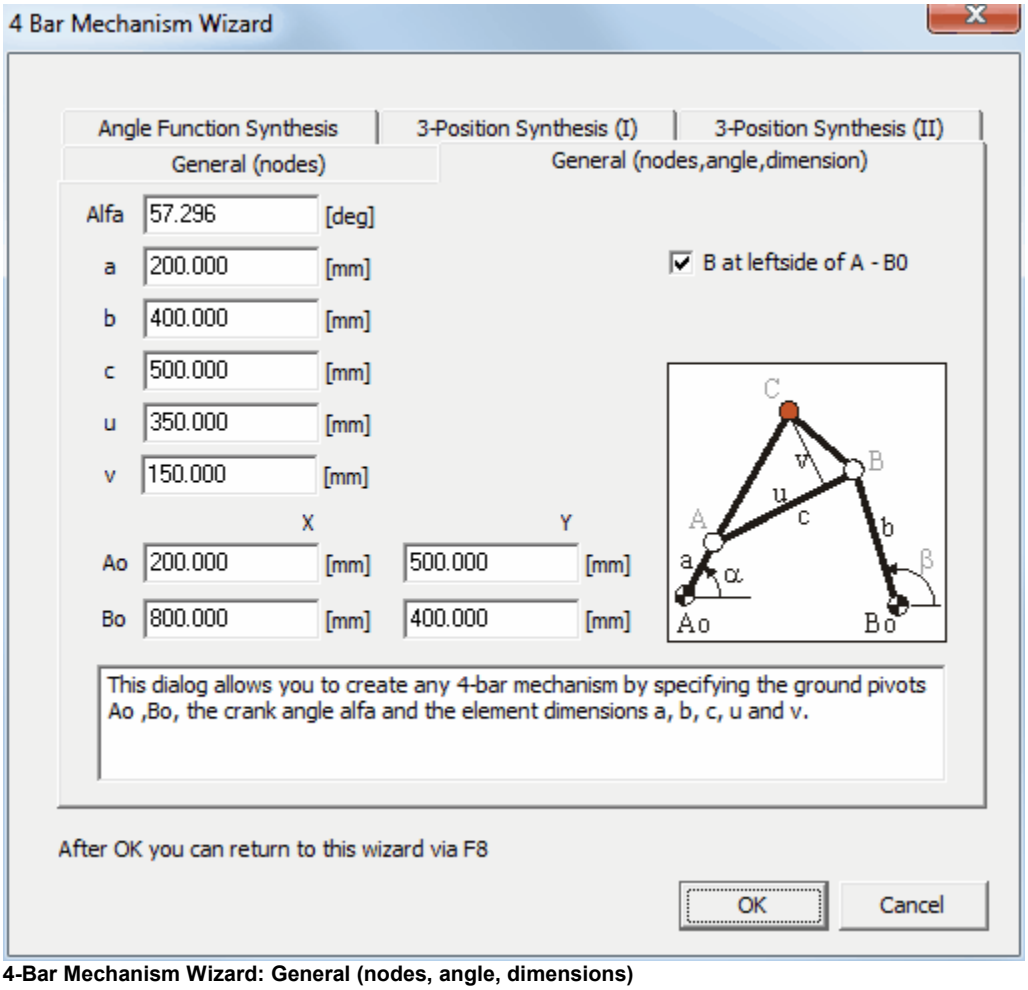


4-Bar Mechanism Wizard: General (nodes)

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

© 2024 ARTAS - Engineering Software

5.1.2 General (nodes, angle, dimension)

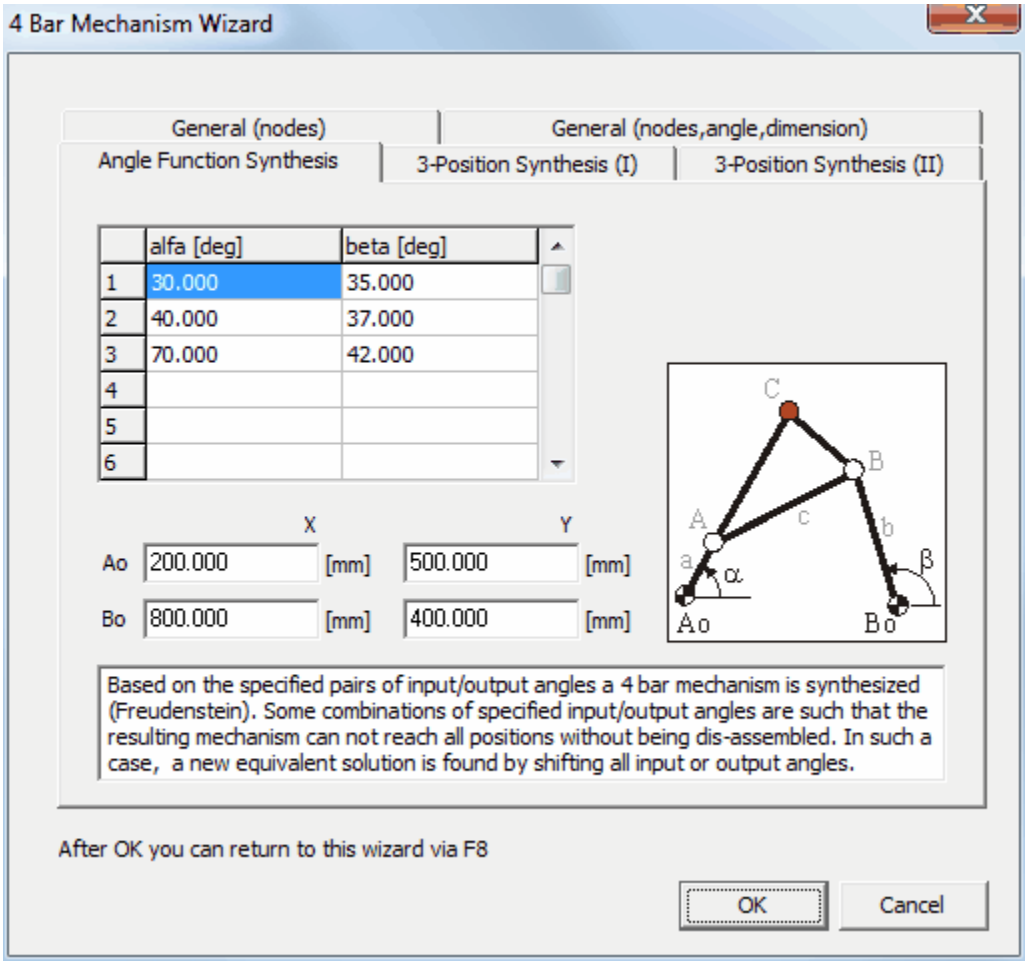


4-Bar Mechanism Wizard: General (nodes, angle, dimensions)

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.

© 2024 ARTAS - Engineering Software

5.1.3 Angle Function Synthesis

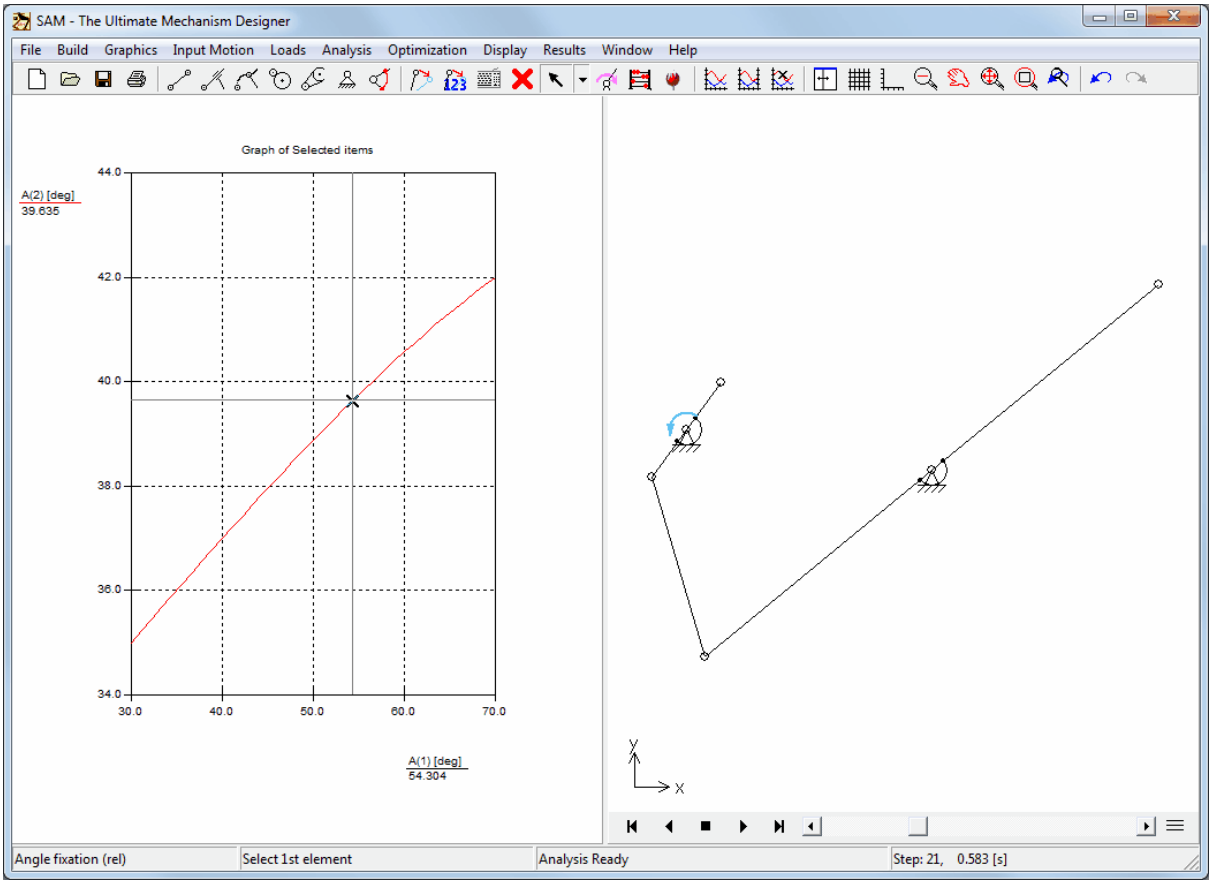


4-Bar Mechanism Wizard: Angle Function Synthesis

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.

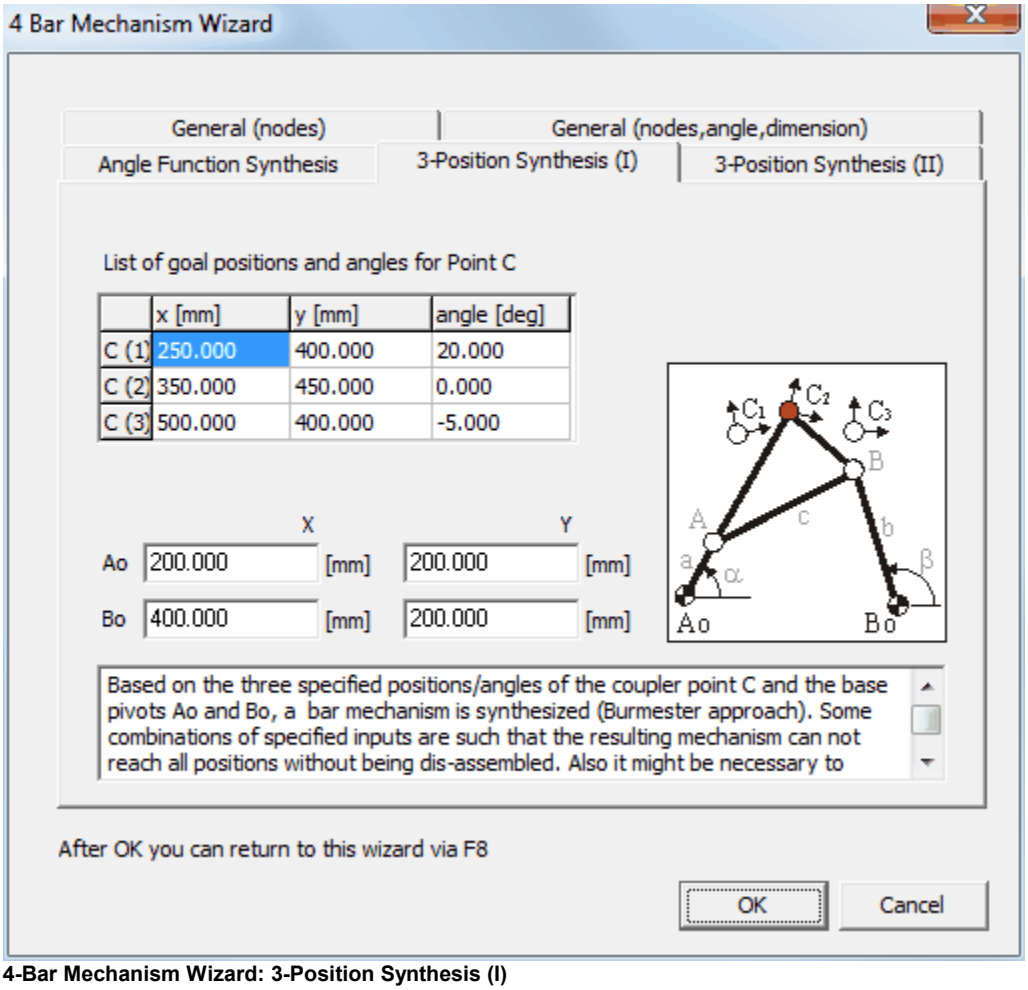
© 2024 ARTAS - Engineering Software



Result of angle function generation (based on default values)

© 2024 ARTAS - Engineering Software

5.1.4 3-Position Synthesis (I)



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

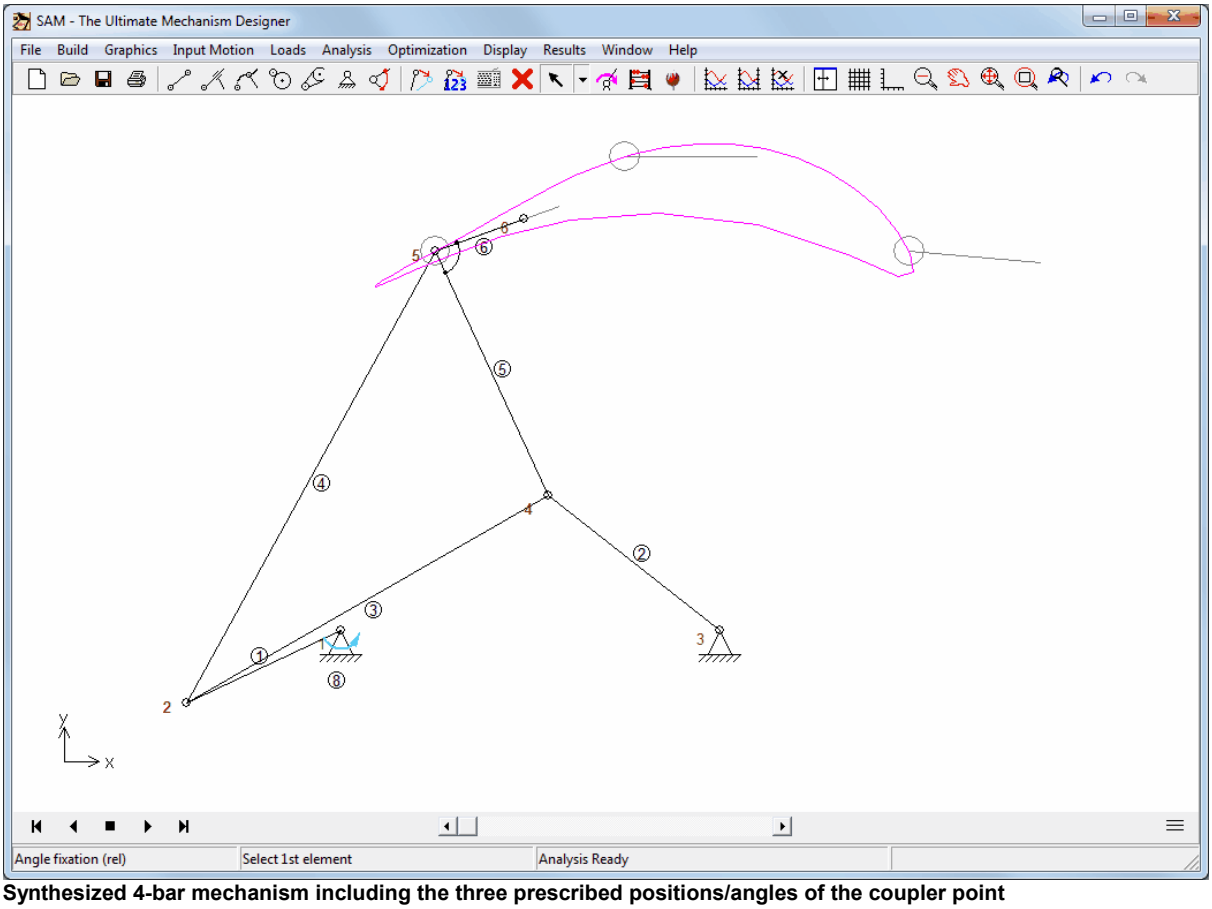
Based on three specified positions/angles of the coupler point C and the location of the base pivots Ao and Bo, a 4-bar mechanism is synthesized, which generates a motion of the coupler point that passes exactly through these three prescibed positions with the prescribed orientations of the coupler plane.

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 may 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.

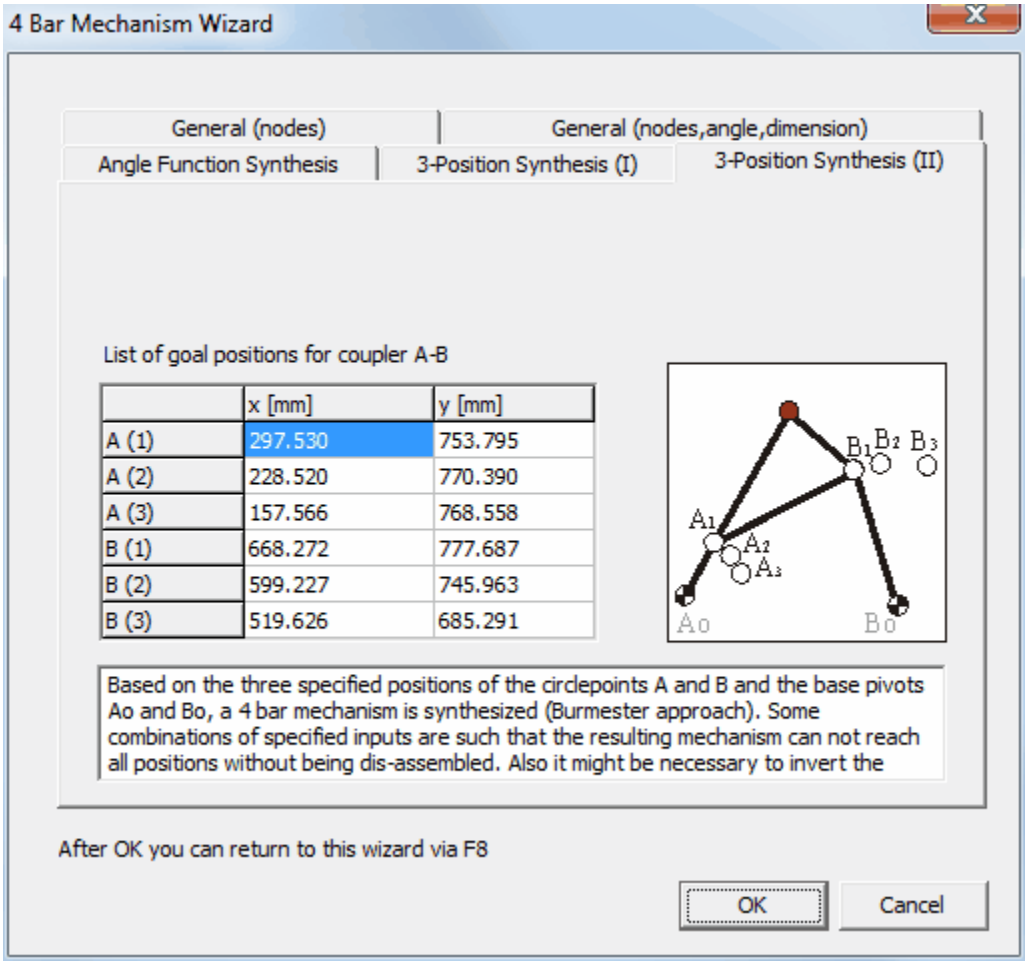
© 2024 ARTAS - Engineering Software



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

© 2024 ARTAS - Engineering Software

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 may design situations only the starting and end position are really fixed and it is tolerable to vary the middle position.

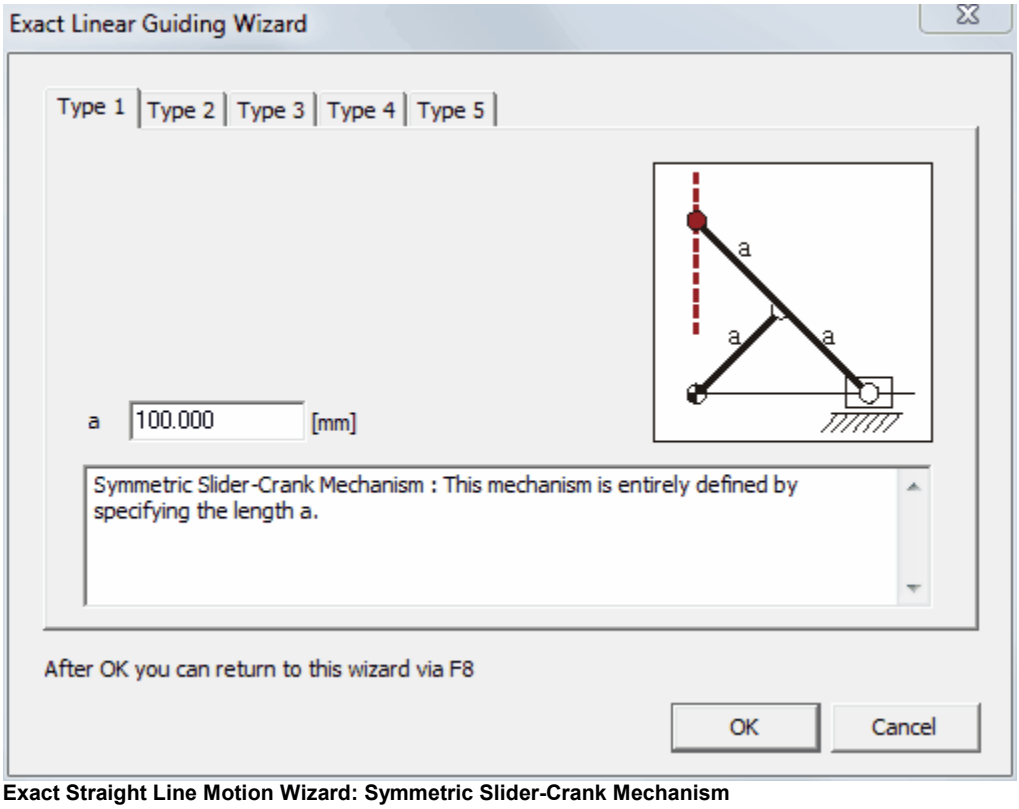
In case the coupler point passes through the prescribed positions in the wrong order, one has to change the direction of the input motion.

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.

© 2024 ARTAS - Engineering Software

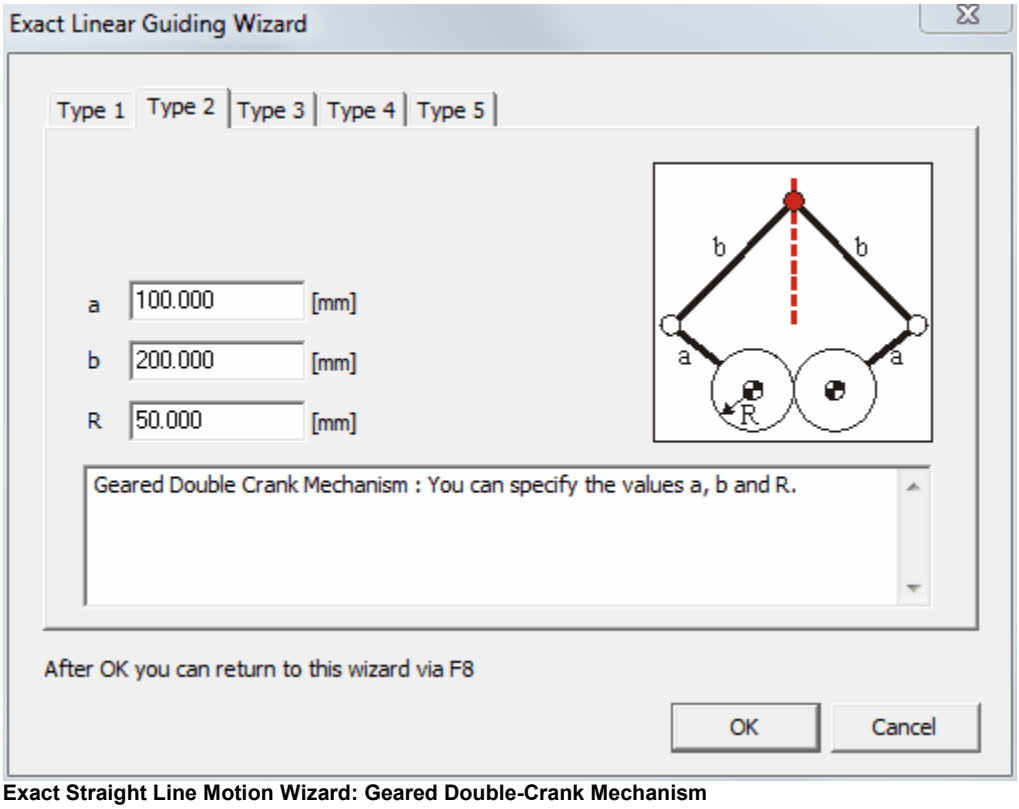
5.2.1 Symmetric Slider-Crank Mechanism



Exact Straight Line Motion Wizard: Symmetric Slider-Crank Mechanism

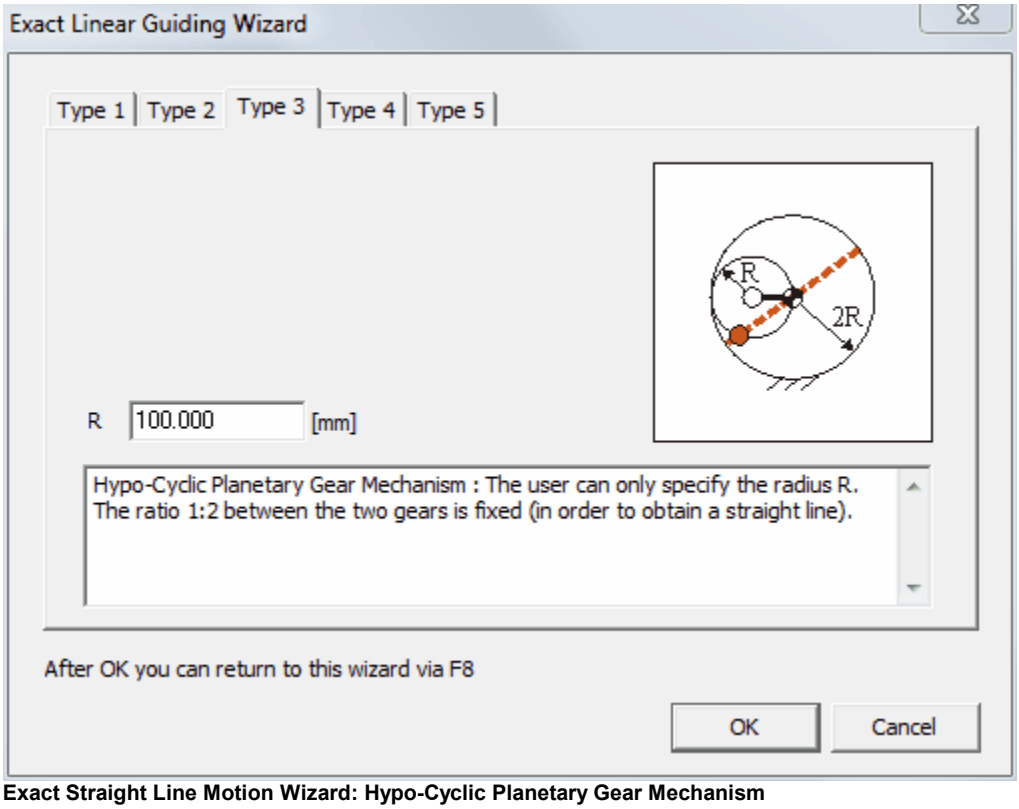
© 2024 ARTAS - Engineering Software

5.2.2 Geared Double-Crank Mechanism



Exact Straight Line Motion Wizard: Geared Double-Crank Mechanism

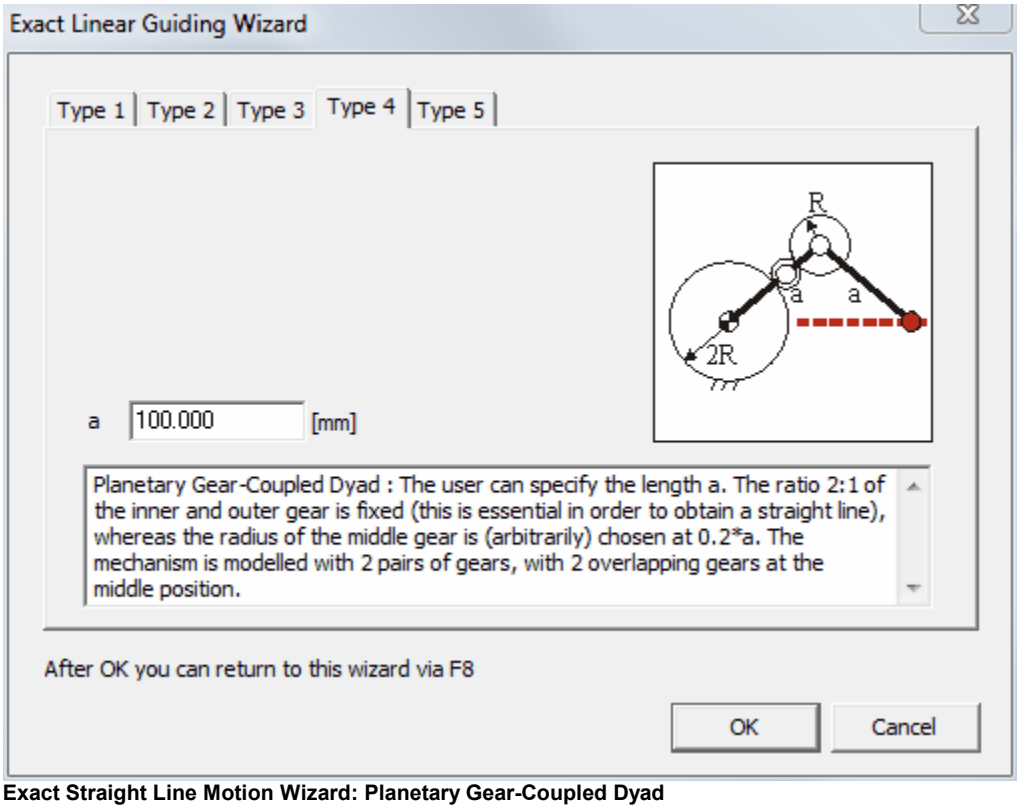
5.2.3 Hypo-Cyclic Planetary Gear Mechanism



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

© 2024 ARTAS - Engineering Software

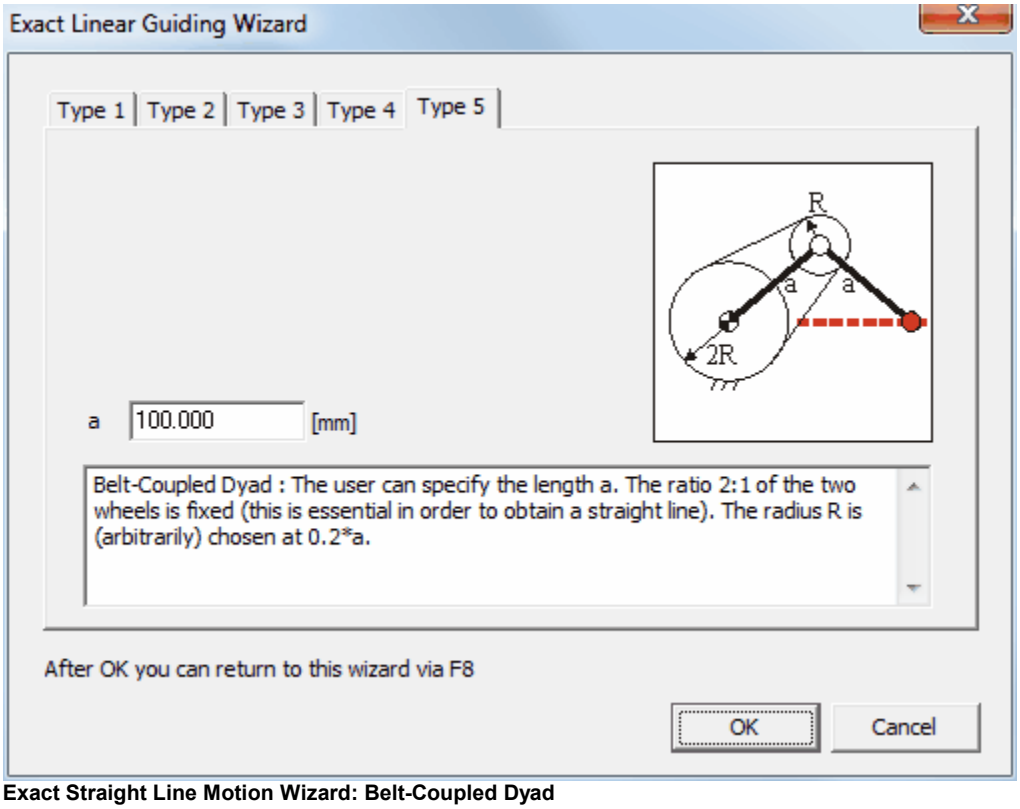
5.2.4 Planetary Gear-Coupled Dyad



Exact Straight Line Motion Wizard: Planetary Gear-Coupled Dyad

© 2024 ARTAS - Engineering Software

5.2.5 Belt-Coupled Dyad



Exact Straight Line Motion Wizard: Belt-Coupled Dyad

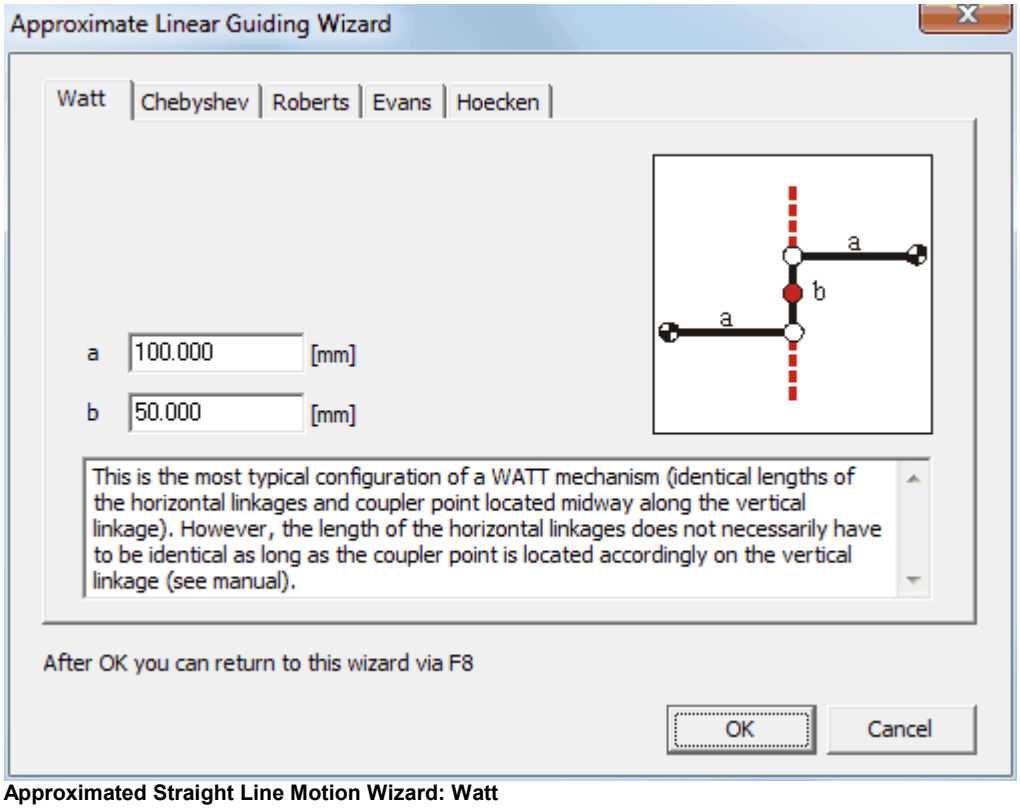
5.3 Approximated Straight Line Mechanism

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

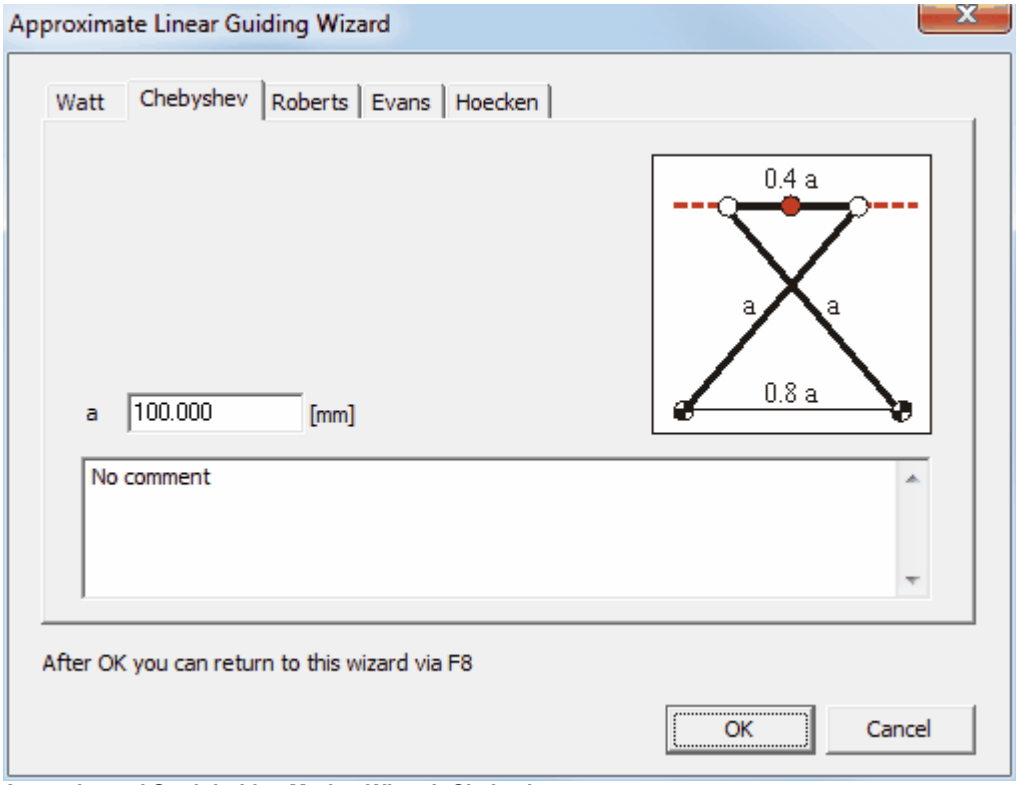
© 2024 ARTAS - Engineering Software

5.3.1 Watt



Approximated Straight Line Motion Wizard: Watt

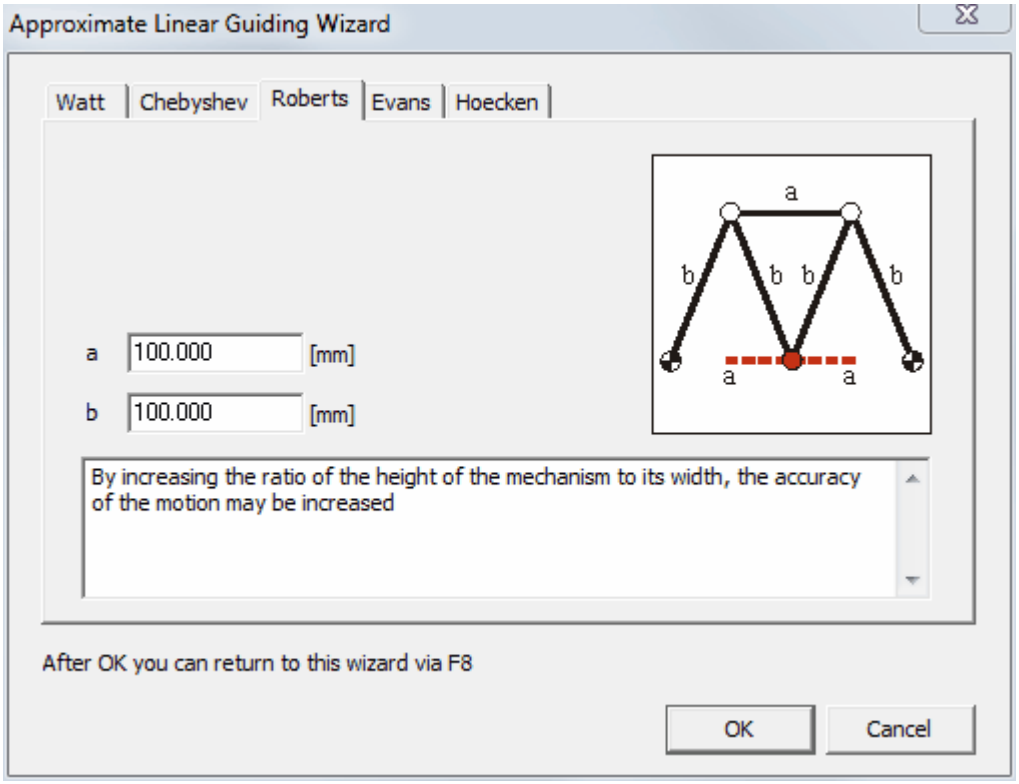
5.3.2 Chebyshev



Approximated Straight Line Motion Wizard: Chebyshev

© 2024 ARTAS - Engineering Software

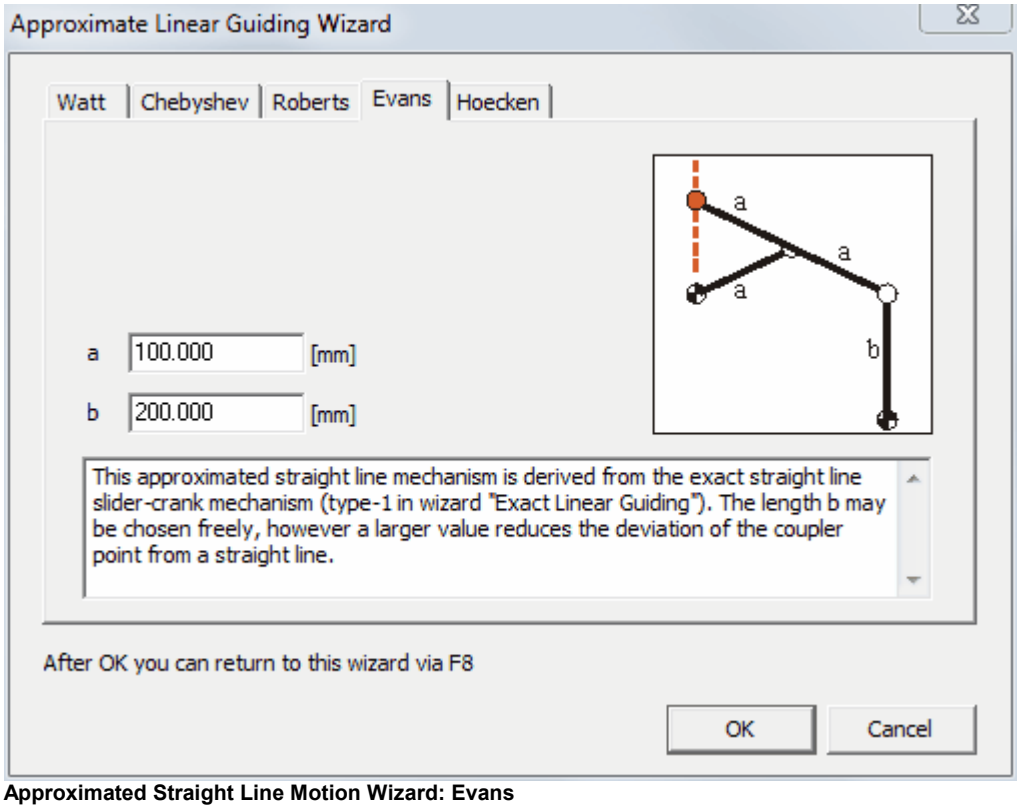
5.3.3 Roberts



Approximated Straight Line Motion Wizard: Roberts

© 2024 ARTAS - Engineering Software

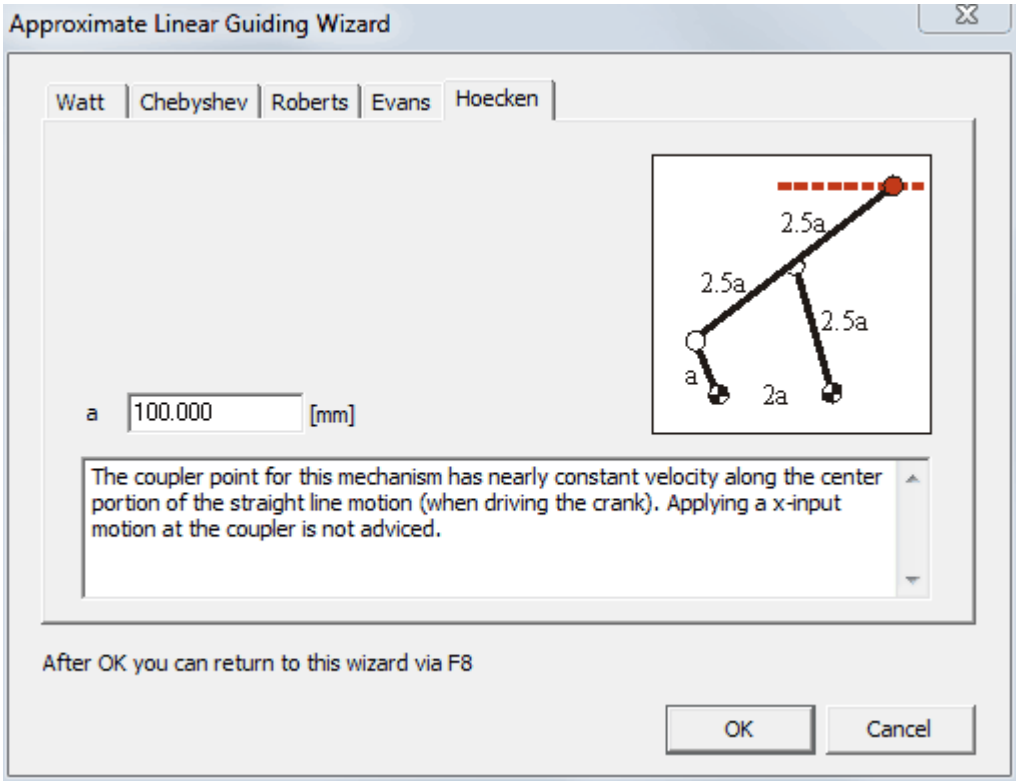
5.3.4 Evans



Approximated Straight Line Motion Wizard: Evans

© 2024 ARTAS - Engineering Software

5.3.5 Hoecken



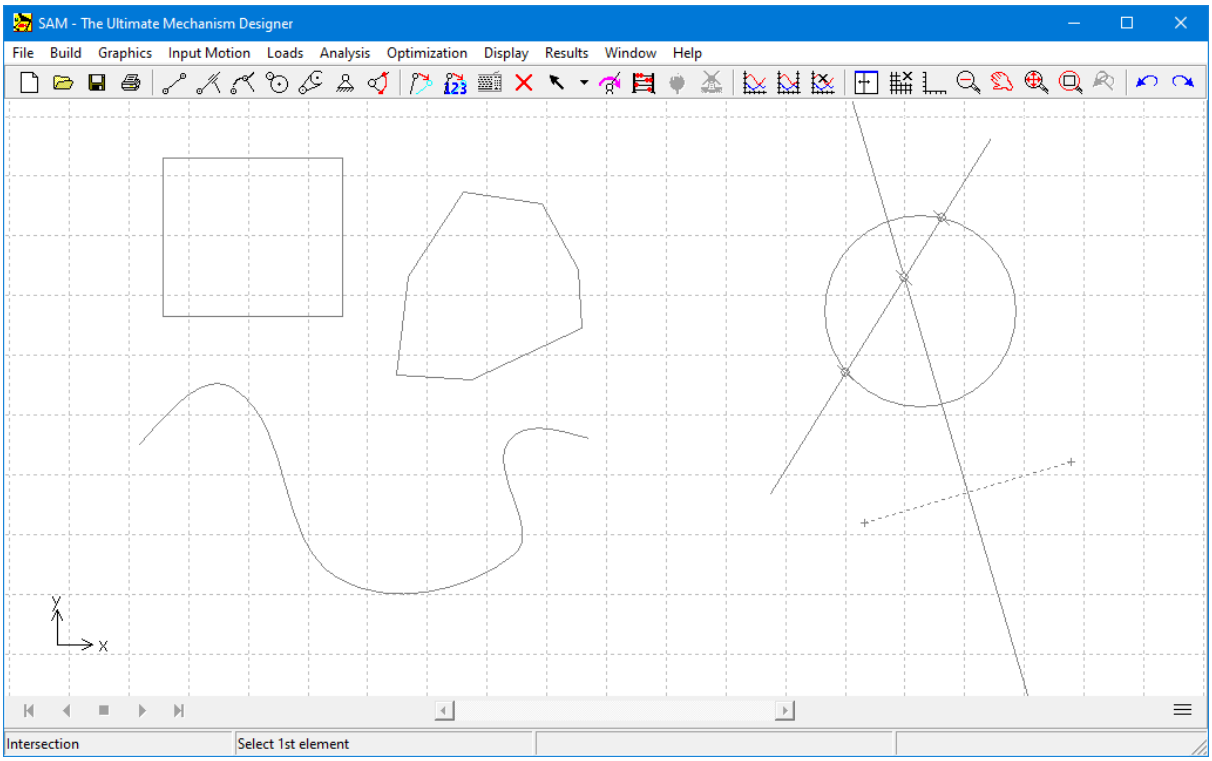
Approximated Straight Line Motion Wizard: Hoecken

Chapter 6

6 Graphics

SAM lets you draw/manipulate/group various graphics shapes. Available shapes are point, line, circle, rectangle, polyline and Bezier, which are supported by and construction aids like centerline and intersection.

- A shape can also be imported in DXF format (File>Import DXF).
- Copy/Past (Ctrl-C/Ctrl-V) allows shapes to be copied and pasted.
- 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.
- Points that have been created via intersections are dynamic, meaning that if the underlying shapes are transformed, either manually or during an animation, the coordinates of that point changes too.
- The function distance(object1,object2), that is available in the formula editor (Results > User-Defined ...), calculates the distance between various objects and also facilitates collision detection.



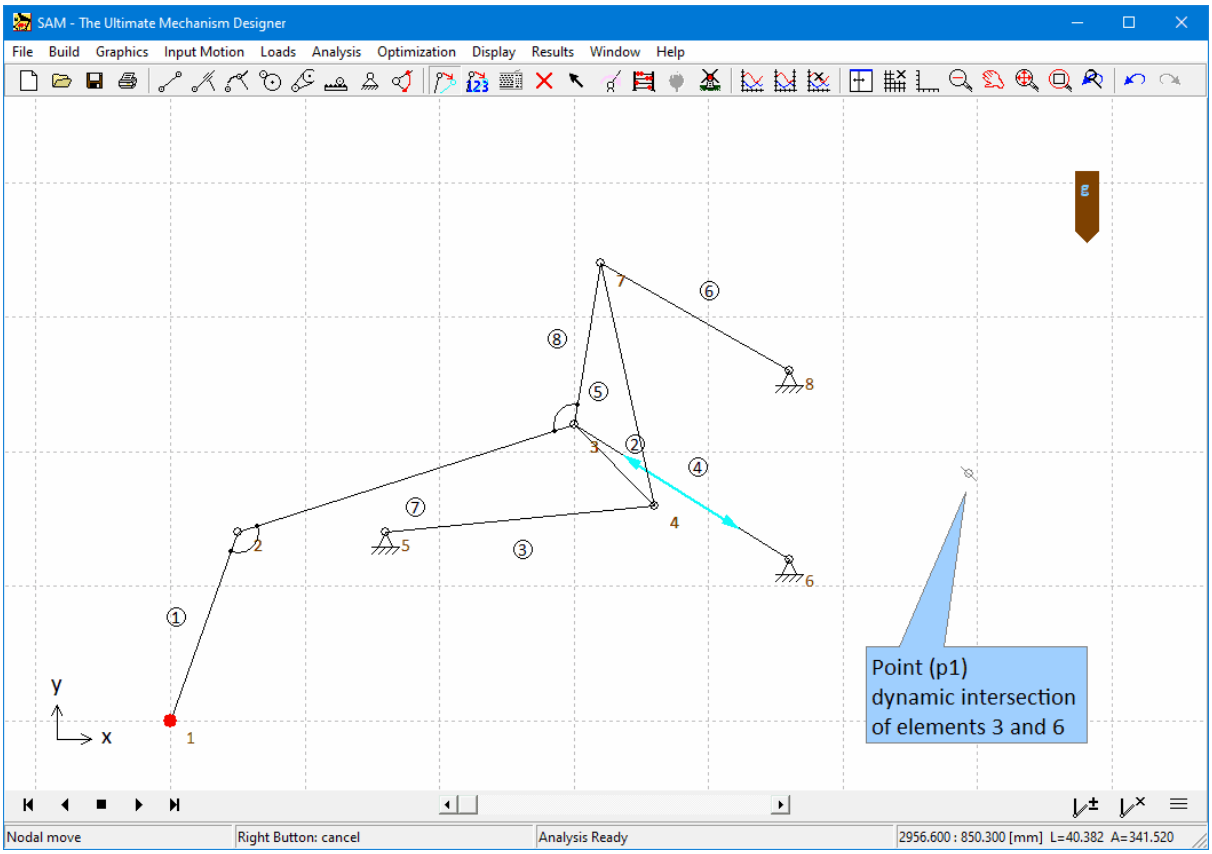
Available shapes (point, line, circle, rectangle, polyline, Bezier) and construction aids (centerline, intersection)

Dynamic Points

The possibility to construct points via the intersection of lines and even the intersection of centerlines has become dynamic. When moving the objects used in the creation of a point, the xy coordinates of that point are adapted automatically. That holds for the situation that an object is move in the reference position, but also when performing an animation.

In this way one can for example create point p1 as the intersection of beam 3 and beam 6, which constitutes the momentary center of rotation of the triangle formed by the triangular coupler. When the mechanism is animated and moves from one position to the next, the location of point p1 is adapted automatically.

© 2024 ARTAS - Engineering Software

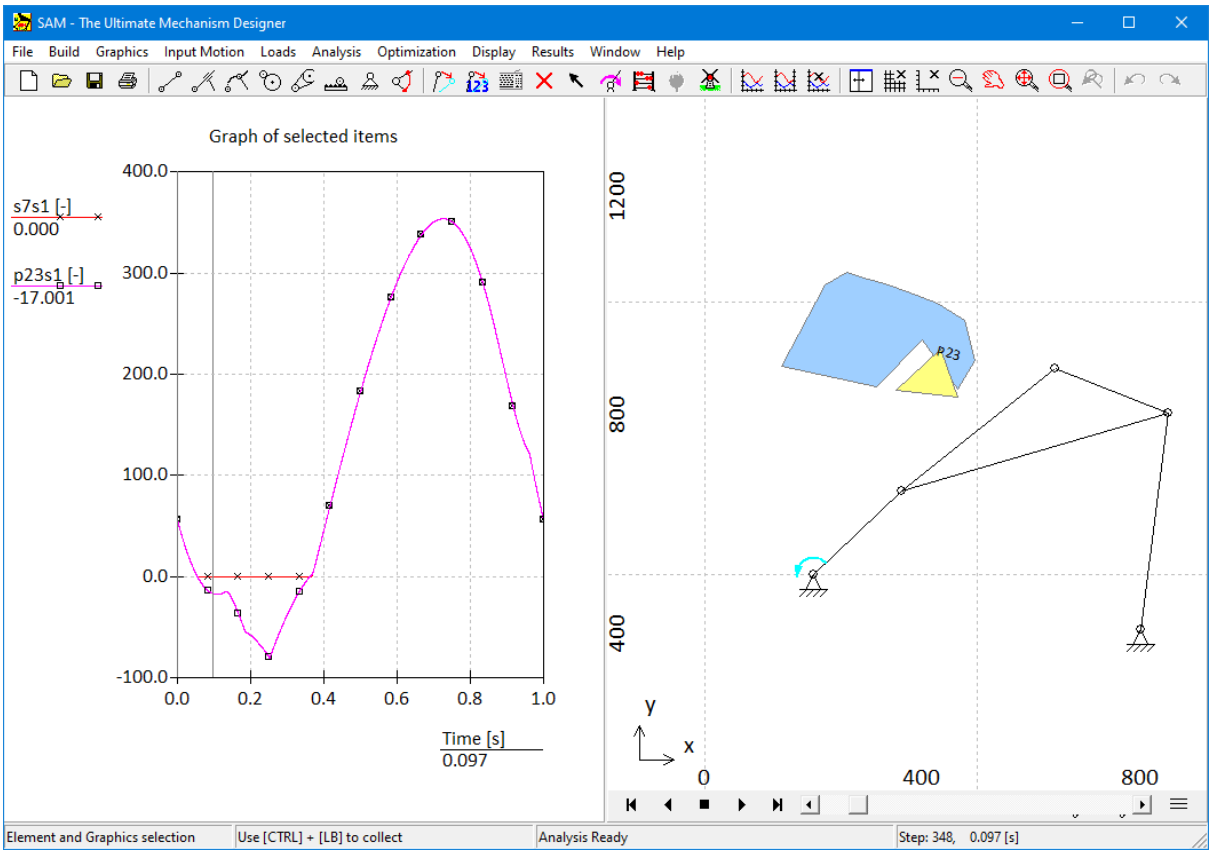


The point p1 is created as an intersection of the two (infinite) lines corresponding to the beam elements 3 and 6. The point is dynamic an moves during an animation.

Collision detection

The distance function can be used to detect collision between objects and depending on the type of objects even the penetration depth (or minimum retraction distance to avoid collision). In the following example the blue polygon is a stationary shape s1, whereas the yellow triangle (shape s7) is attached to the coupler. In the plot one sees the variable s7s1, which corresponds to the distance between the two shapes and the variable p23s1, which corresponds to the distance between point p23 on the triangle and the blue polygon. In case of two colliding shapes the distance is set to zero, whereas in the case of a point or node moving into a shape both the positive and negative distance (negative distance = penetration depth) is shown in the plot.

© 2024 ARTAS - Engineering Software



Example of collision detection via use of the distance function.

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 indicated 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.

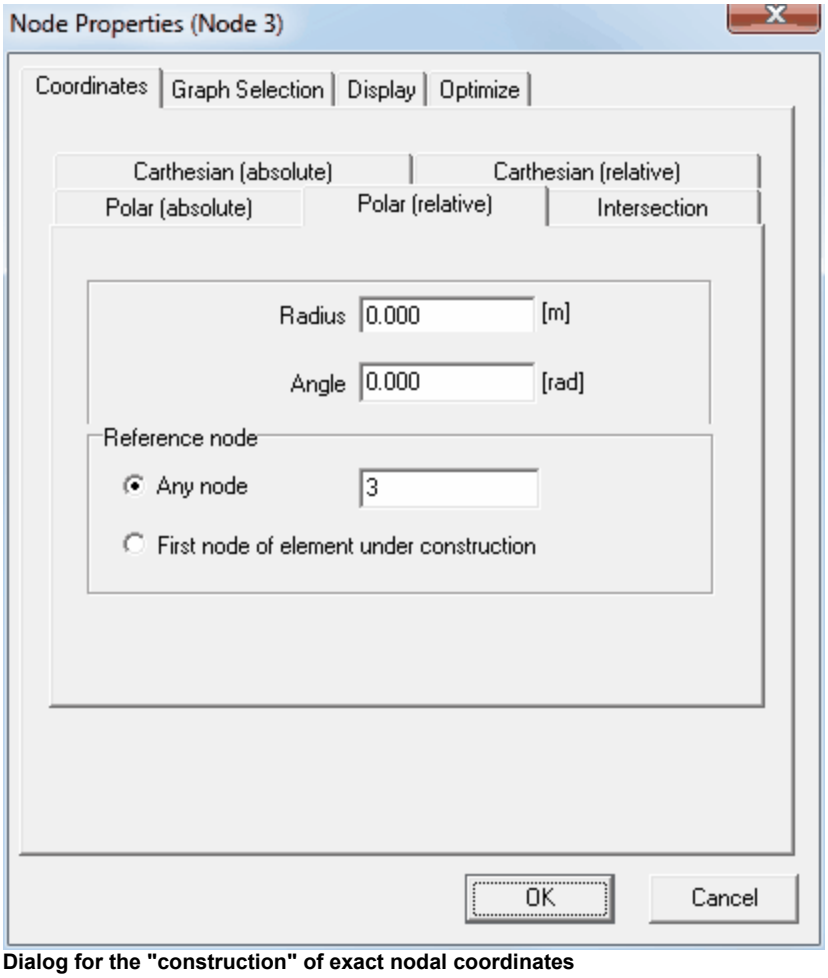
Note: The number of digits that you select only affects the way the number is displayed. The internal precision equals that of the floating point processor, so any digits that you typed in are preserved (but perhaps not displayed if you have selected a smaller number of digits).

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.

© 2024 ARTAS - Engineering Software



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/s^2 and rad, rad/s, rad/s^2. 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.

© 2024 ARTAS - Engineering Software

7.4.1 Format

A motion file is a simple ASCII text file that defines the motion profile iat 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/s^2 and rad, rad/s, rad/s^2) 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

© 2024 ARTAS - Engineering Software

a valid selection of export data. A valid set consists of Ux, Vx, Ax (or Uy, Vy, Ay) in case of displacement data. The selection must exclusively consist of these three data items of one node. Other valid selections are A, Va, Aa (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/s^2 and rad, rad/s, rad/s^2), 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

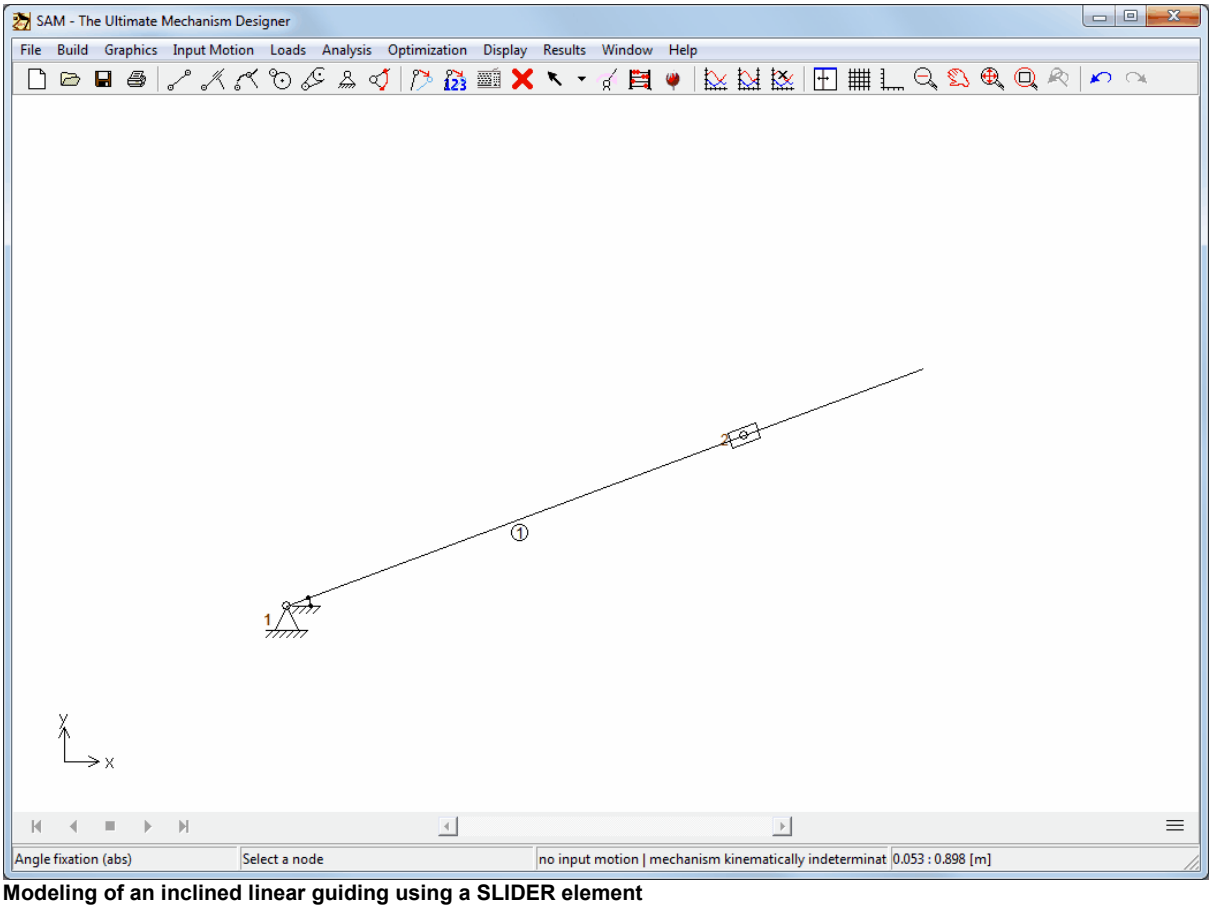
A linear actuator such as a hydraulic cylinder is modeled as follows.

- 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.

© 2024 ARTAS - Engineering Software



Modeling of an inclined linear guiding using a SLIDER element

7.8 Building Multi-Node Linkages

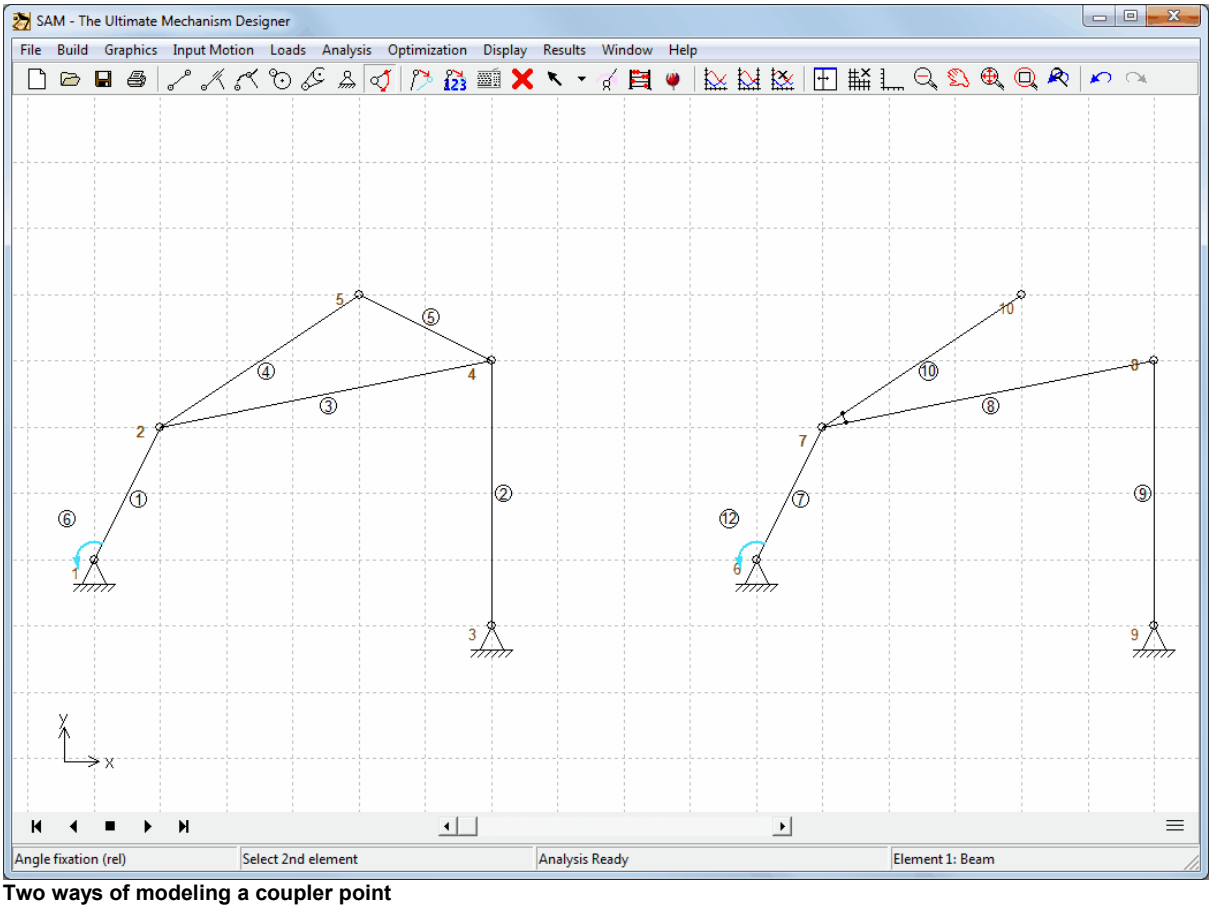
It is sometimes desirable or necessary to build a rigid linkage with more than two connecting points. This can be achieved by an assembly of several standard beam elements. In the case of a 4-bar mechanism with a coupler point, two approaches can be adopted:

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.

Note: We advice not to apply **load dependent rotational friction** in a node that is part of a triangle assembly of three beams. It is better to used the approach in which two beams are rigidly connected together. In that way any potential ambiguity in the definition of the bearing force used in the load dependent friction is avoided. See the chapter 'Kinetostatics' for more information.

© 2024 ARTAS - Engineering Software



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.

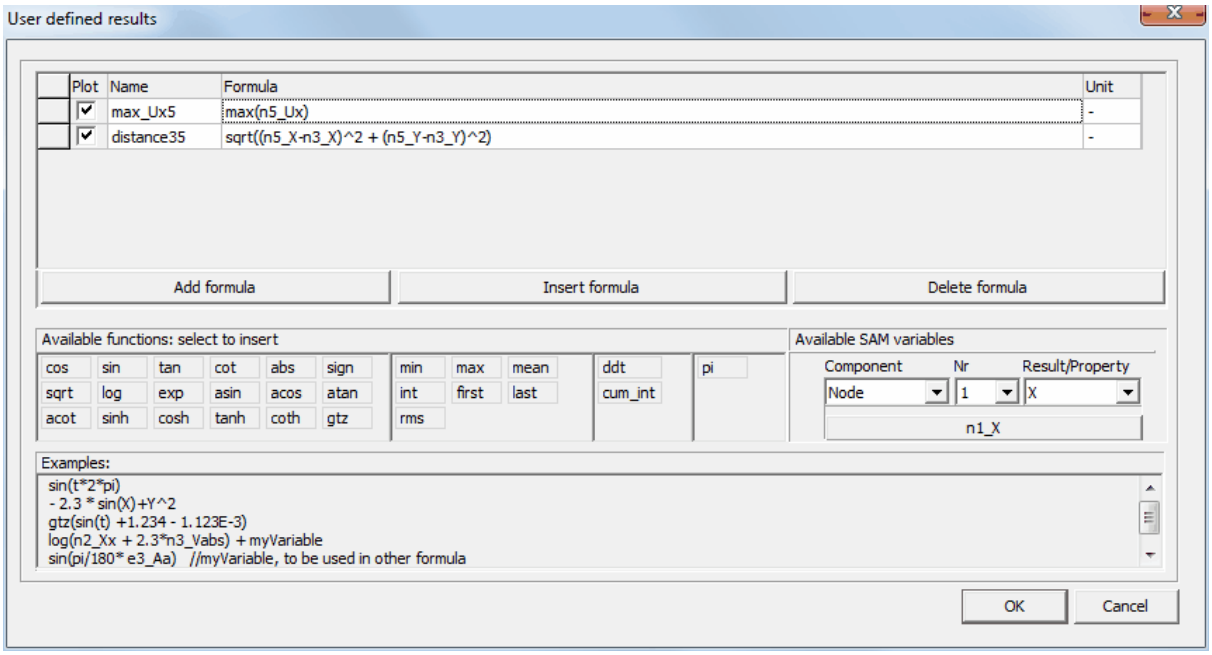
Only the following DXF-entities are supported : POINT, LINE, POLYLINE, RECTANGLE, CIRCLE, ARC and TEXT.

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

© 2024 ARTAS - Engineering Software

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
  var1 | sqrt(var2)
  var2 | 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 sqr(2)
- (a+b)*(a-b) x^2+3*x+1

© 2024 ARTAS - Engineering Software

- 256.33*exp(-t/12) (1+(2-5)*3+8/(5+3)^2)/sqrt(5^2+3^2)
- 2+3x+2x^2 0.25x + 3.5y + 1
- (12.3)/(856.6) (-1)^(2n+1)*x^n/n!
- 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)
- distance(object1, object2) in which objects can be elements (e1,e2,...), nodes(n1,n2,...), points (p1, p2,...) or shapes (s1,s2,...)

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*-sin(a); -5*-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*(-sin(a)); -5*(-2)

© 2024 ARTAS - Engineering Software

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

7.10.1 Distance

Distance calculation between objects is a powerful feature, which can also be used to detect/avoid collision or to limit a node position as a optimization parameter to an arbitrary polygon. The functionality is implemented as function in the user-defined formulas editor with the call:

distance(object1, object2)

The sequence of object1 and object2 do not effect the outcome, so distance(object1,object2) = distance(object2,object1)

Objects:
- Nodes (n1, n2, ...)
- Elements (e1, e2, ...) of type Beam and Slider
- (Dynamic) Points (p1, p2, ...)
- Shapes (s1, s2, ...) of type Line, Polyline, Circle, Rectangle, Convex/Concave Polygon (Bezier curves are excluded)

Return Value: Depending on the objects, possible return values can be positive, zero or negative. In case of overlap, a negative distance indicates the shortest vector length to reach the situation of distance=0. Sometimes this is also called minimum separation distance.

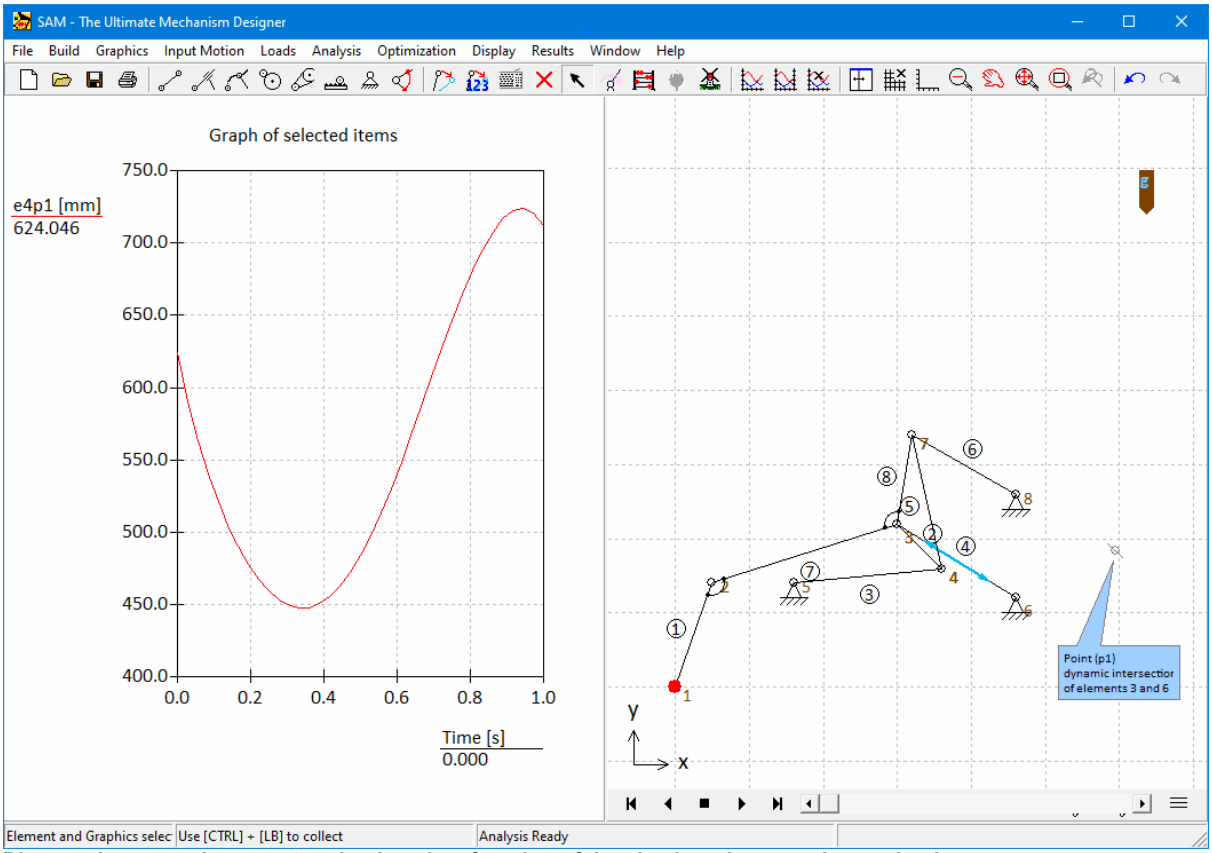
	Point / Node	Line* / Element*	Polyline*	Circle / Rectangle	Polygon (Convex)	Polygon (Concave)
Point / Node	>0: distance =0: collision <0: n.a.**	>0: left of line =0: on line <0: right of line	>0: distance =0: collision <0: n.a.**	>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision <0: overlap
Line* / Element*		>0: parallel =0: non parallel <0: n.a.**	>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision <0: overlap
Polyline			>0: no overlap =0: collision/overlap <0: n.a.***	>0: no overlap =0: collision/ overlap <0: n.a.***	>0: no overlap =0: collision/ overlap <0: n.a.***	>0: no overlap =0: collision/overlap <0: n.a.***
Circle / Rectangle				>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision/overlap <0: n.a.***
Polygon (Convex)					>0: no overlap =0: collision <0: overlap	>0: no overlap =0: collision/overlap <0: n.a.***
Polygon (Concave)						>0: no overlap =0: collision/overlap <0: n.a.***

** a line (or an element) is drawn between its first and second point which defines the direction of the line. Though it is only drawn between these two points it is considered to extend until infinity in both directions, which is an important property when determining the shortest distance from the line to any other object. The same holds for a beam and its nodes. A polyline is a collection of joined line segments, each with a finite length, and can become a polygon, if start and end point coincide. A polyline consisting of just one single line segment can be used to mimic a line segment (finite length).*
*** a negative distance can not occur due to the nature of the objects.*
**** the minimum separation distance can not be determined by SAM and the return value is set to zero, to indicate either a collision or overlap situation.*

© 2024 ARTAS - Engineering Software

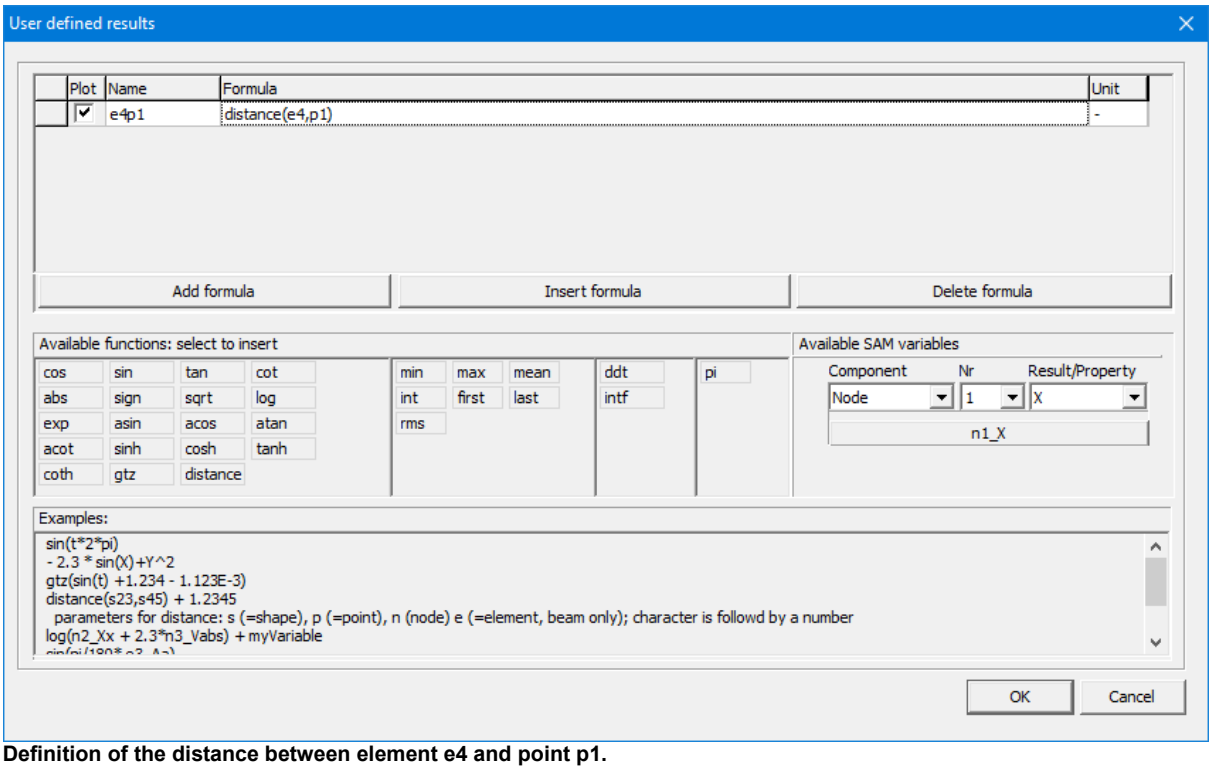
Use Case 1: Distance between line of actuation and momentary pole of rotation

In the example below it is useful to plot the distance between the driving cylinder (element e4) and the dynamic point p1 in order to monitor if that distance is always positive and does not reach zero, since in that situation the mechanism would be blocked. The dynamic point p1 corresponds to the momentary pole of rotation of the coupler plane and moves along the fixed centrode.



Distance between element e4 and point p1 as function of time is plotted next to the mechanism

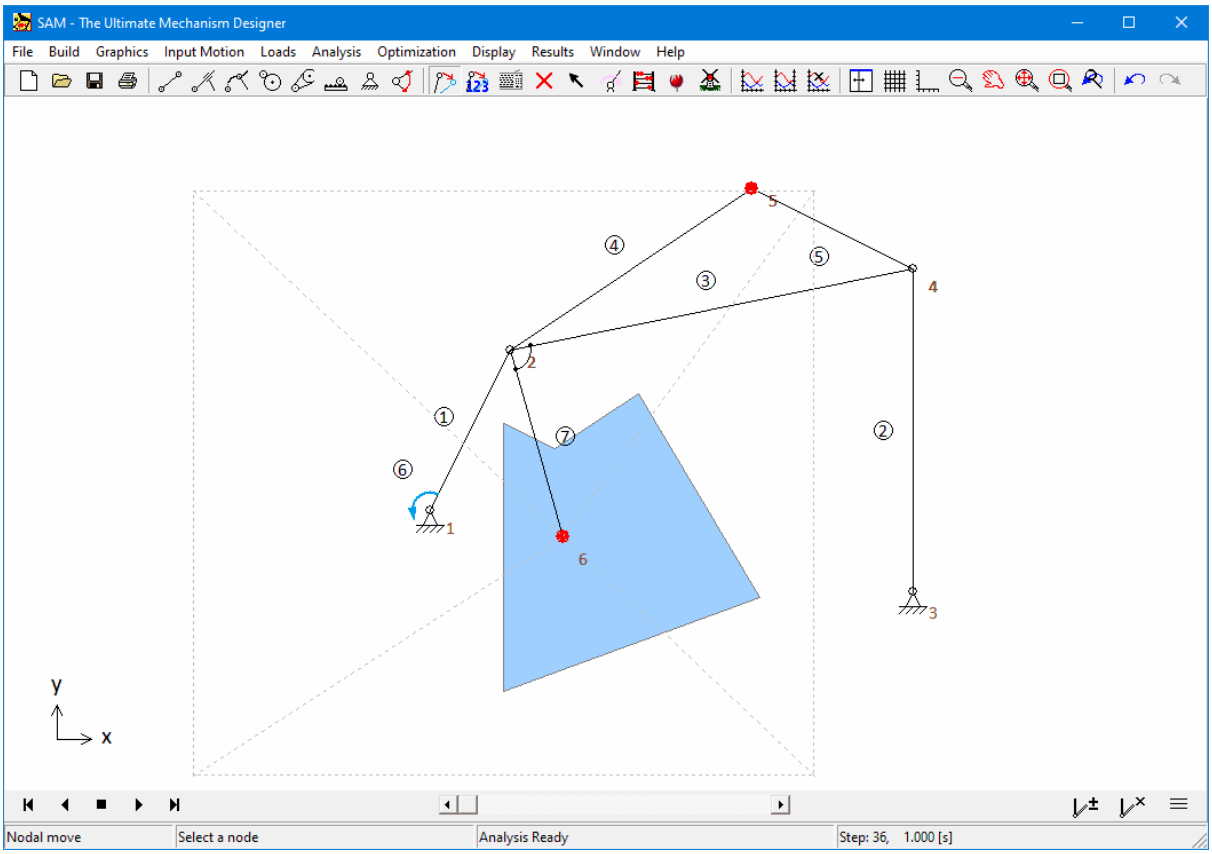
© 2024 ARTAS - Engineering Software



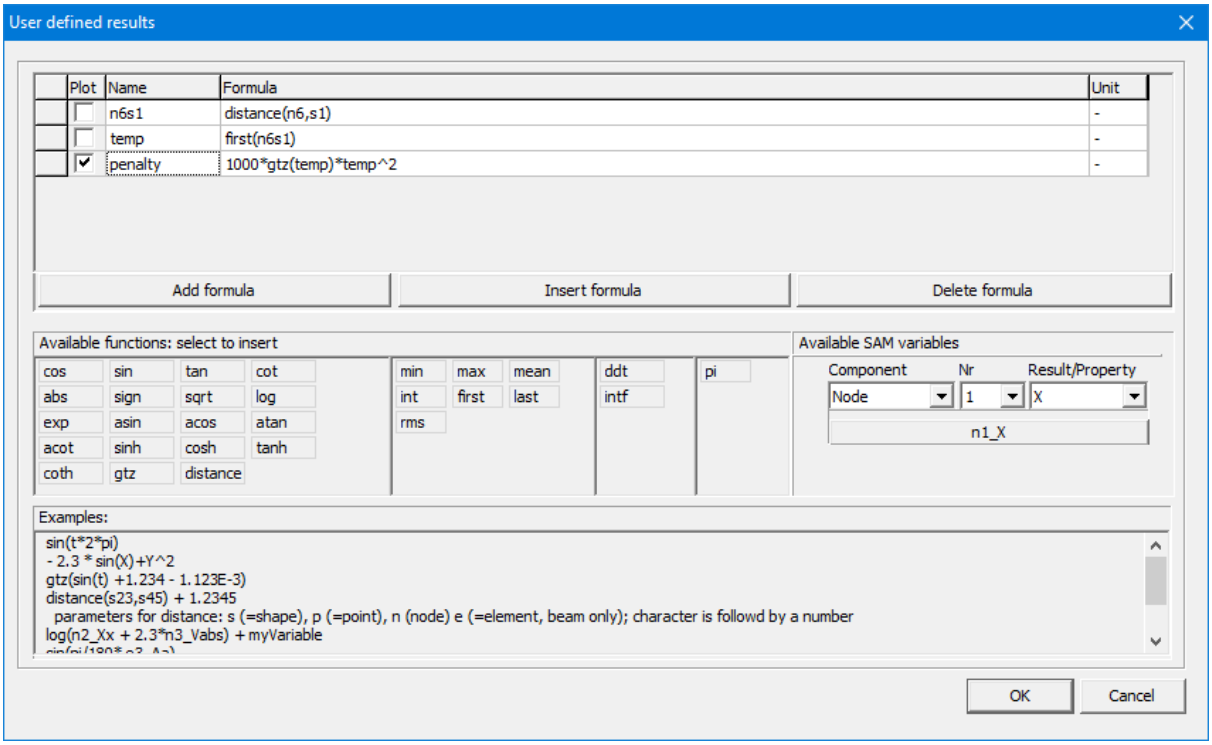
Definition of the distance between element e4 and point p1.

Use Case 2: Constraining the xy position of a node to a non-rectangular region during an optimization

During an optimization, the xy-location of a node can be an optimization parameter. Using the distance function one can define a penalty function for the optimization in order to constrain the xy-location of that node to any polygon.



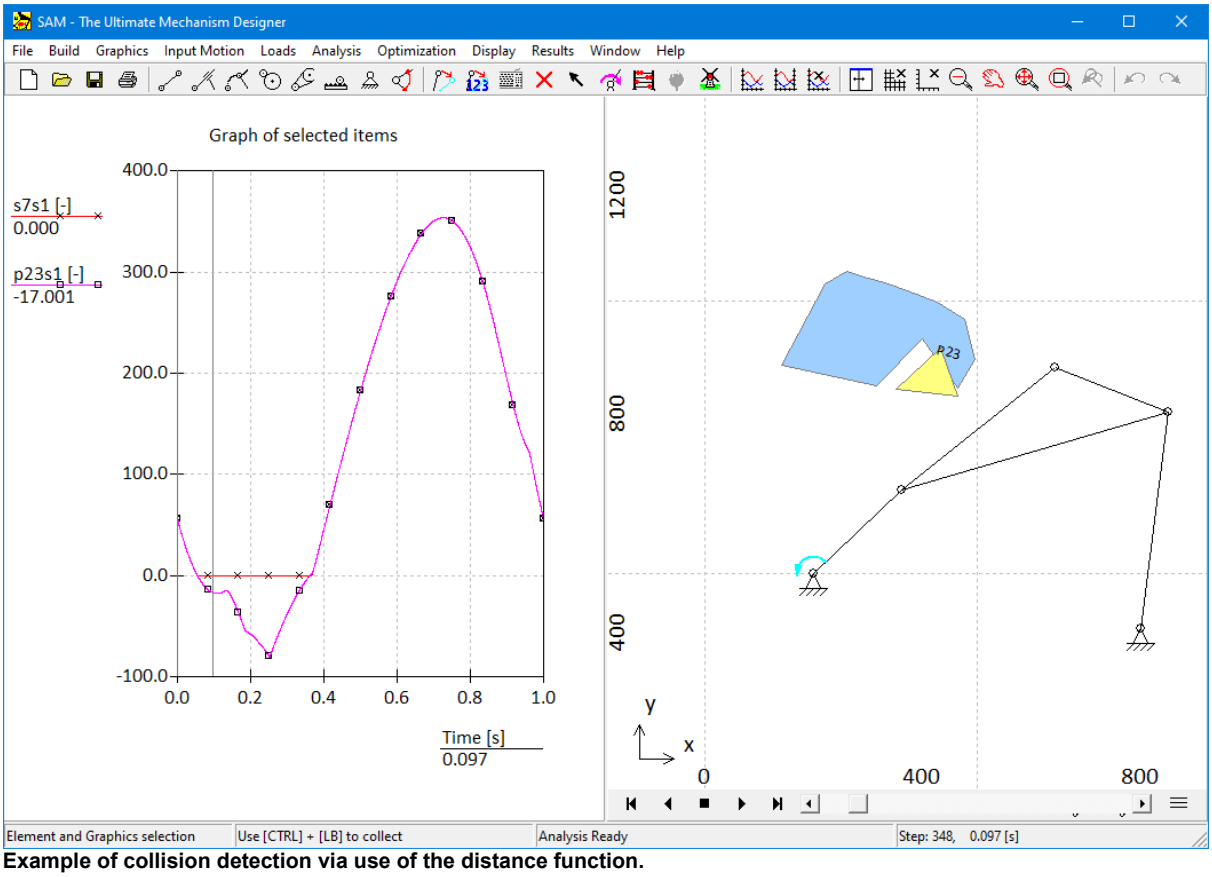
Example of an optimization in which node n6 is a design parameter but should be constrained to the area defined by the polygon (shape s1).



Definition of the penalty function to constrain node n6 to the polygon s1. As soon as node n6 lies outside of the polygon in the initial position of the mechanism, the penalty is larger than 0.

Use Case 3: Collision detection

The distance function can be used to detect collision between objects and depending on the type of objects even the penetration depth (or minimum retraction distance to avoid collision). In the following example the blue polygon is a stationary shape s1, whereas the yellow triangle (shape s7) is attached to the coupler. In the plot one sees the variable s7s1, which corresponds to the distance between the two shapes and the variable p23s1, which corresponds to the distance between point p23 on the triangle and the blue polygon. In case of two colliding shapes the distance is set to zero, whereas in the case of a point or node moving into a shape, both the positive and negative distance (negative distance = penetration depth) is shown in the plot.



Example of collision detection via use of the distance function.

© 2024 ARTAS - Engineering Software

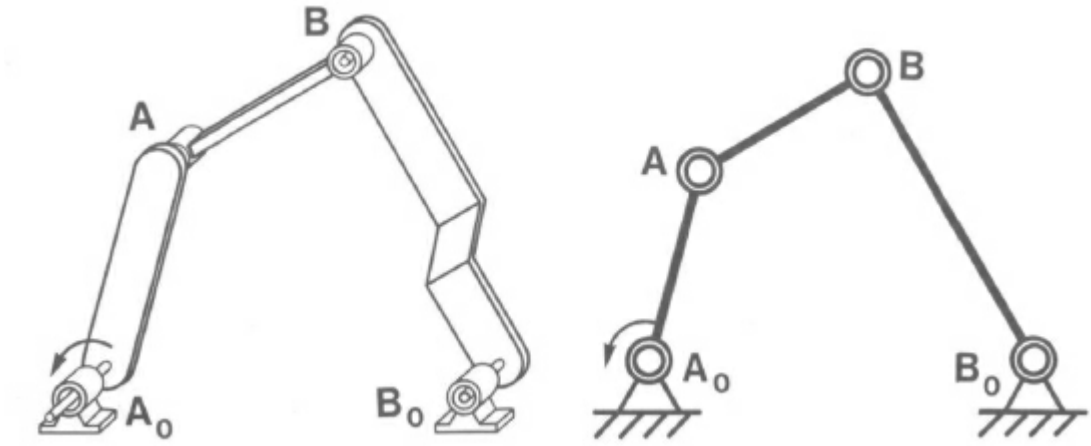
Chapter 8

8 Kinetostatics

SAM can perform a force analysis based on kinetostatics, 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 are calculated based on the specified motion, whereas in a full dynamic multi-body analysis the motion is a result of the forces and is found via integration procedures. Results of such an analysis are:

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

Real mechanisms are generally not coplanar, but the links that constitute the mechanism move in parallel planes as indicated below (reference [3]).



Real mechanism (a) and coplanar model (b)

In the first design stages a coplanar model of the mechanism, in which the links are considered to move all in the same plane instead of in different parallel planes, is very useful. From the kinematical point of view such a model is identical to the real mechanism and it serves the study of the primary function of the mechanism; that is how it generates a special motion pattern or force distribution. With the coplanar model of the mechanism the description of deformation and load transmittance is not perfect, since it neglects effects cause by this simplification, such as force example torsional effects. Nevertheless the results obtained from the coplanar model are valuable to the designer, because the calculated forces in the links of the coplanar model have a strong predictive and indicative value and are of great help to the engineer when dimensioning and materializing the various links and interconnections.

8.1 Reaction/Bearing Forces

Next to the driving force/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

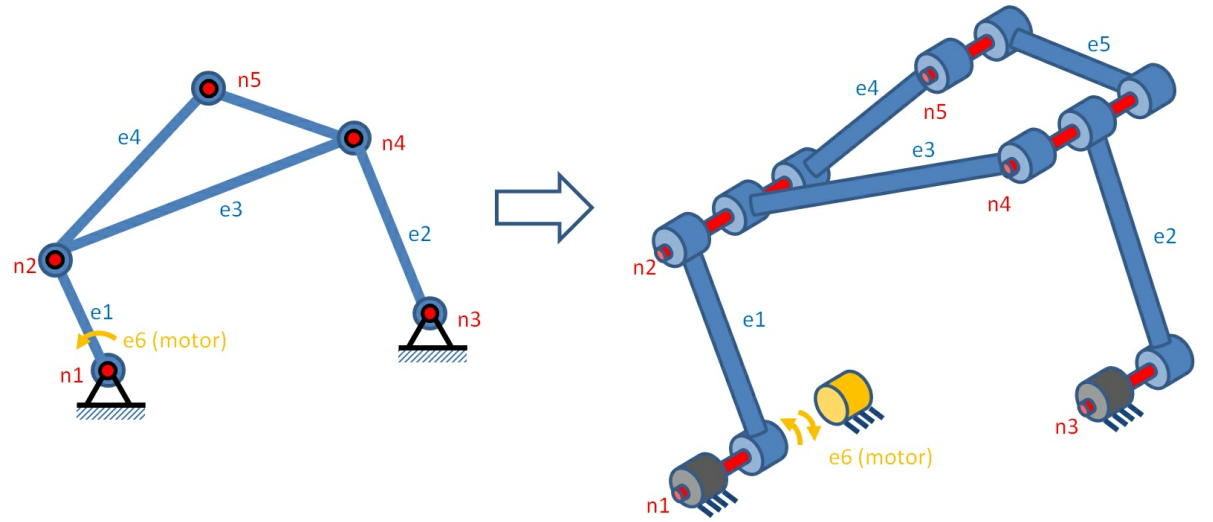
© 2024 ARTAS - Engineering Software

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).

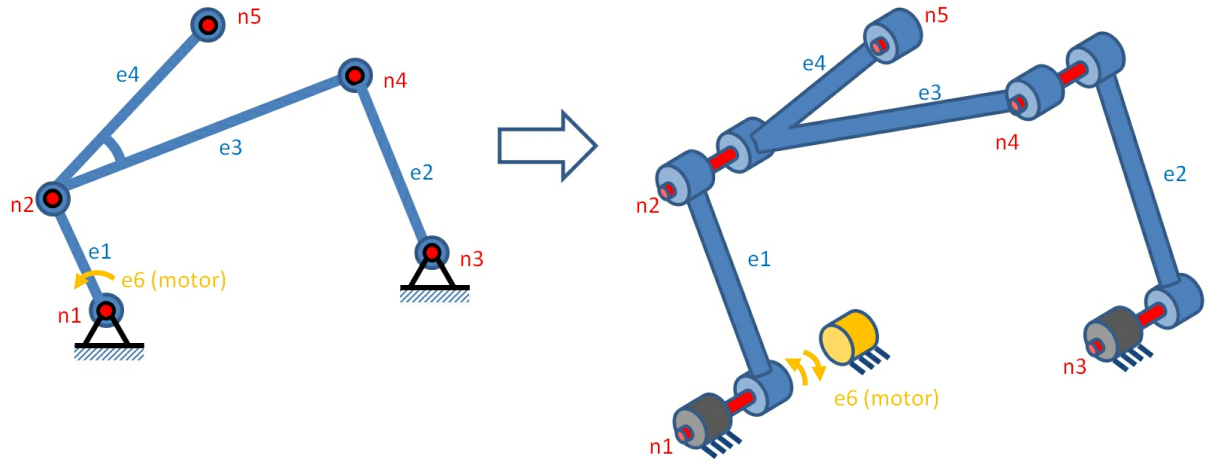
Sometimes it is necessary to build a rigid linkage with more than two connecting nodes. In the case of a 4-bar mechanism with a coupler point, two approaches can be adopted:

1. 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.

2. Two beam elements with a rigid connection (Build>Fix Relative Angle)



Schematic representation of a 4bar mechanism with coupler point (triangle configuration)



Schematic representation of a 4bar mechanism with coupler point (fixation between two elements)

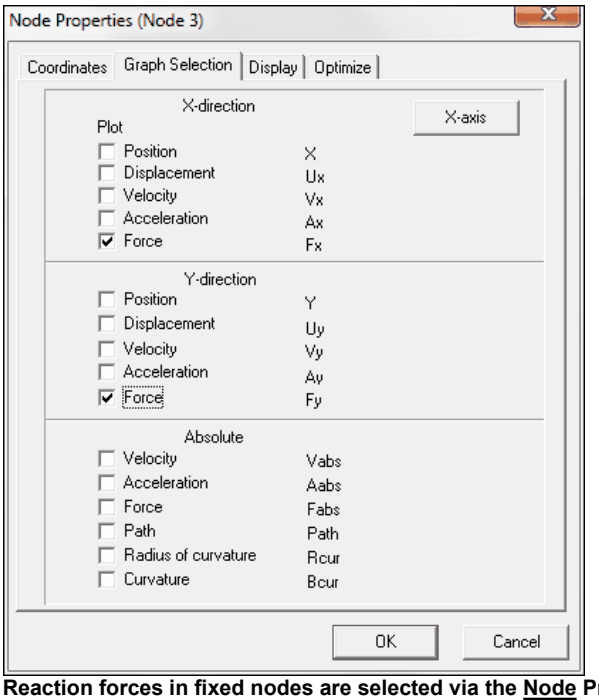
IMPORTANT: In the triangle configuration, each of the elements e3 and e4 exerts a force via its corresponding bearing to the joint shaft indicated at node n2. In the case of the fixed connection between elements e3 and e4, the **two elements have one shared bearing via which the force is transmitted to the shaft**. **Consequently, this combined force to the shaft is plotted regardless of whether is was selected as bearing force of element e3 or as bearing force of element e4.**

© 2024 ARTAS - Engineering Software

Advice: We **advice not to apply load dependent rotational friction in a node that is part of a triangle assembly of three beams**, like for example node n2. It is better to used the approach in which two beams are rigidly connected together. In that way any potential ambiguity in the definition of the bearing force used in the load dependent friction is avoided.

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 (the 'world'), 'one would have to select Fx, Fy and/or Fabs in the Graph Selection tab of the Node Properties.



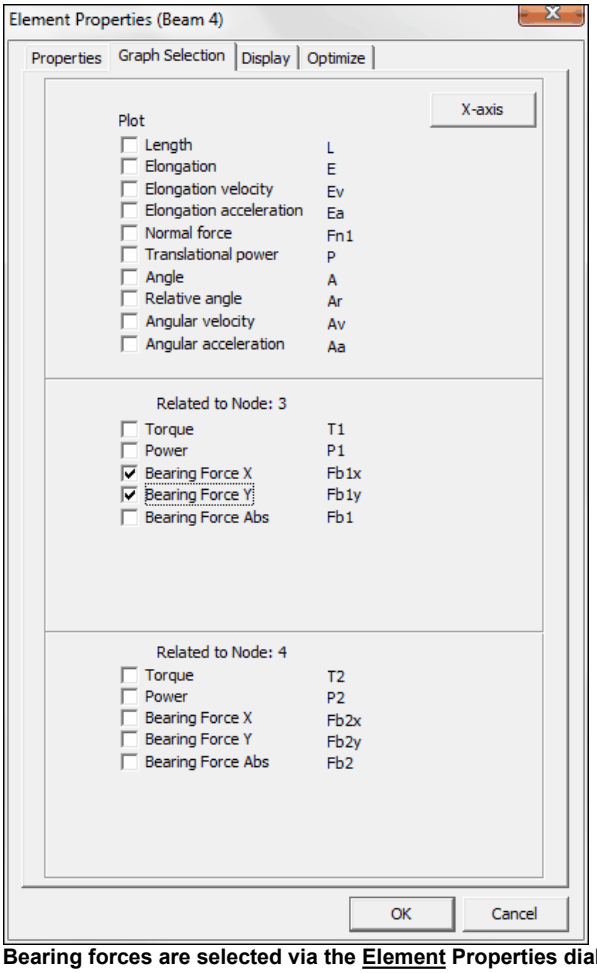
Reaction forces in fixed nodes are selected via the Node Properties dialog (example: reaction forces in node n3)

Bearing forces exerted by 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 to select Fbx, Fby and/or Fb in the element selection dialog. Two important aspects:

- **Direction/sign of the force**: This represents the force that the element exerts via its bearing to the shaft (and not the force on the element), which is crucial in case one is not only interested in the amplitude of the force but also in its direction or sign.
- **Fixation of elements**: In the case of a fixed connection between two or more elements, these elements have one shared bearing via which the force is transmitted to the shaft. Consequently, this combined force to the shaft is plotted when selecting the bearing force from a properties dialogue of any of these elements.

© 2024 ARTAS - Engineering Software



Bearing forces are selected via the Element Properties dialog (example: bearing forces of element 4 at node/shaft n3)

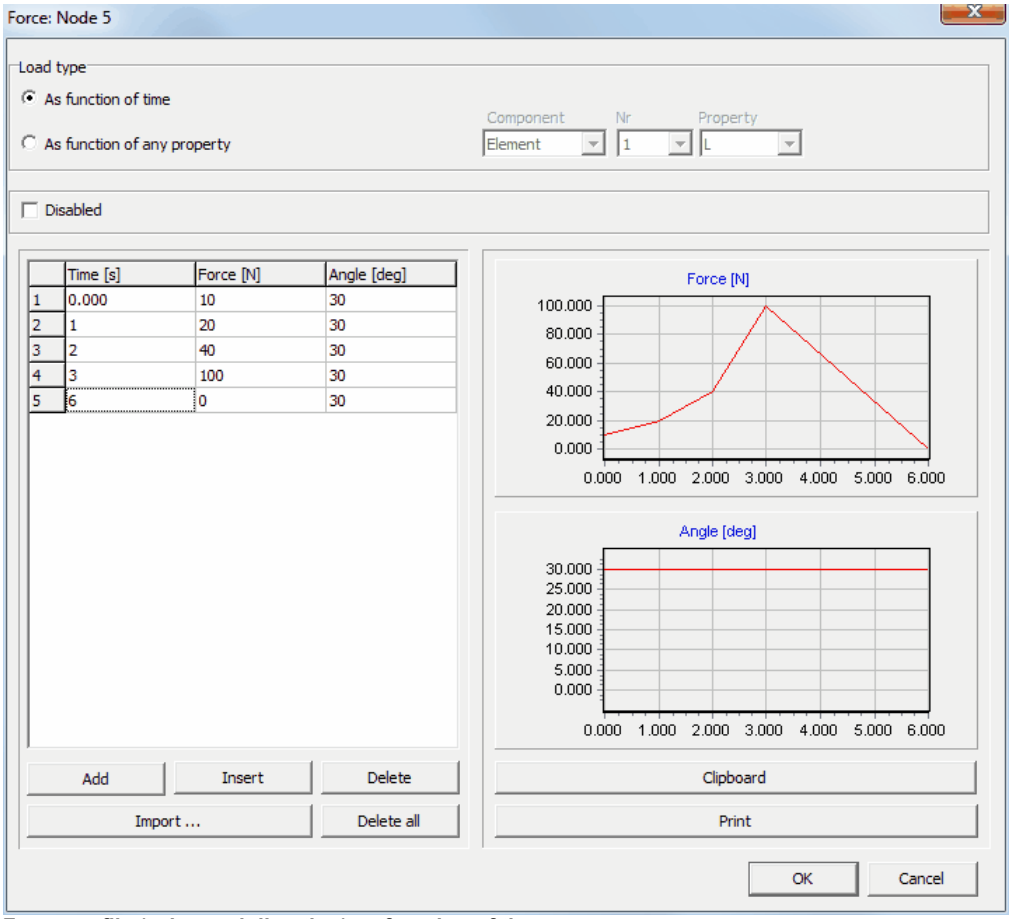
8.2 External Force/Torque

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 profile (value & direction) as function of:

- time
- any kinematic property

Force (value & direction) as function of time

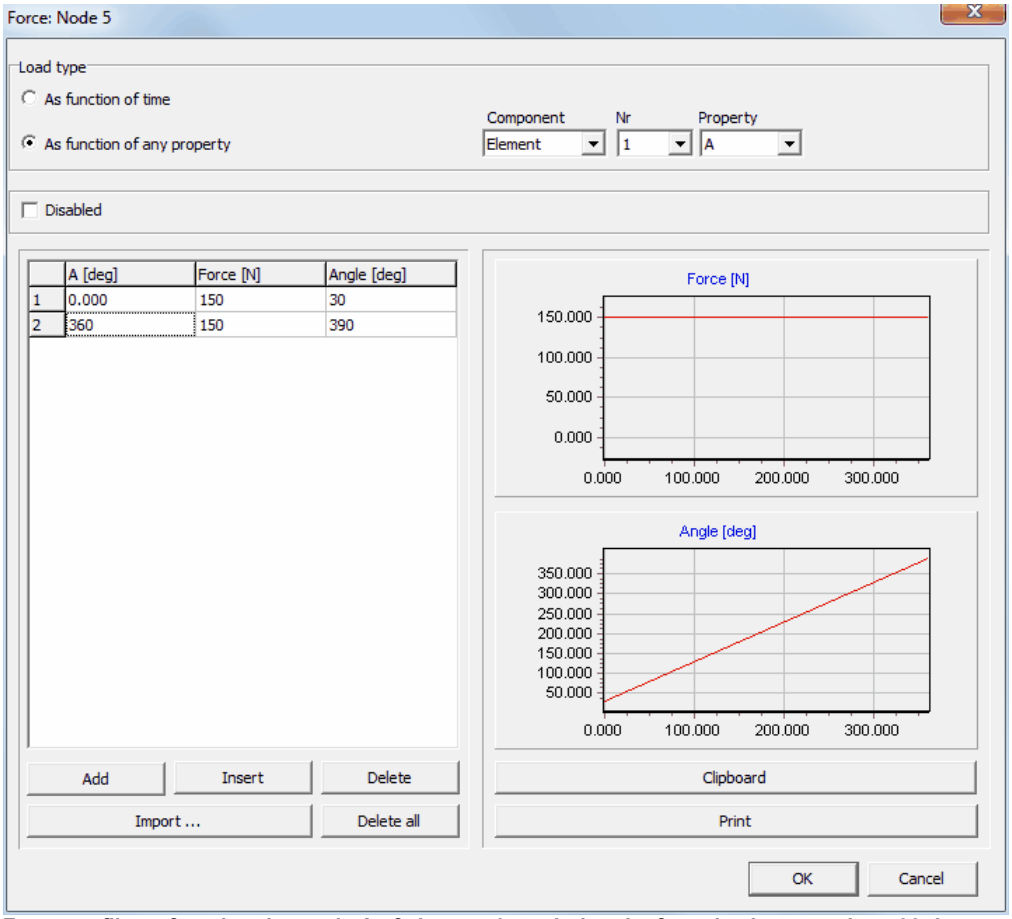
© 2024 ARTAS - Engineering Software



Force profile (value and direction) as function of time

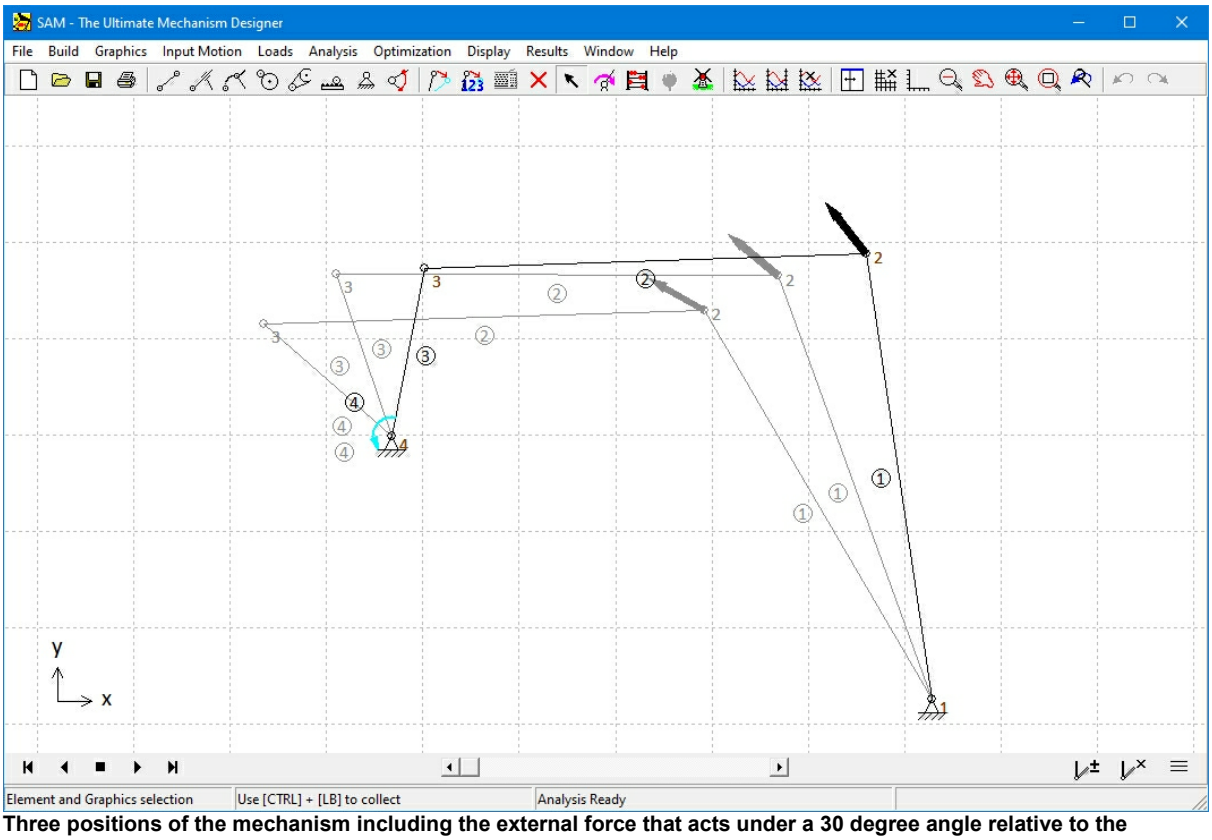
Forces as function of any kinematic property

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 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

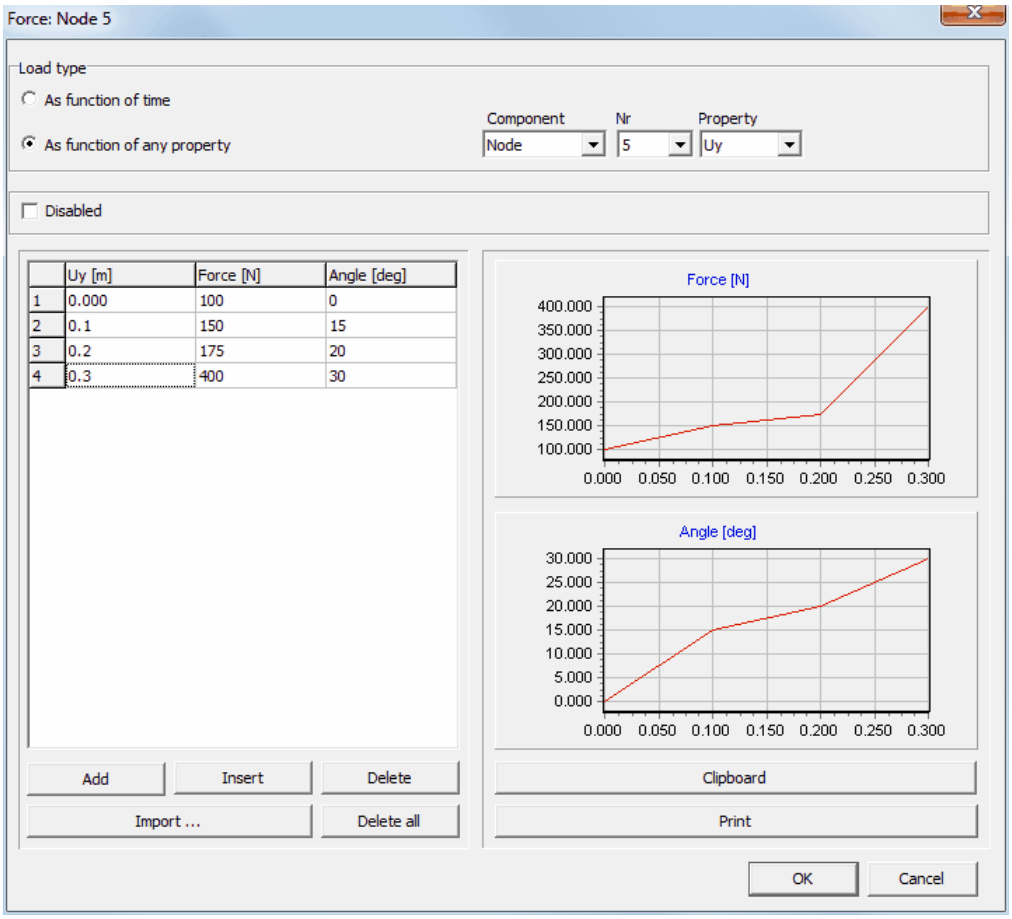
© 2024 ARTAS - Engineering Software



Three positions of the mechanism including the external force that acts under a 30 degree angle relative to the orientation of element 1

© 2024 ARTAS - Engineering Software

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

© 2024 ARTAS - Engineering Software

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. The distance function is a powerful function in the formula parser to consider collision during an optimization or to constrain a node to a non-rectangular region.

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 of 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.

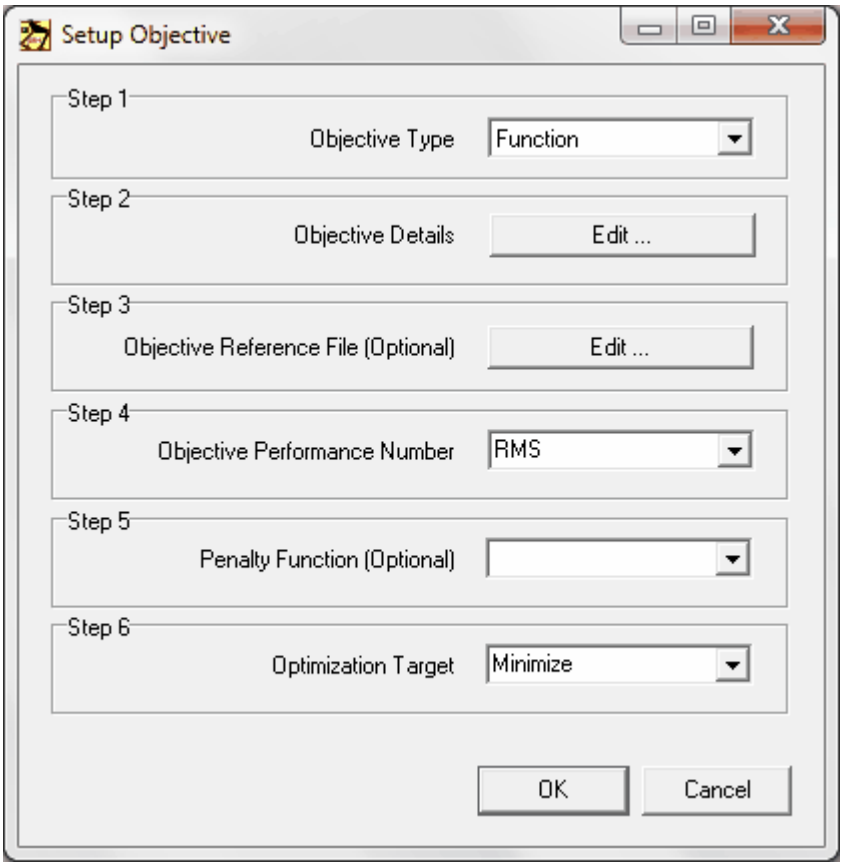
Next, the designer can select one of the results from the Explore window and start a local optimization. This local search can be either based on a Simplex technique or on a Evolutionary Algorithm with a smaller parameter range centered around the selected solution.

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.

© 2024 ARTAS - Engineering Software

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.



Dialog to define the optimization target

The following choices and setup actions have to be performed :

Step 1: Objective Type

a. Function: this can be any motion/force quantity

b. Path: this can be the motion of any node

c. 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:

© 2024 ARTAS - Engineering Software

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.

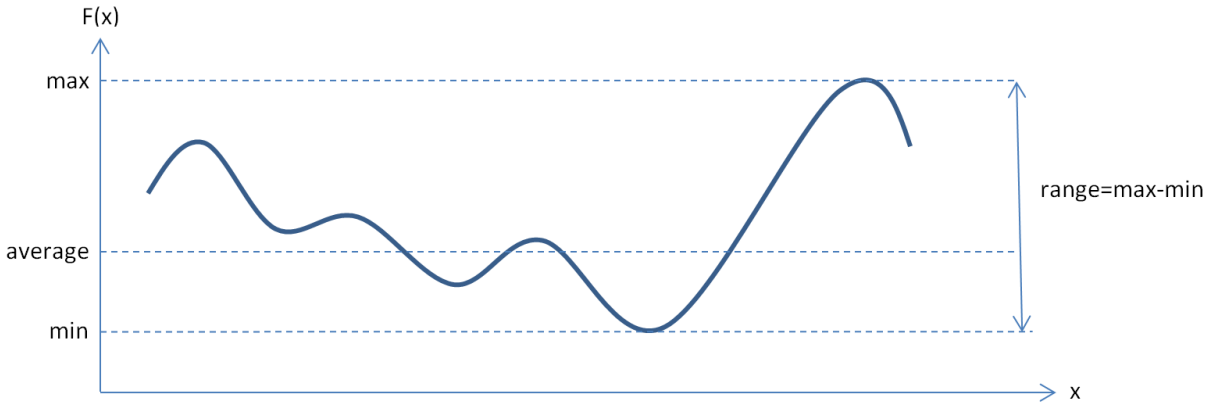
In case the optimization target has been set to: Maximize(cost function+penalty) it is essential that the penalty must be a negative value in order to make a solution that violates a constraint less favorable.

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.

© 2024 ARTAS - Engineering Software

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(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
```

© 2024 ARTAS - Engineering Software

```
  :     :
x_n   y_n
```

Path including timing

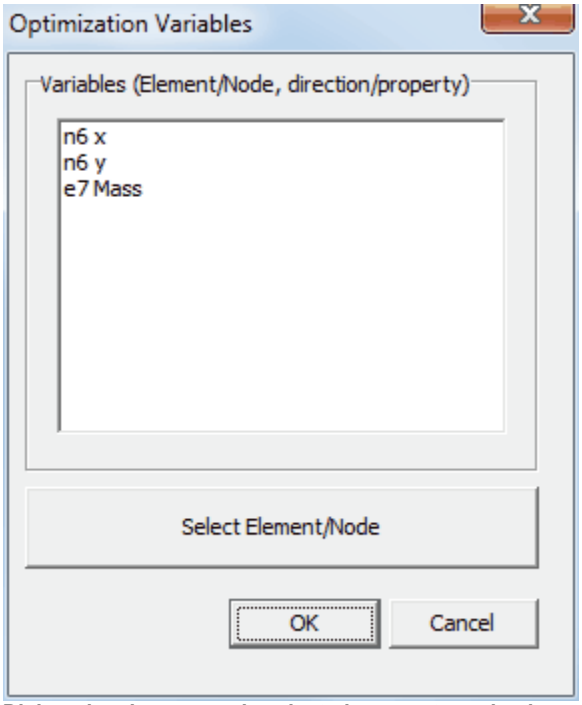
```
time_1   x_1   y_1
time_2   x_2   y_2
  :       :     :
time_n   x_n   y_n
```

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

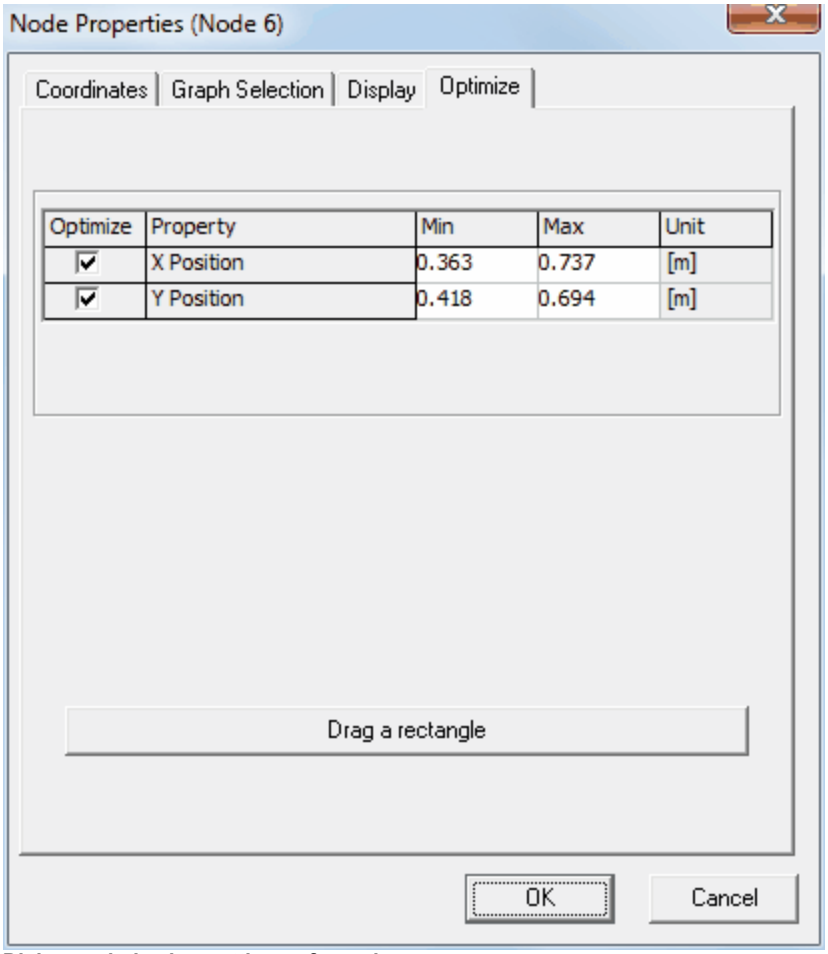
The second step in the optimization of a mechanism is the definition of the parameters and the range within which the parameters can be varied in order to optimize the objective function. The menu Optimization/Parameters invokes the following dialogs, which shows the currently selected parameters plus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

© 2024 ARTAS - Engineering Software



Dialog optimization settings of a node

In case a simple rectangular area as allowable parameter space for a node is not sufficient, one can make use of a penalty function based on the distance between that node and an arbitrary shape. See the following chapter 'Distance' for an example.

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, that can be defined in the post processing formula editor. The distance

© 2024 ARTAS - Engineering Software

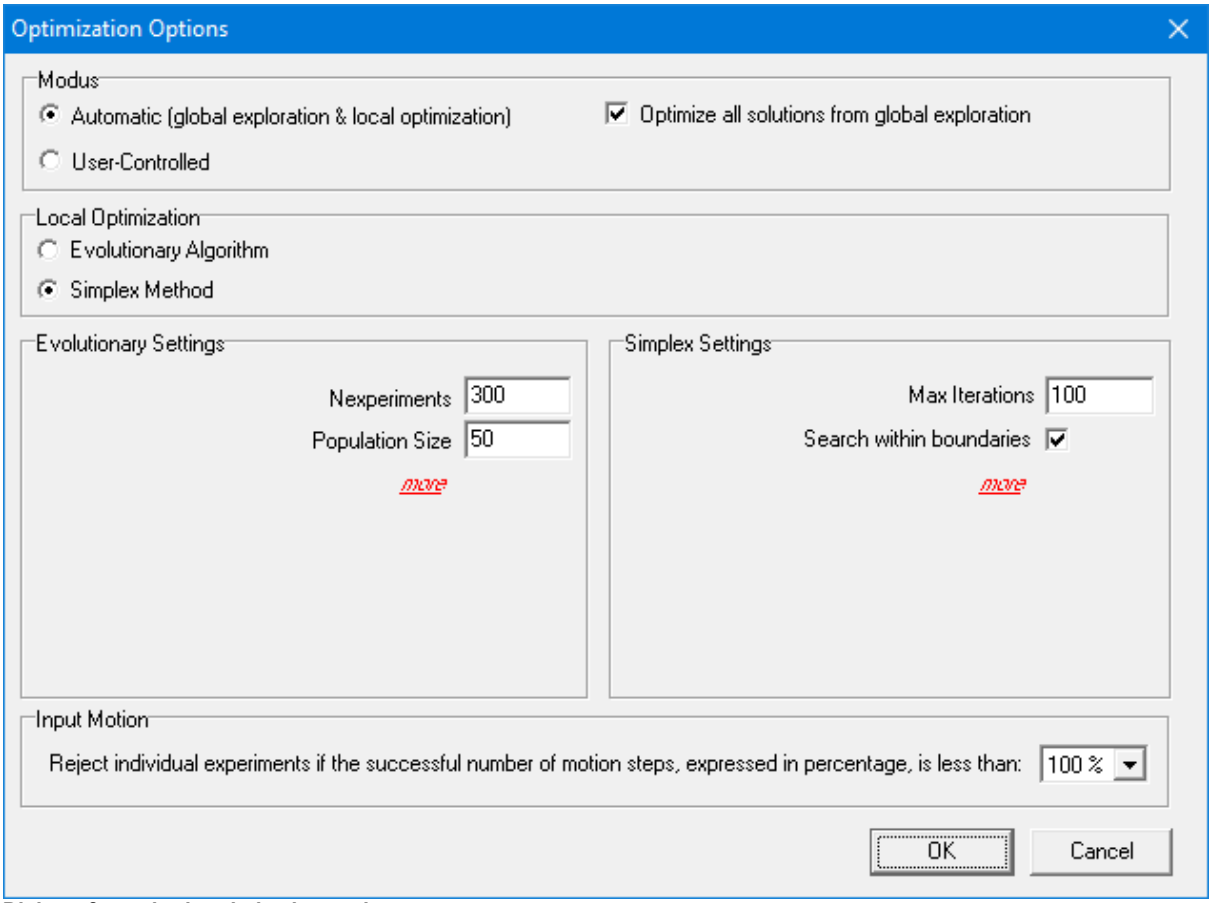
function is a powerful function in the formula editor to consider collision during an optimization or to constrain a node to a non-rectangular region.

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:



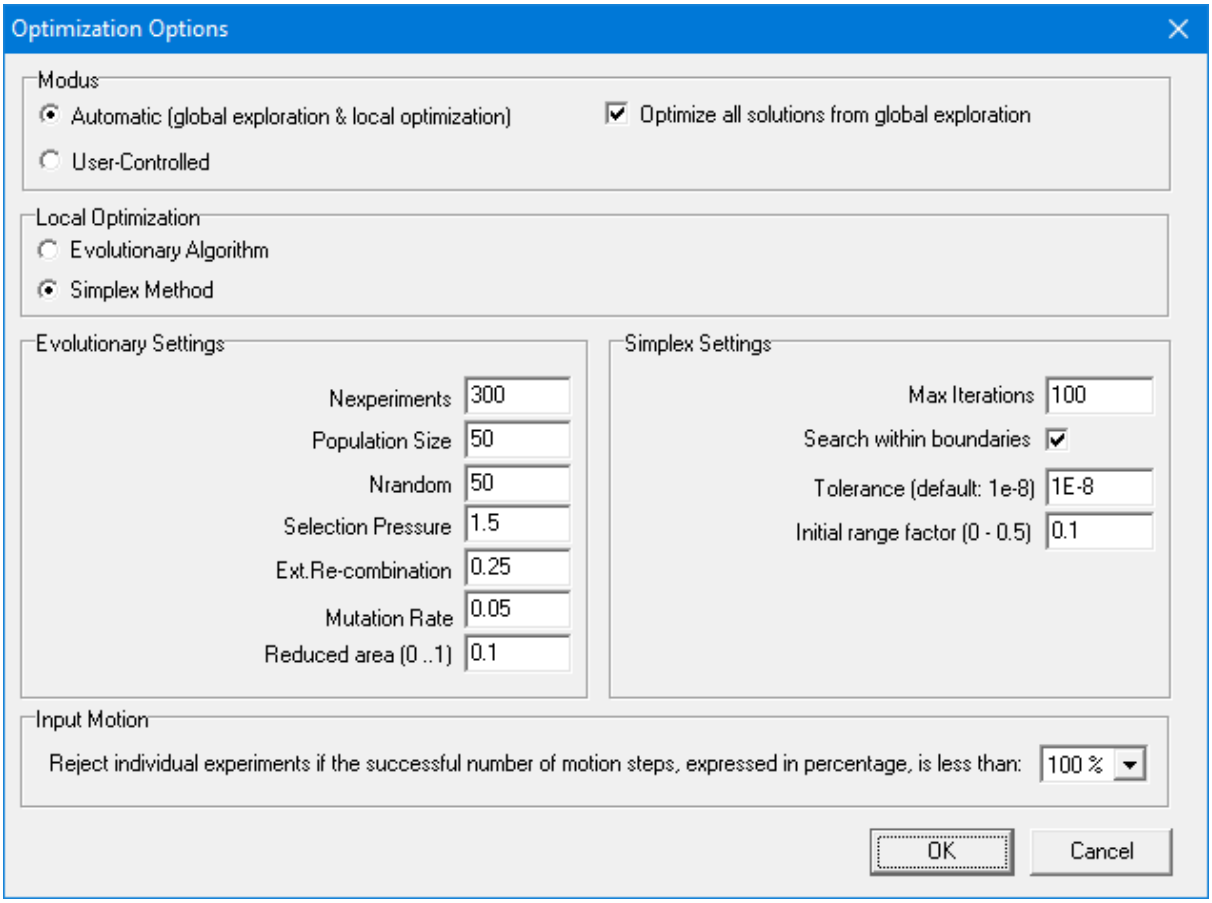
Dialog of standard optimization options

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).

© 2024 ARTAS - Engineering Software

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**.



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.

© 2024 ARTAS - Engineering Software

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 extenting 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 pertubations 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 pertubation to each of the parameters of the starting parameter set. The small pertubation 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.

© 2024 ARTAS - Engineering Software

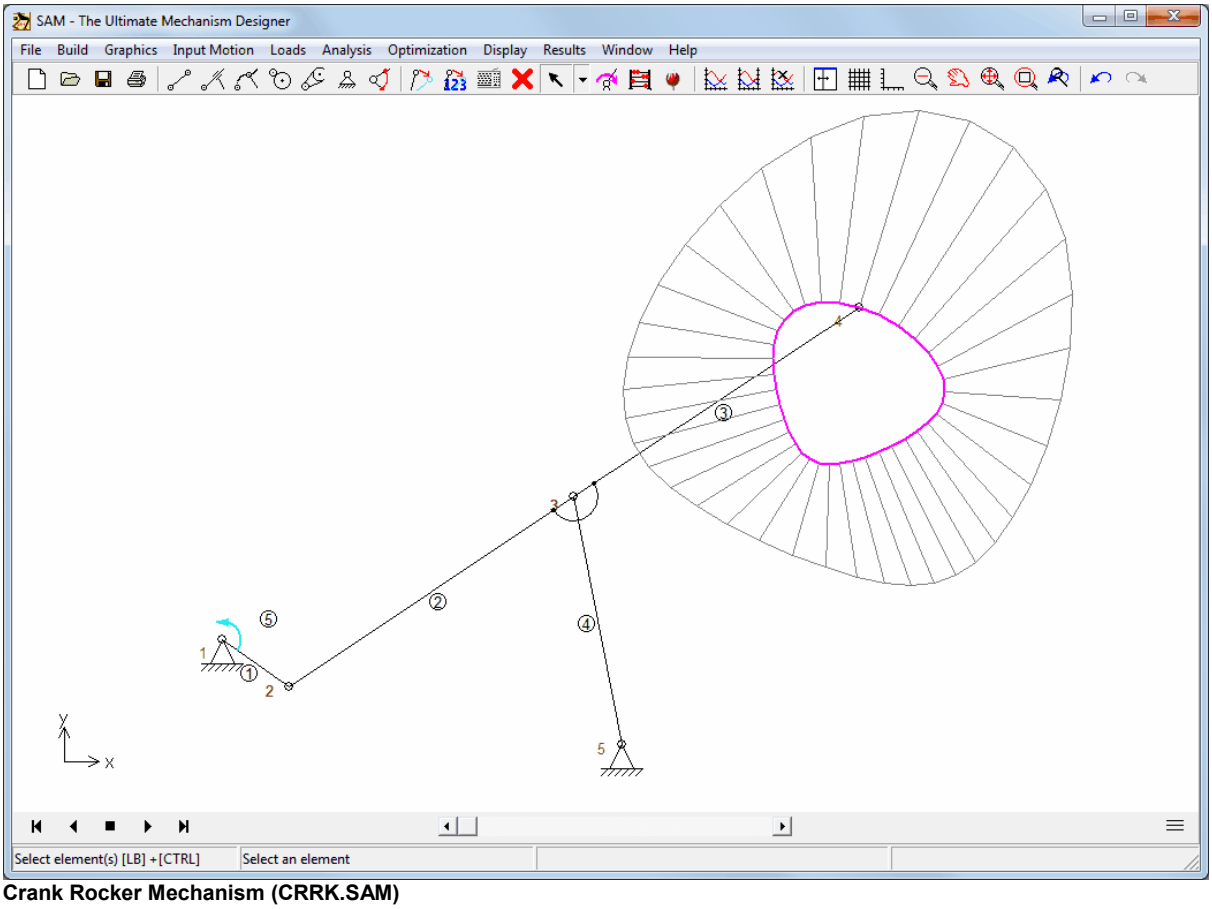
Chapter 10

10 Examples

10.1 Motion Analysis

10.1.1 Crank Rocker Mechanism

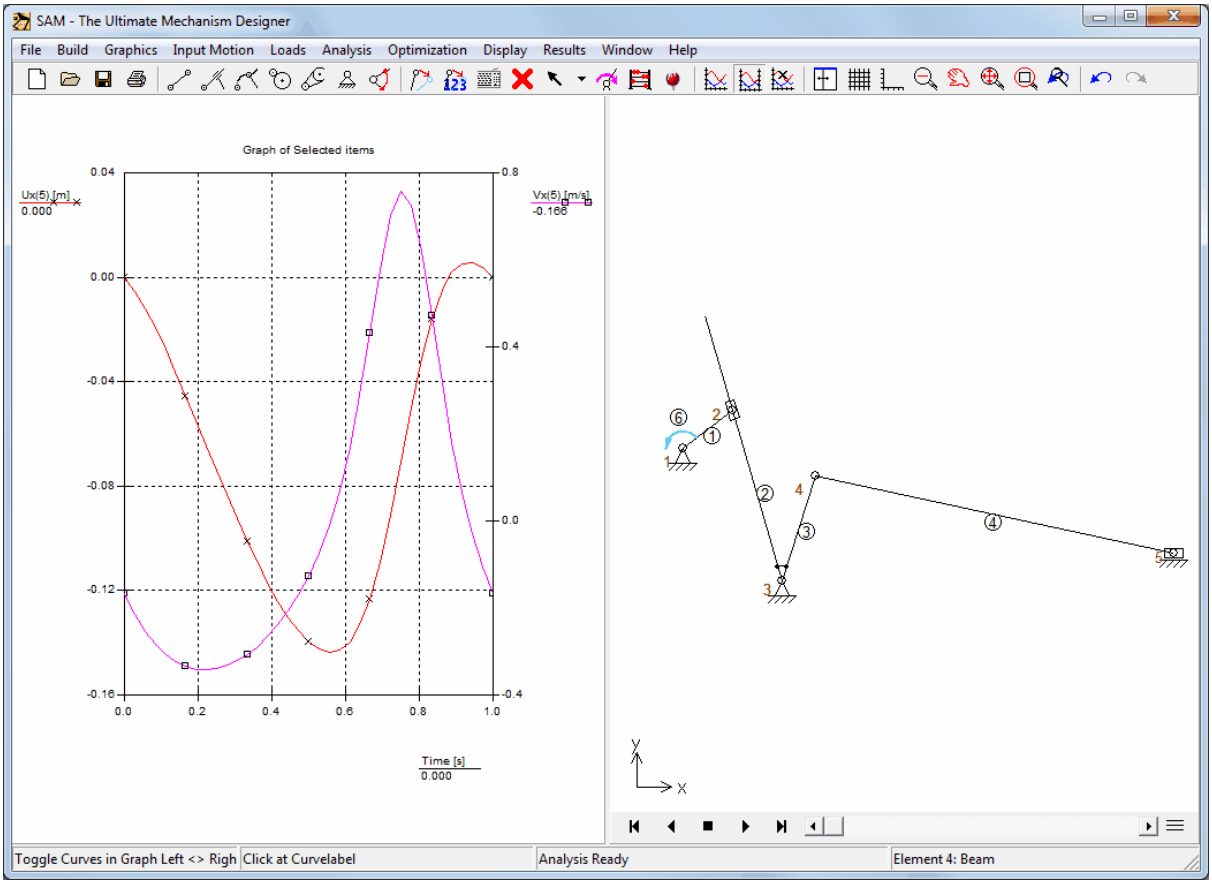
Example file CRRK.SAM defines a crank rocker mechanism, the crank of which is driven with constant speed. The path and velocity of node 4 is analyzed.



Crank Rocker Mechanism (CRRK.SAM)

10.1.2 Quick-Return Mechanism

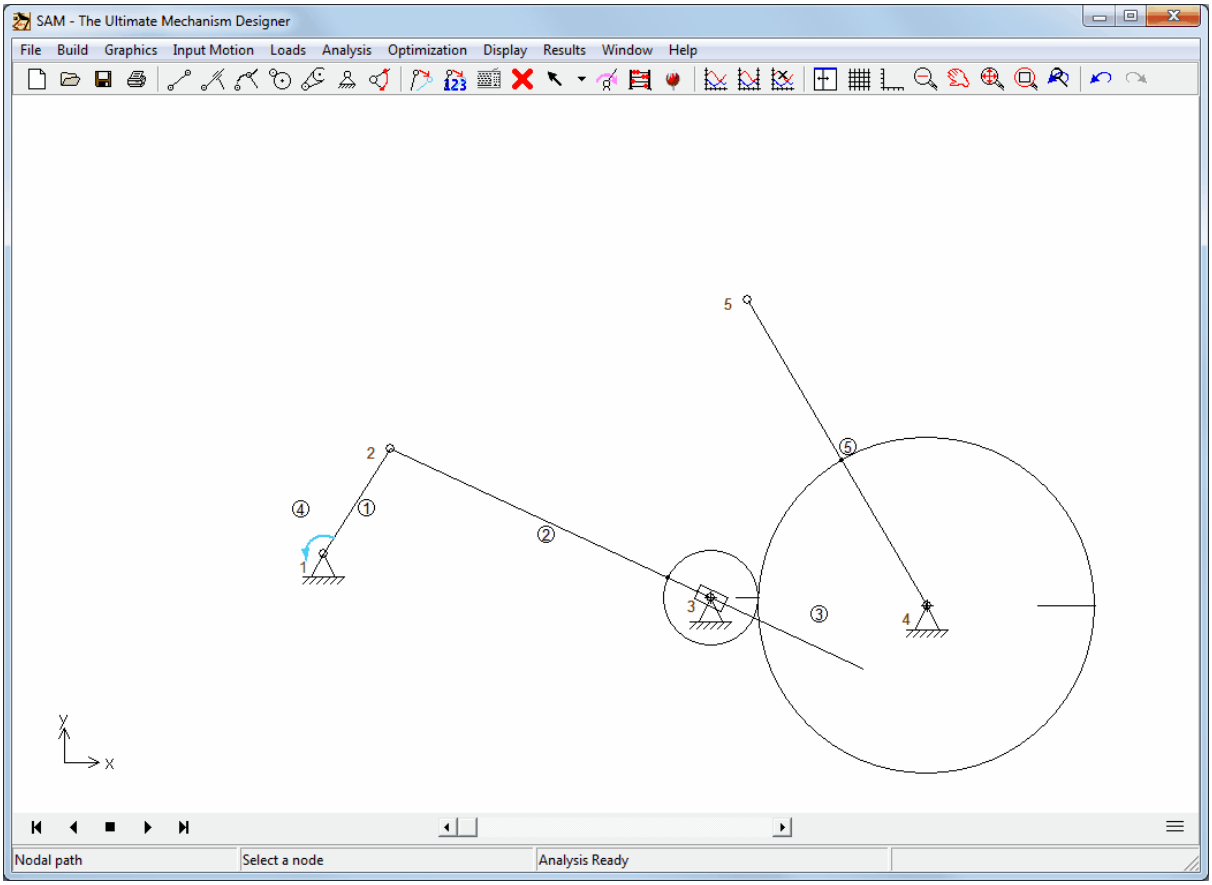
In push-up units, a quick return mechanism is often used. Important in these mechanisms is the displacement and velocity diagram of the translating node.



Quick-Return Mechanism (QKRT.SAM)

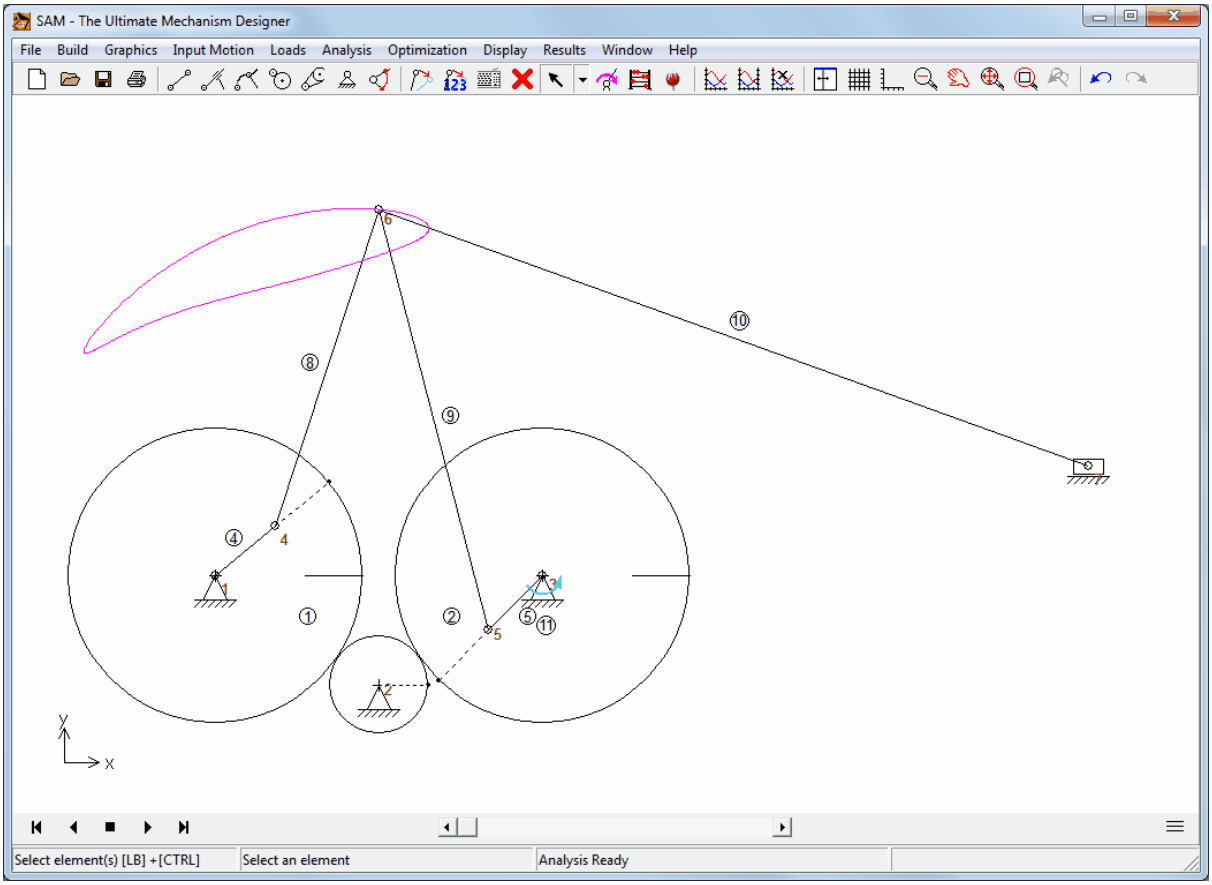
© 2024 ARTAS - Engineering Software

10.1.3 Slider Driven Gear Transmission



Slider Driven Gear Transmission (SLGR.SAM)

10.1.4 Double Crank Mechanism

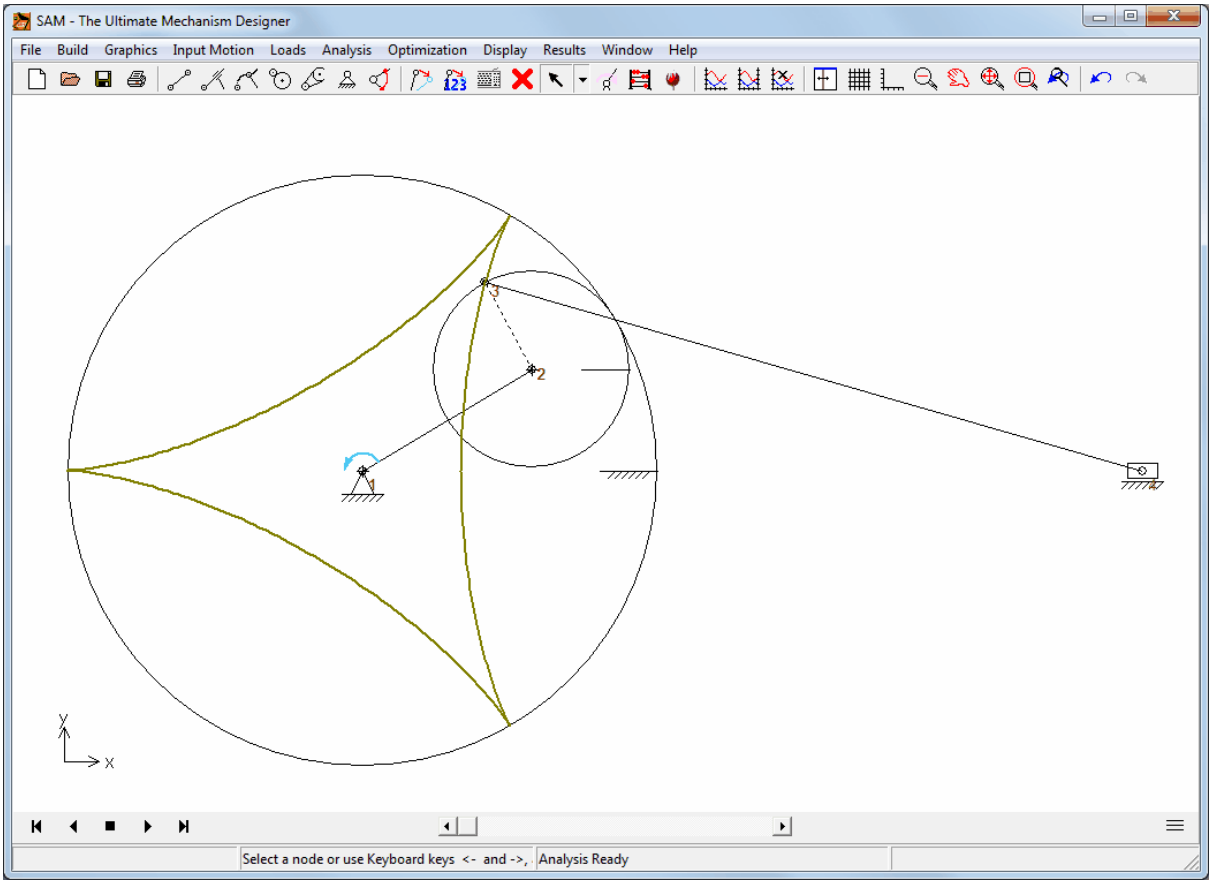


Double Crank Mechanism (DBCR.SAM)

10.1.5 Planetary Mechanism (Hypo-Cyclic)

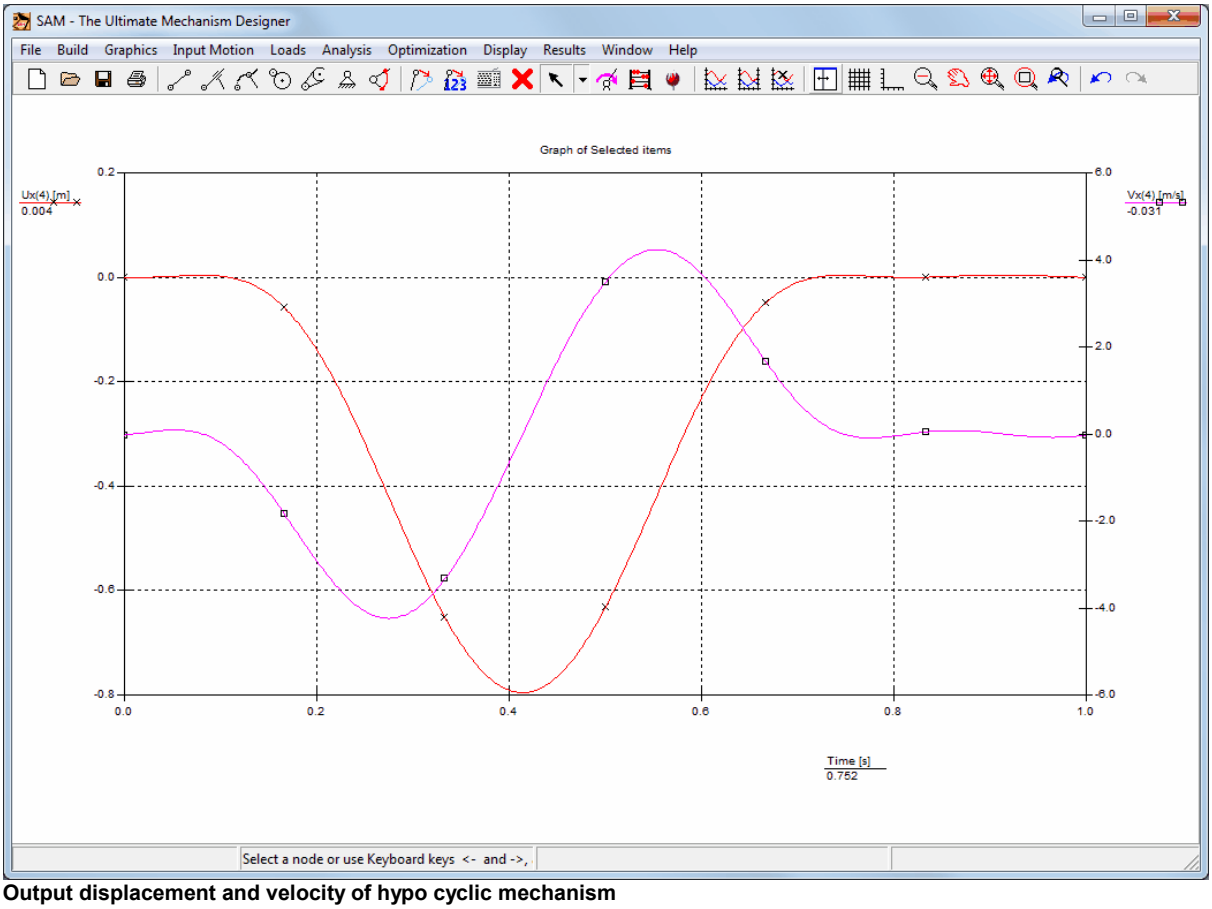
A hypo cyclic planetary mechanism can be used to generate an approximated dwell. This is achieved by attaching an output linkage to the gear in such a way that the length of that linkage equals the radius of curvature of the connection (gear) node.

© 2024 ARTAS - Engineering Software



Hypocyclic Mechanism (HYPO.SAM)

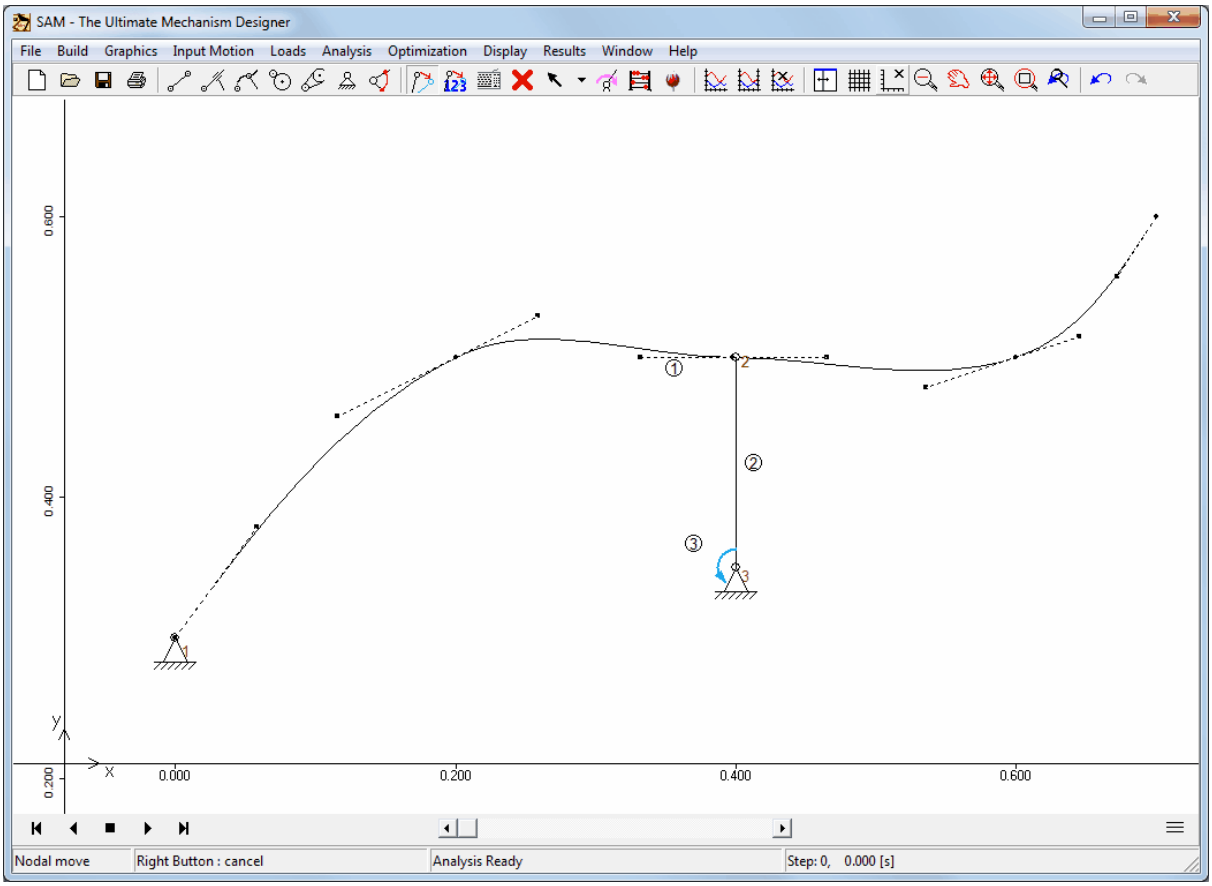
© 2024 ARTAS - Engineering Software



Output displacement and velocity of hypo cyclic mechanism

10.1.6 Curved Slider

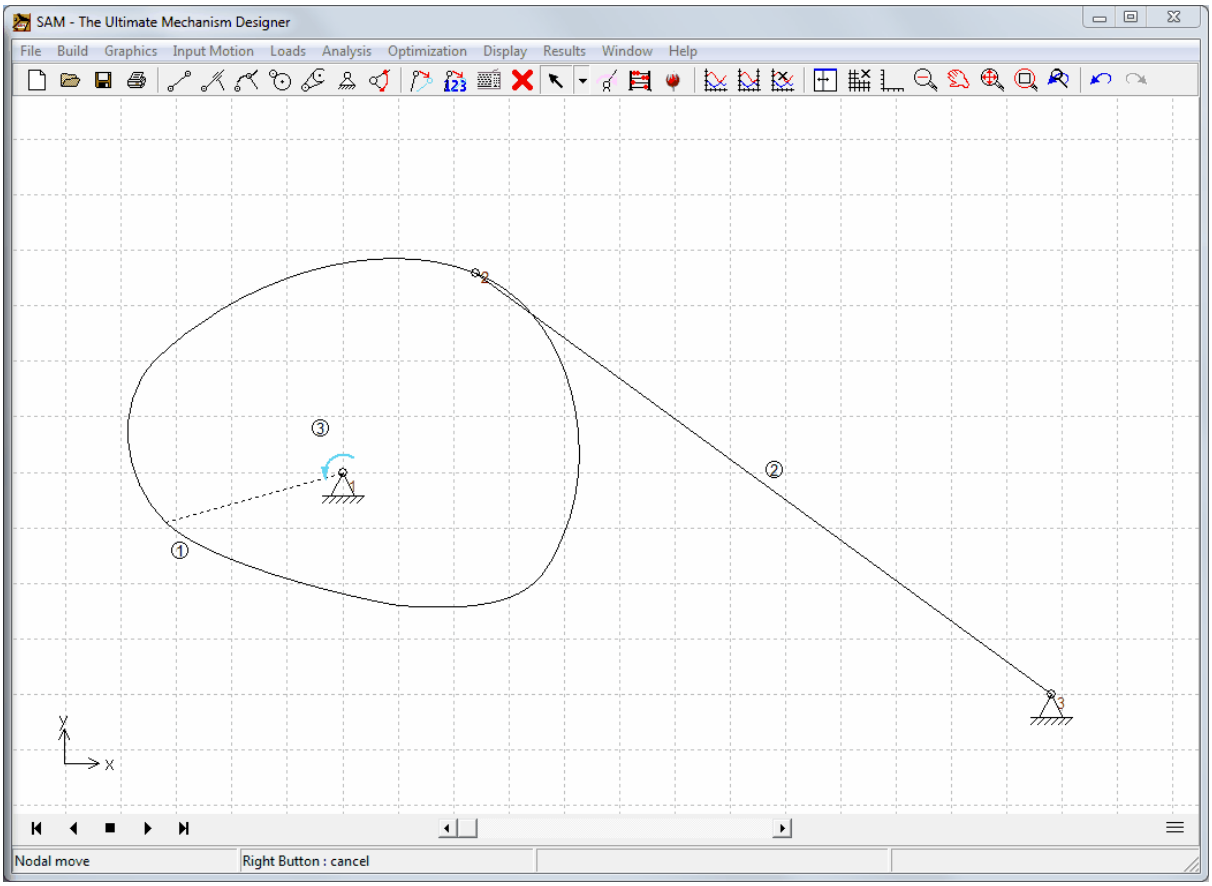
The curved slider element is a very versatile element that can be used to model any sort of non-straight slider based on Bezier segments.



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.



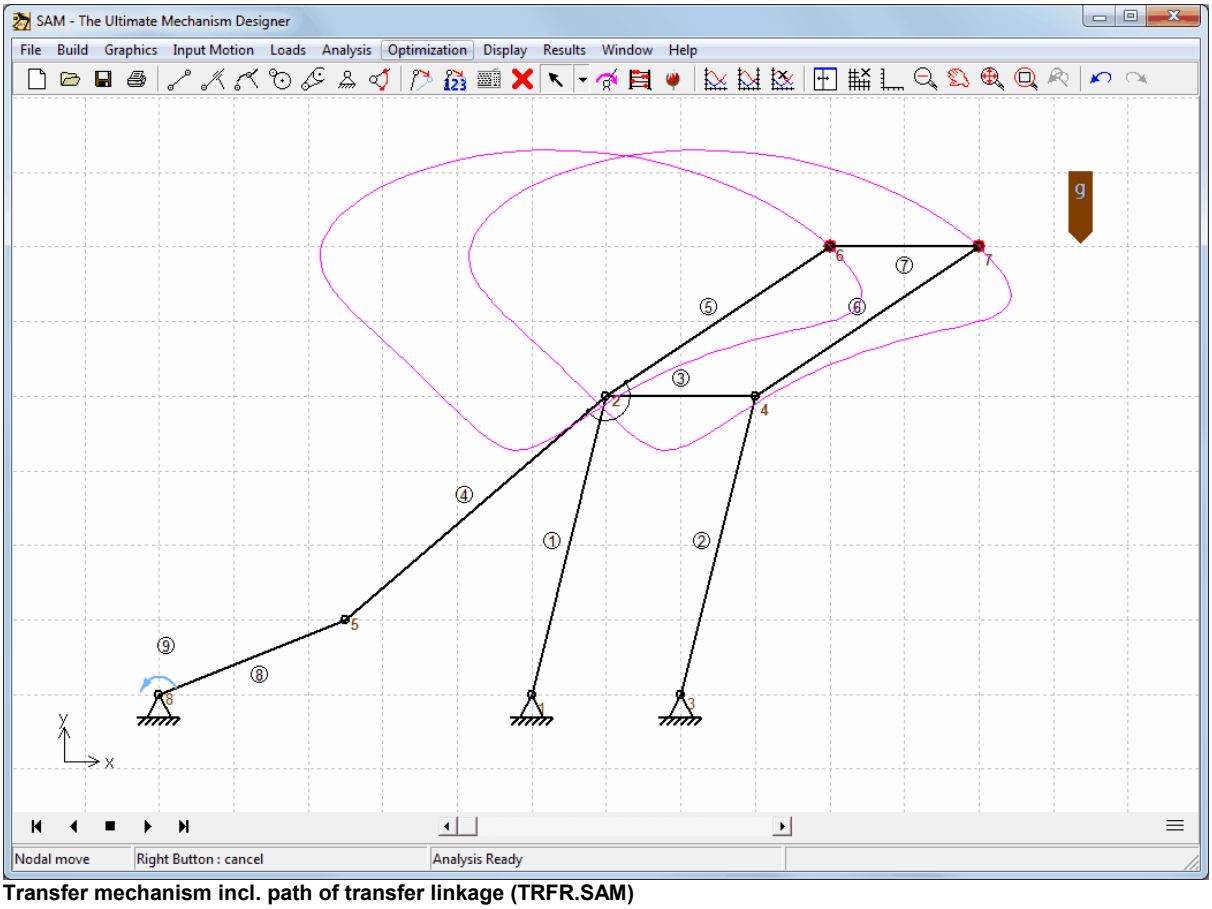
CAM Mechanism (CAM.SAM)

10.2 Force Analysis

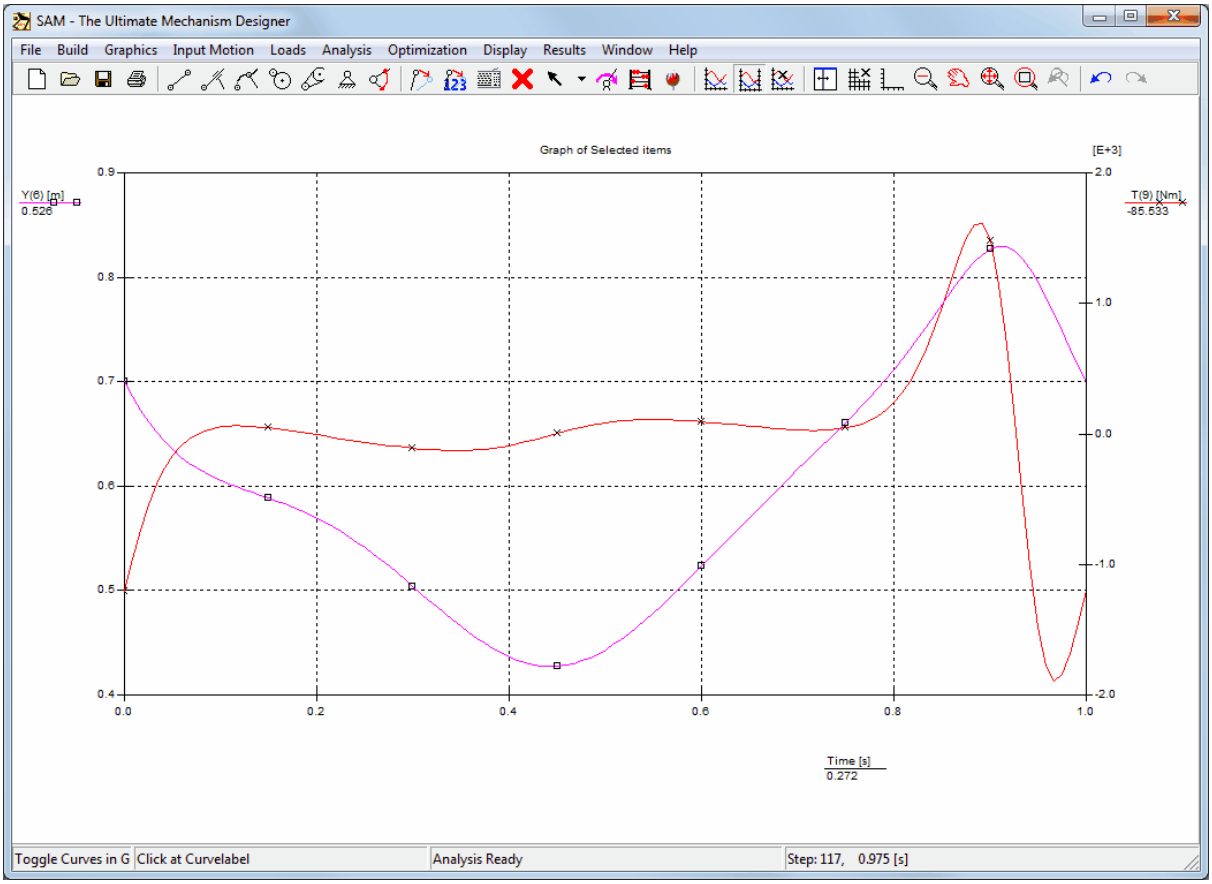
10.2.1 Transfer Mechanism (incl. gravity effects)

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.

© 2024 ARTAS - Engineering Software



Transfer mechanism incl. path of transfer linkage (TRFR.SAM)

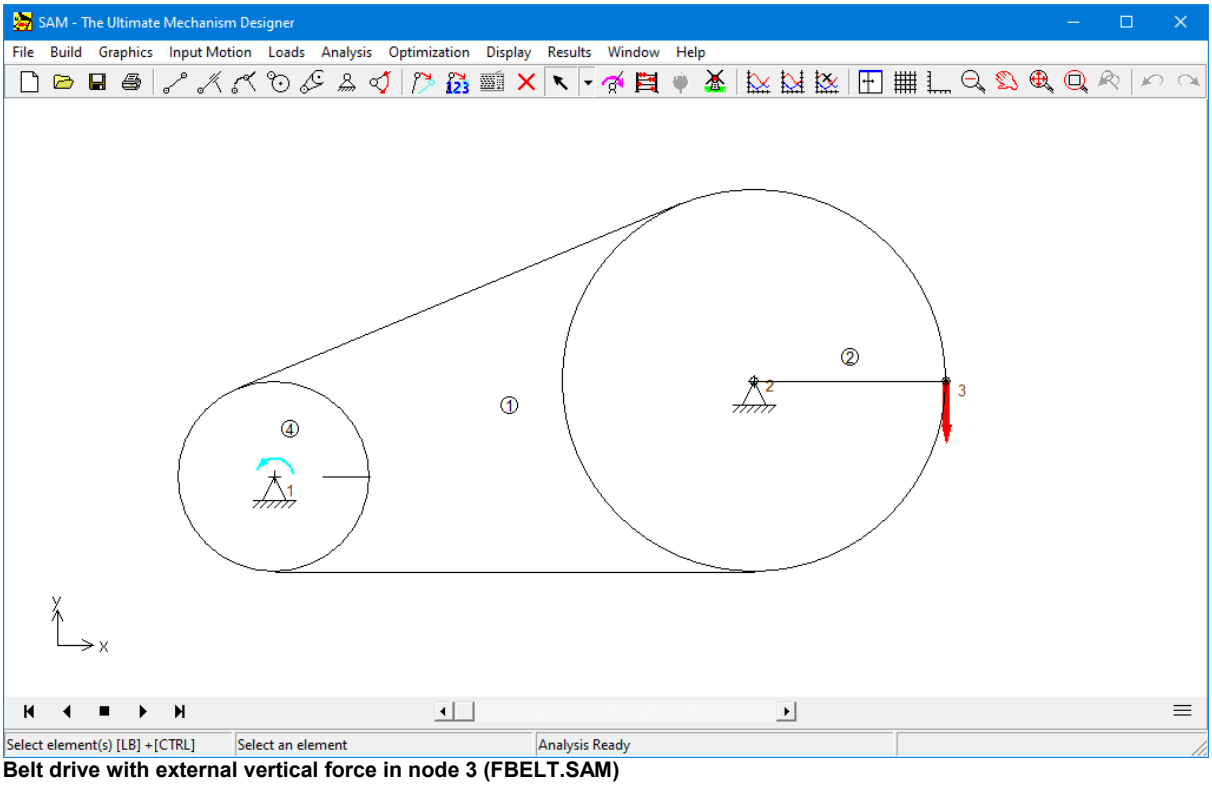


Vertical position transfer linkage & static driving torque

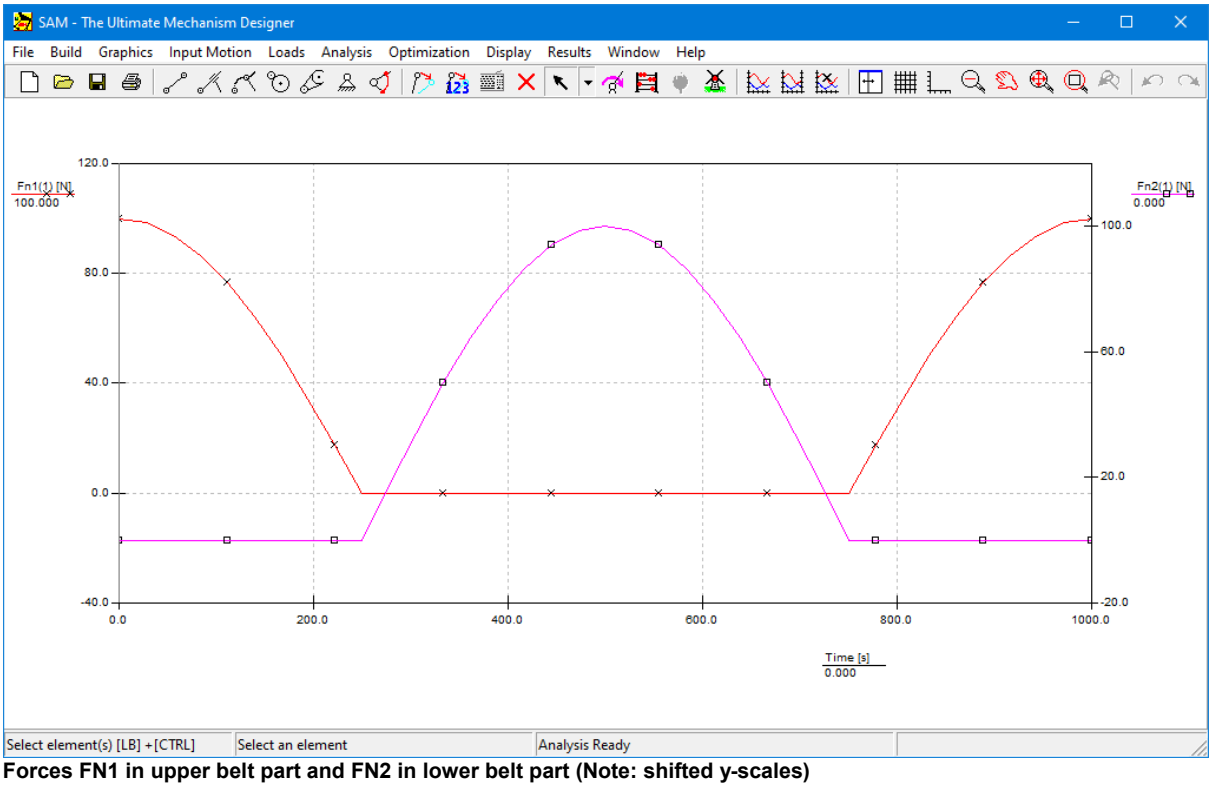
10.2.2 Belt drive with external force

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 beltdrive 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.

© 2024 ARTAS - Engineering Software



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

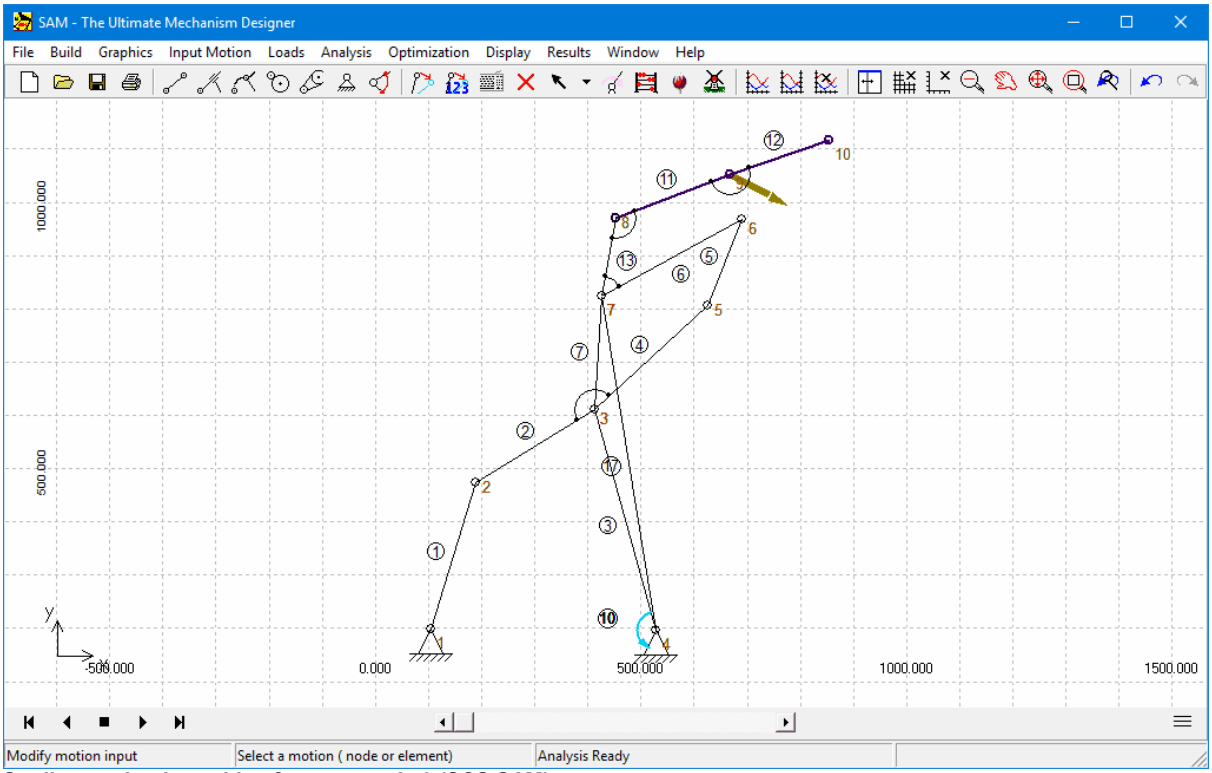


Forces FN1 in upper belt part and FN2 in lower belt part (Note: shifted y-scales)

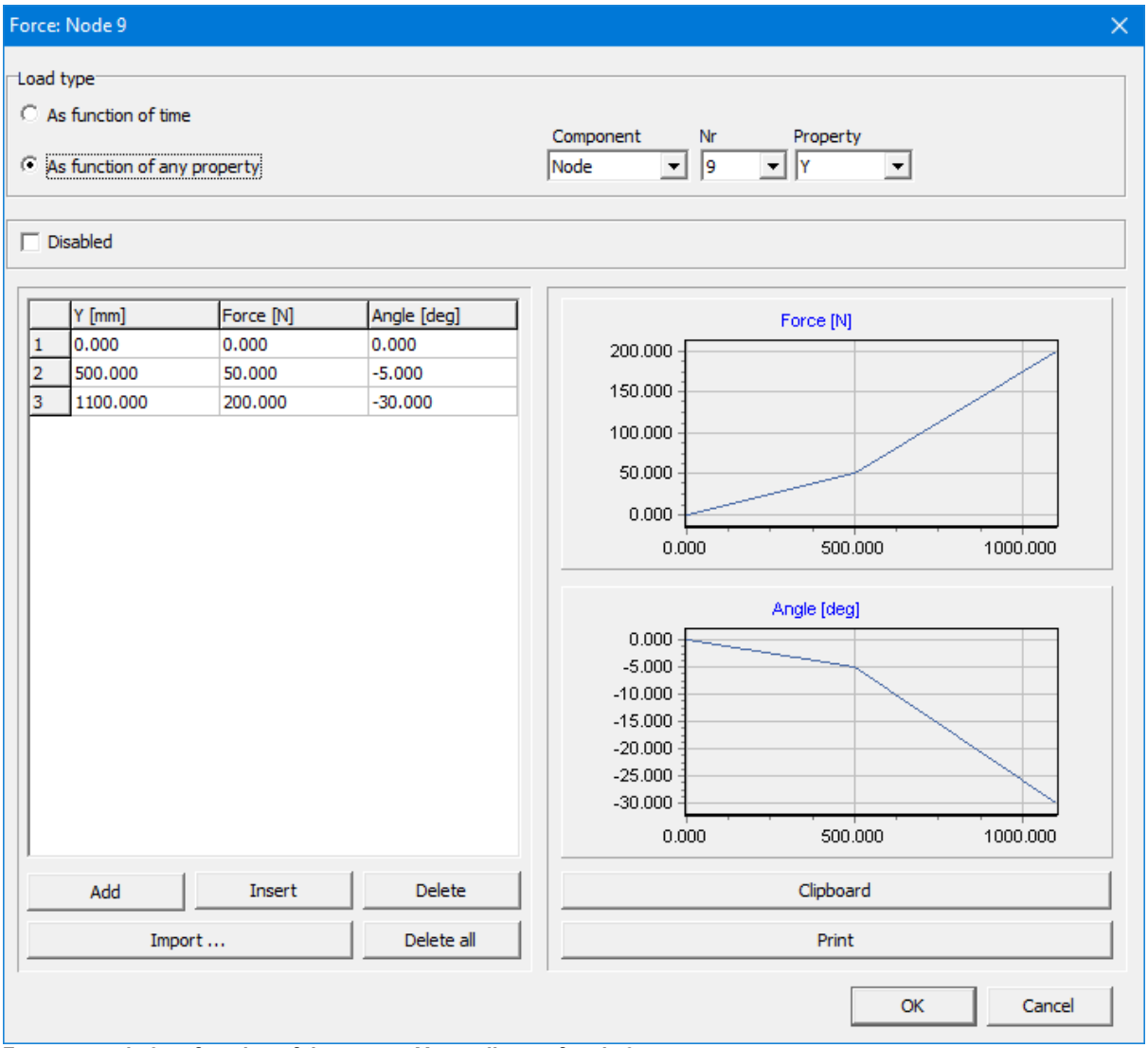
© 2024 ARTAS - Engineering Software

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

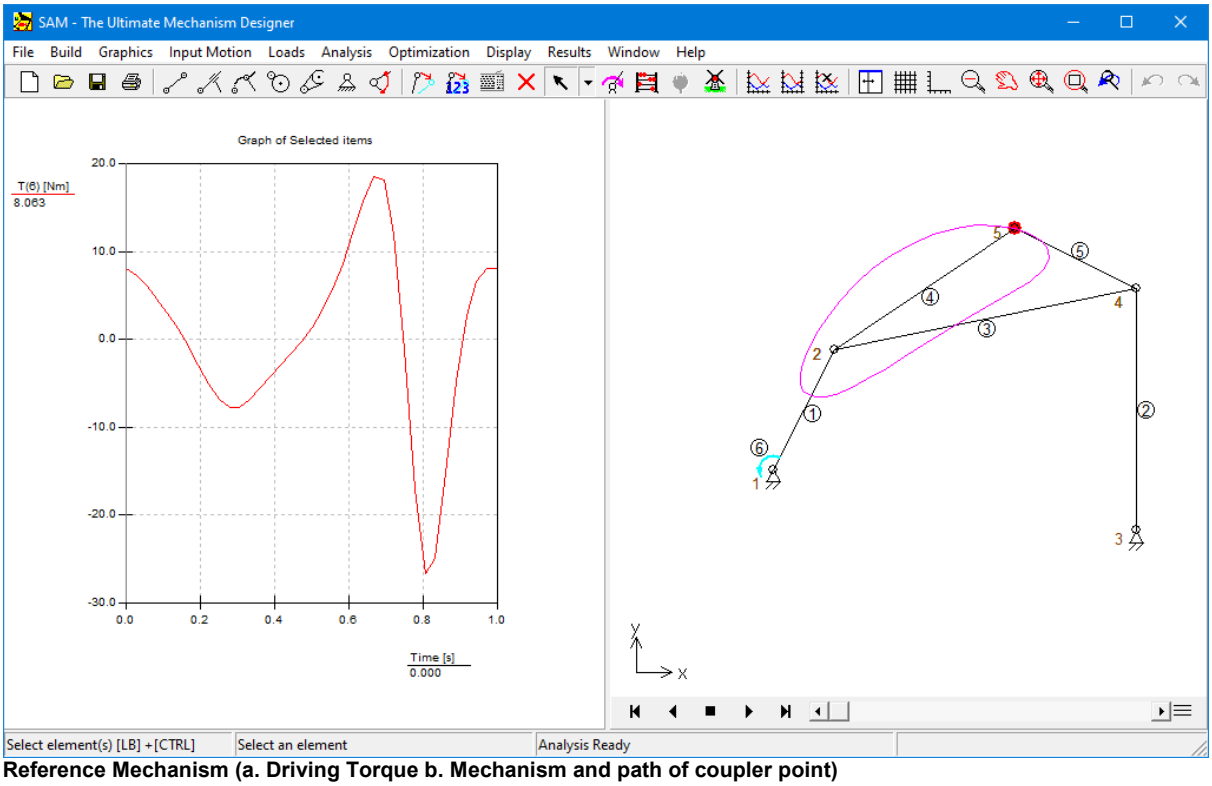
10.3.1 Driving Torque Minimization

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.

© 2024 ARTAS - Engineering Software



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

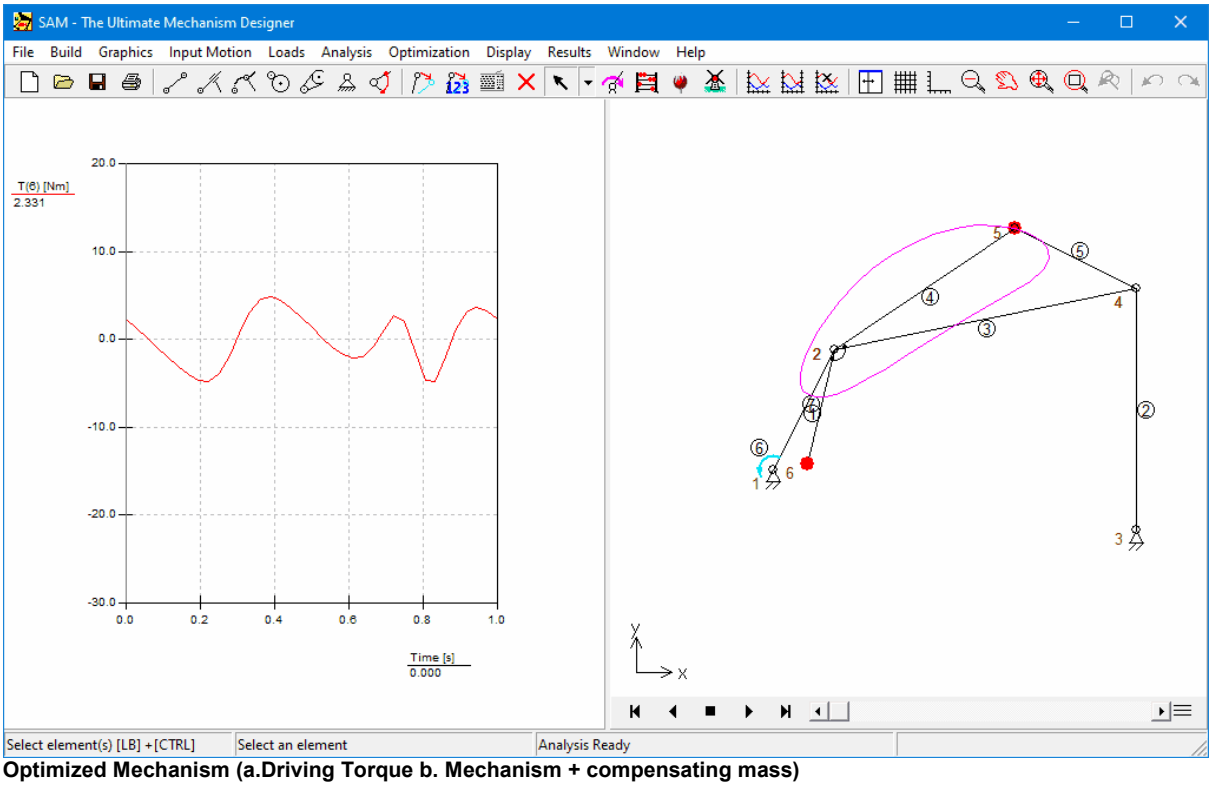
Optimization Parameter Range

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=0.257m and y=0.531m) 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.

© 2024 ARTAS - Engineering Software



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

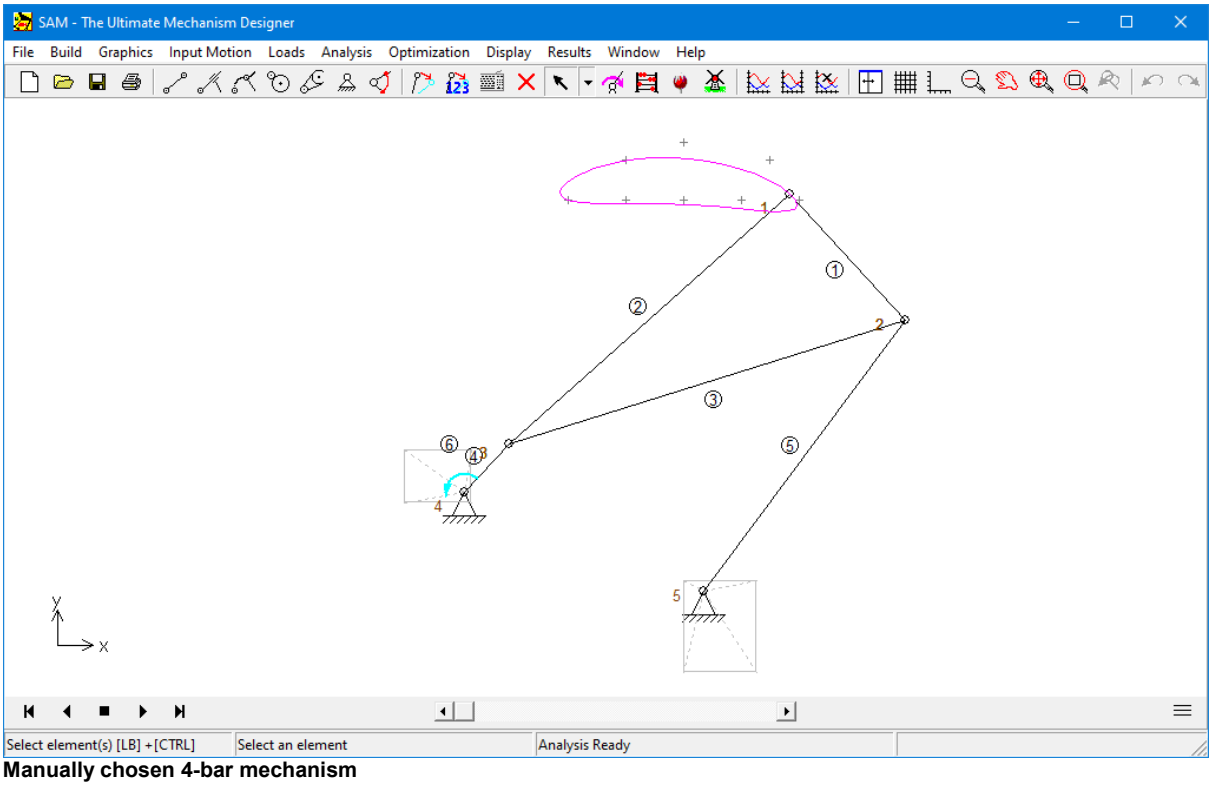
A significant reduction of the driving torque was achieved by adding a compensating mass to the coupler plane. The value of the compensating mass and its location was automatically determined by an optimization procedure based on an evolutionary algorithm.

10.3.2 Optimization of trajectory

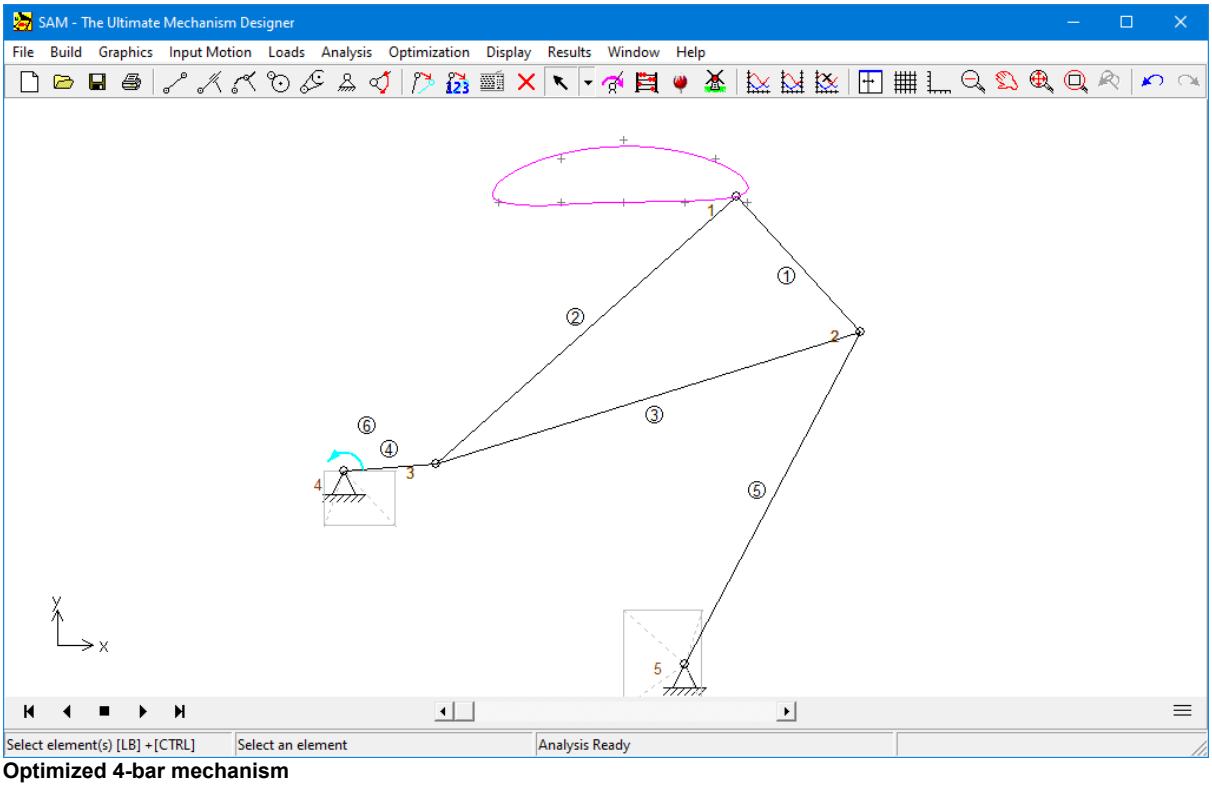
SAM can also optimize the parameters of a mechanism such that a desired trajectory is followed as good as possible. The first screenshot below shows the result of a manual attempt to find a 4-bar mechanism with a coupler point motion that mimics the Bezier curve through 8 specified points.

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.

© 2024 ARTAS - Engineering Software



Manually chosen 4-bar mechanism



Optimized 4-bar mechanism

© 2024 ARTAS - Engineering Software

10.3.3 Constrained Optimization

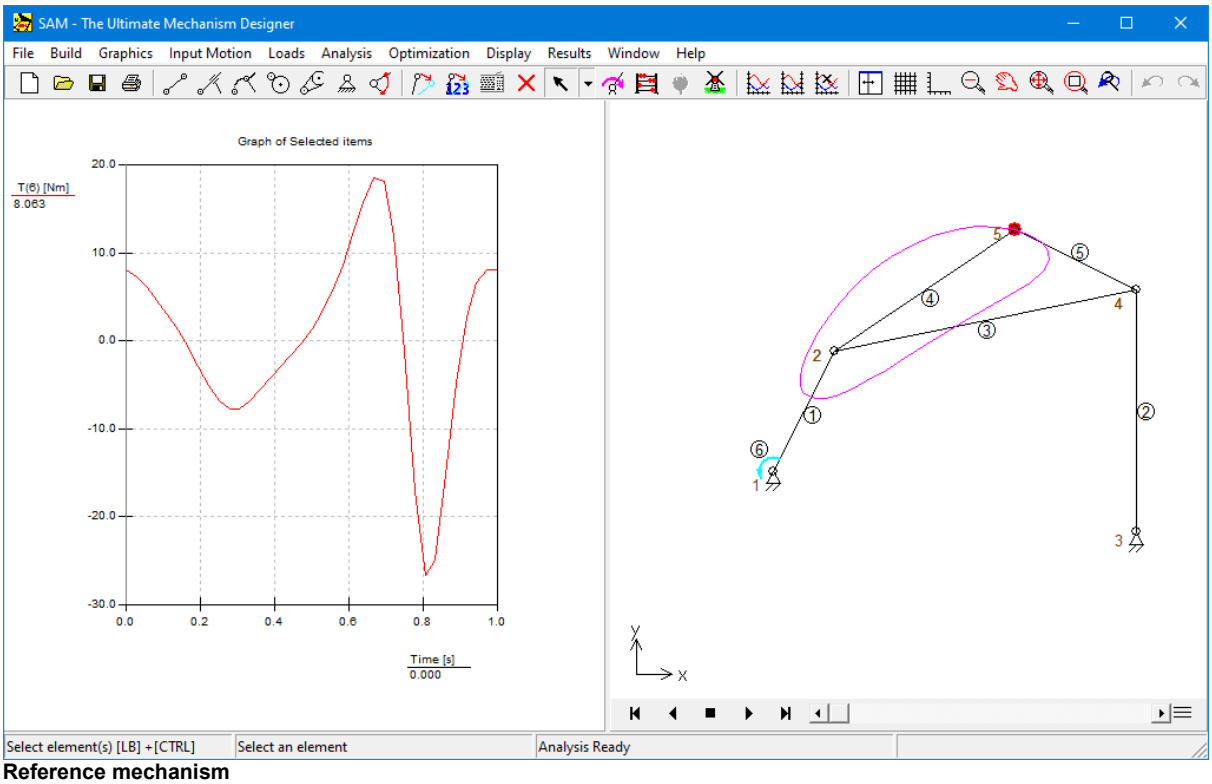
In this example we will show constrained optimization by adding a penalty function to the original cost function in order to incorporate the constraint.

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

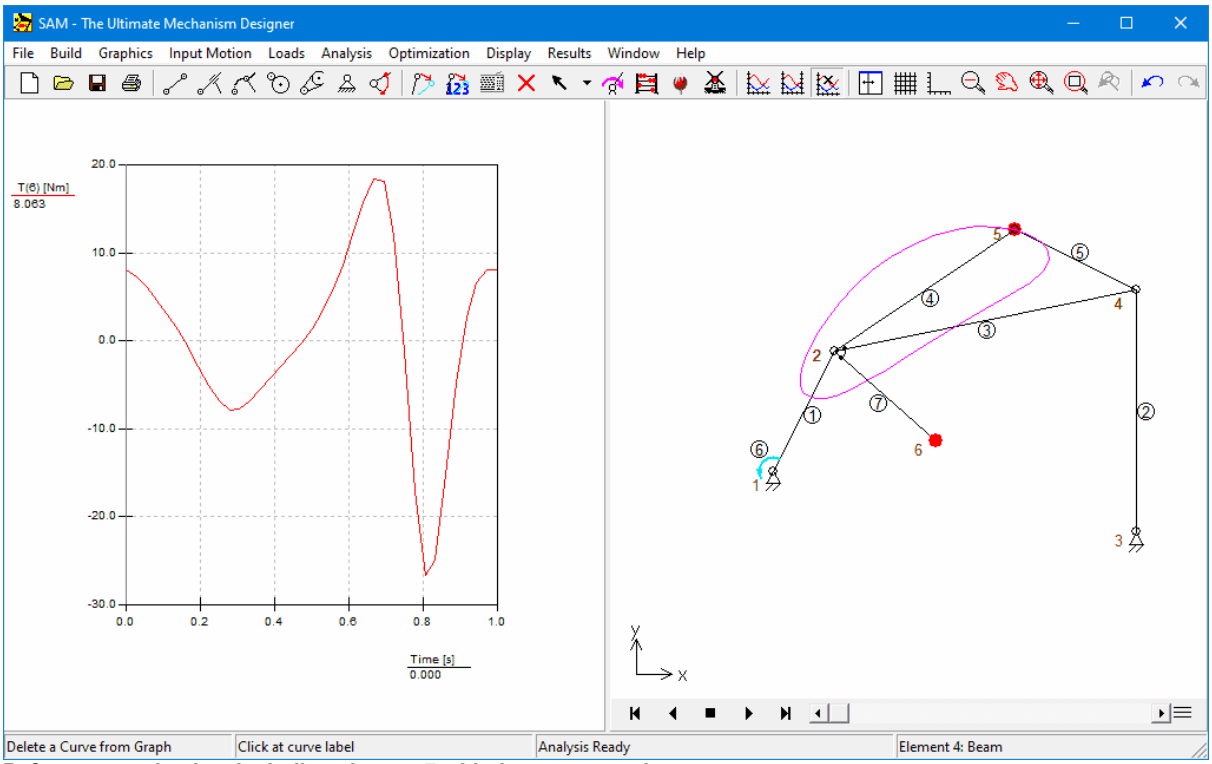
In the reference situation the driving torque as a function of time will look like depicted in the following picture. The absolute maximum of the driving torque equals 26.6Nm.



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.

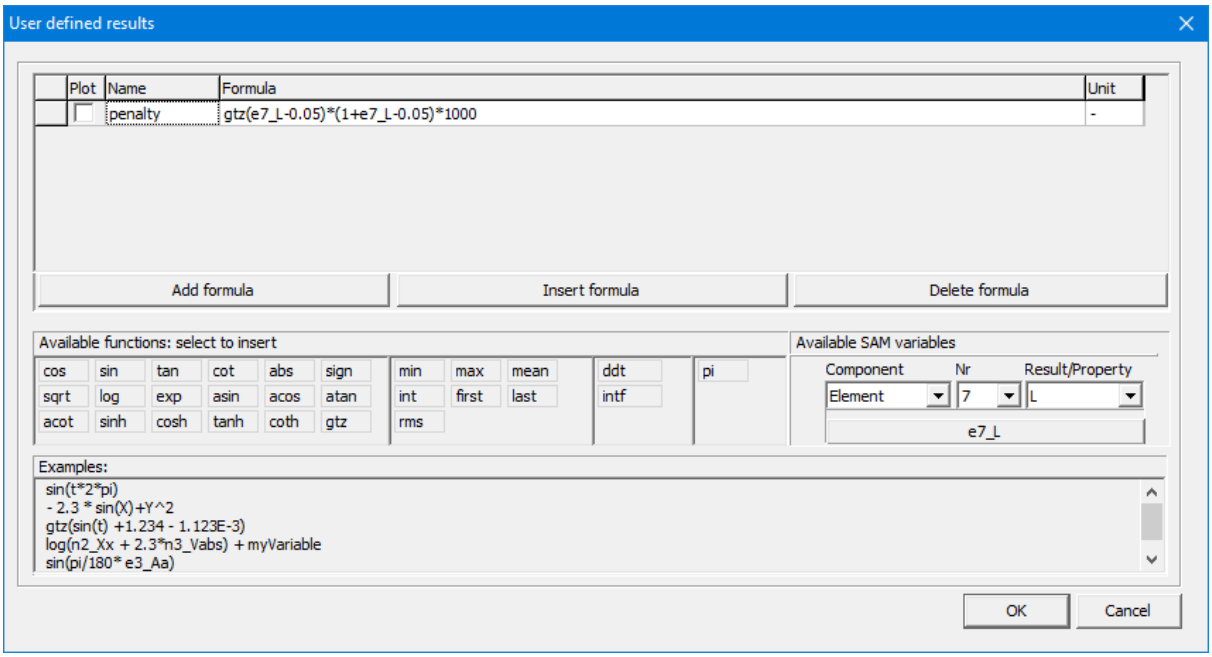
© 2024 ARTAS - Engineering Software



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...



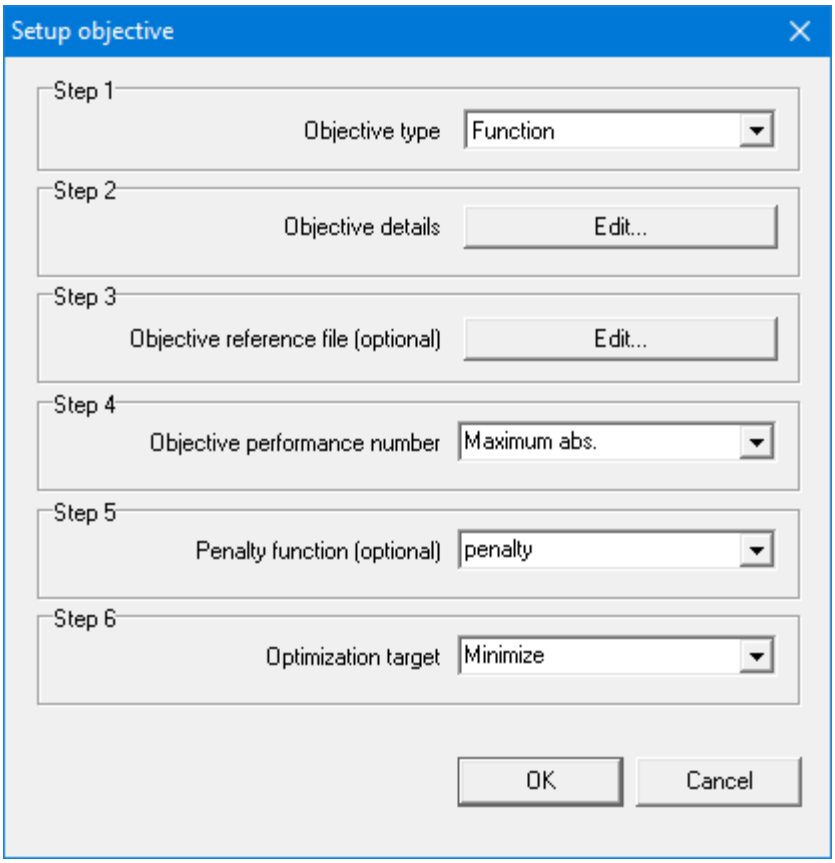
The constraint of the length of element 7 is converted into a penalty

© 2024 ARTAS - Engineering Software

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>=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

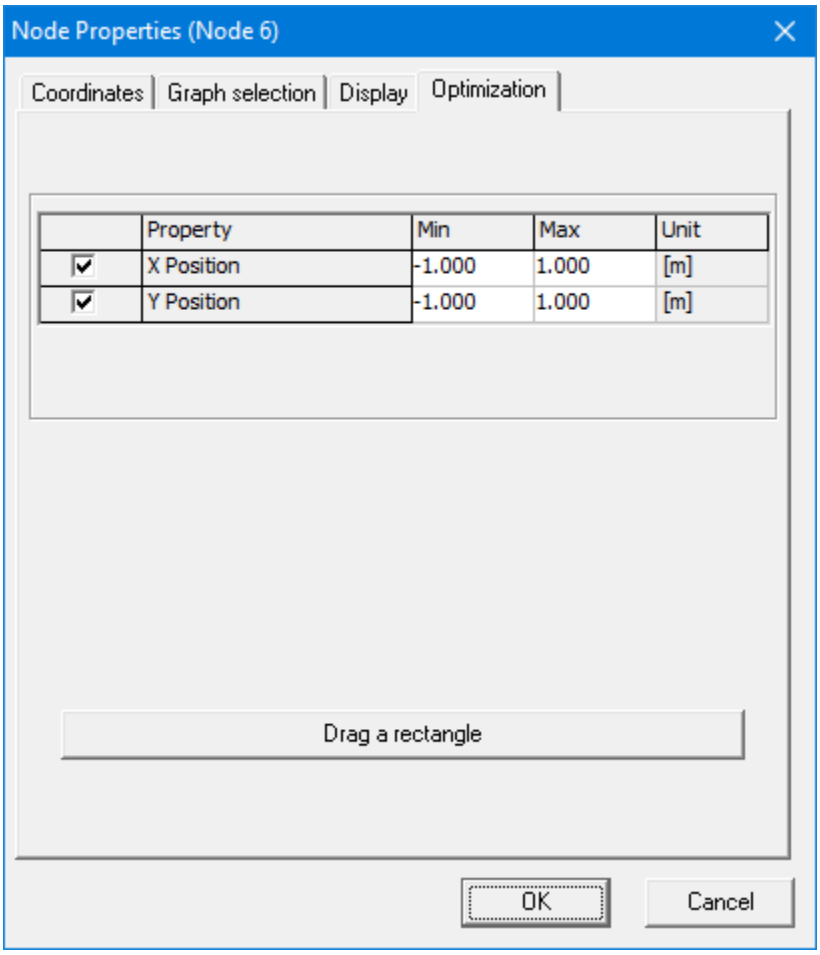
The settings of the optimization are shown in the following screenshot. The maximum absolute value of torque T1 of element 1 is chosen as objective to minimize given the constraint that the length of element 6 should be smaller than 0.05m. This constraint is translated into a penalty that is added to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



Optimization Parameter Range

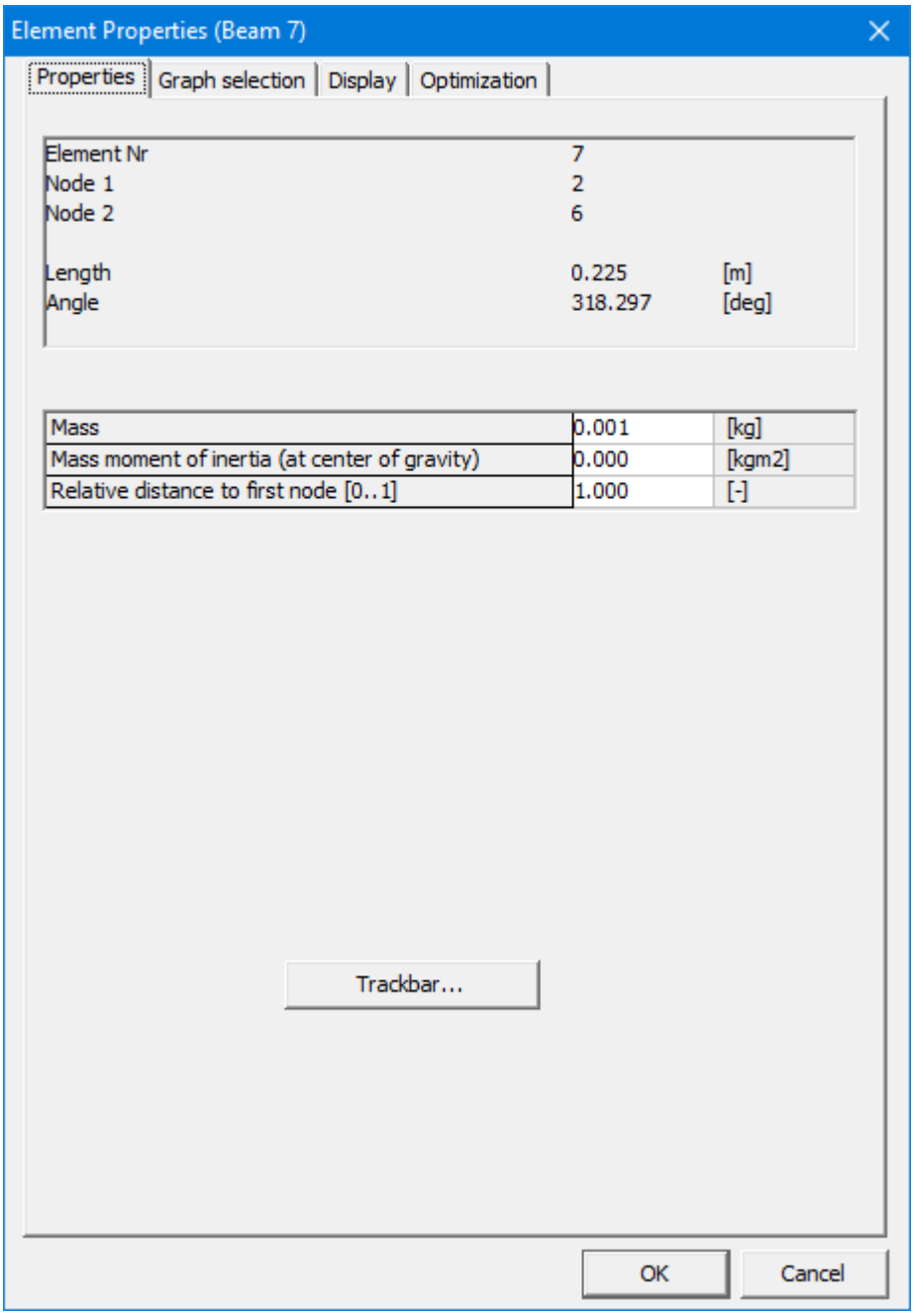
The maximum value of the driving torque (or actually the maximum value of the cost function) is reduced by adding a single mass to the coupler linkage. The search region for the location was set at (-1.0,-1.0) up to (1.0,1.0).

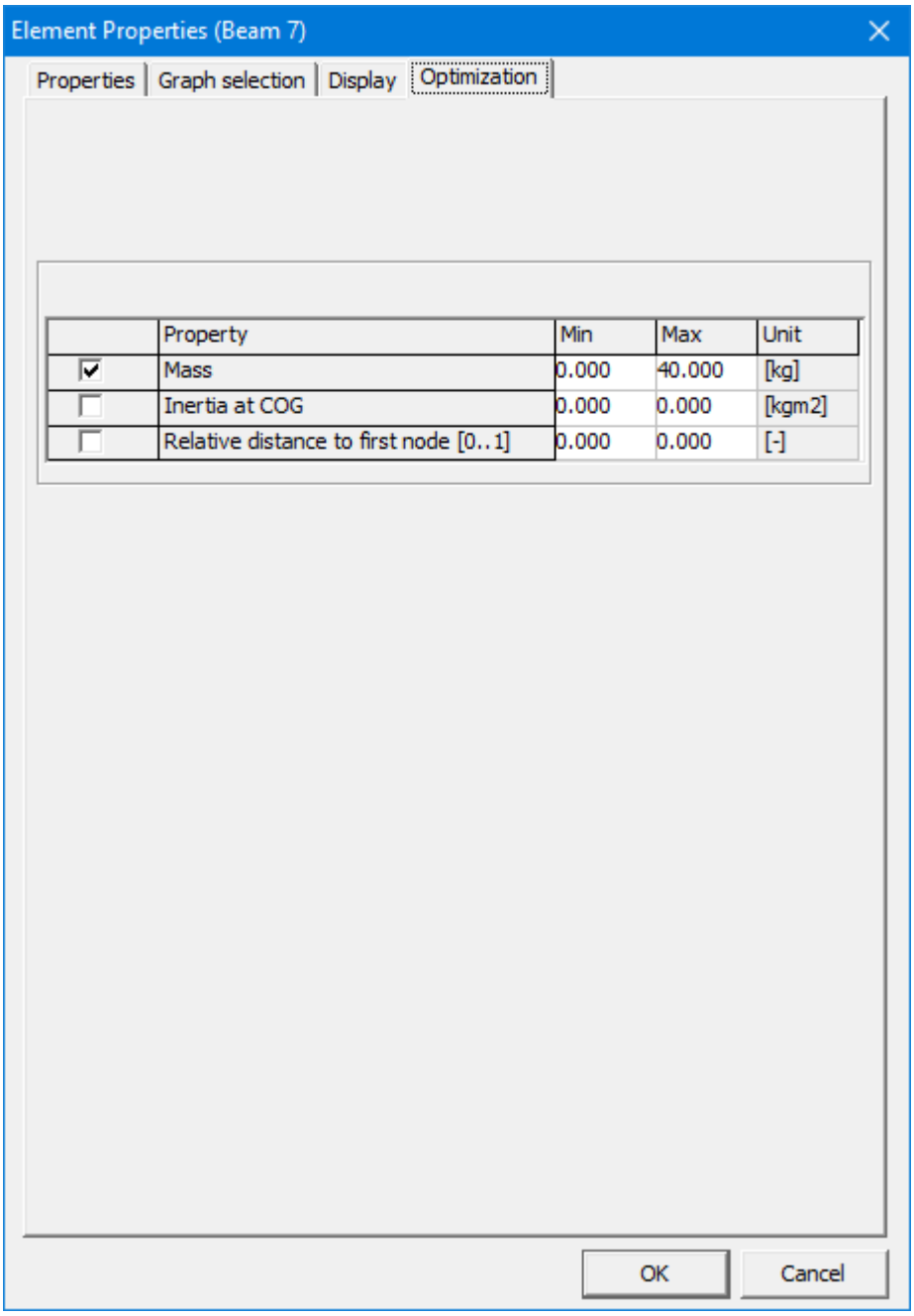
© 2024 ARTAS - Engineering Software



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).

© 2024 ARTAS - Engineering Software


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

© 2024 ARTAS - Engineering Software

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.

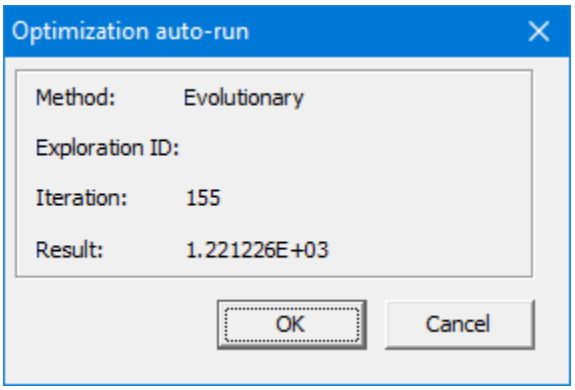
In principle the boundaries of the optimization space should always be respected. However, sometimes it is very interesting – and one may be triggered to find new solutions – to allow the algorithm to disrespect these boundaries. This is the reason why the user has a choice.

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.

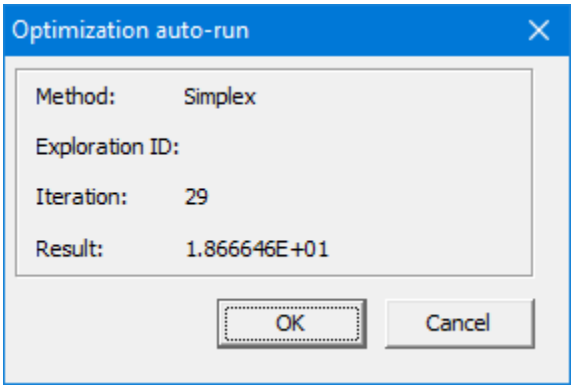


Method: The method for global exploration is always the Evolutionary Algorithm.
Explore id: Empty (has only meaning when doing the local optimization).

© 2024 ARTAS - Engineering Software

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:

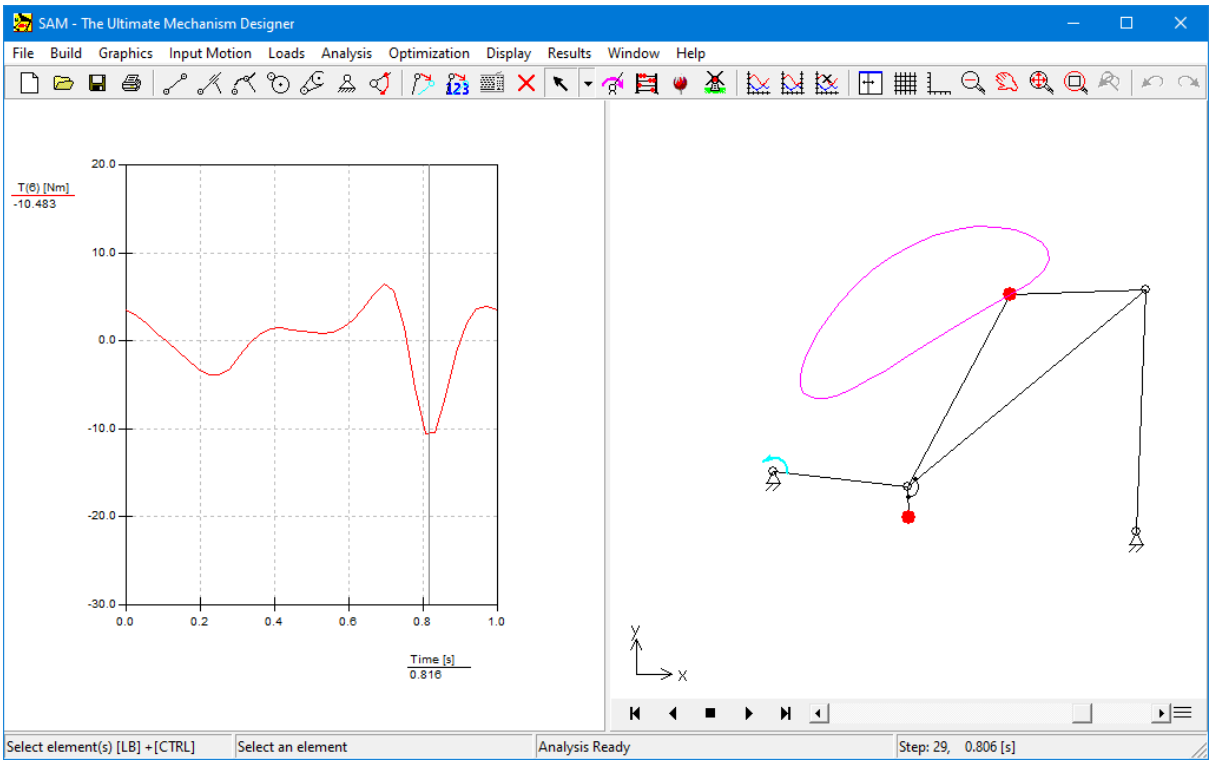


Method: Chosen method for local optimization
Explore id: ID of the solution of the global exploration that is currently locally optimized.
Iteration: Current iteration step in the local optimization process
Result: Current value of the optimization goal.

Optimized Mechanism

The value (m=39.875kg) and location (x=0.28m and y=0.65m) 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.

© 2024 ARTAS - Engineering Software



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).

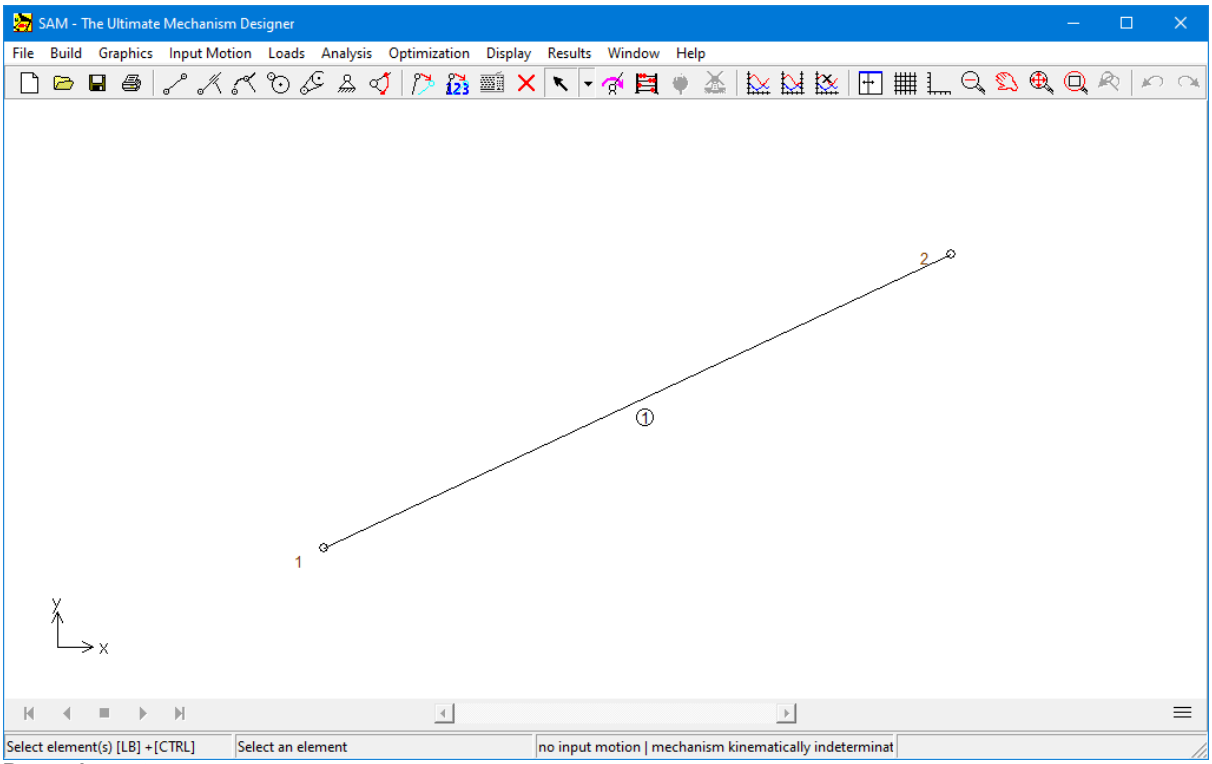
© 2024 ARTAS - Engineering Software

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.



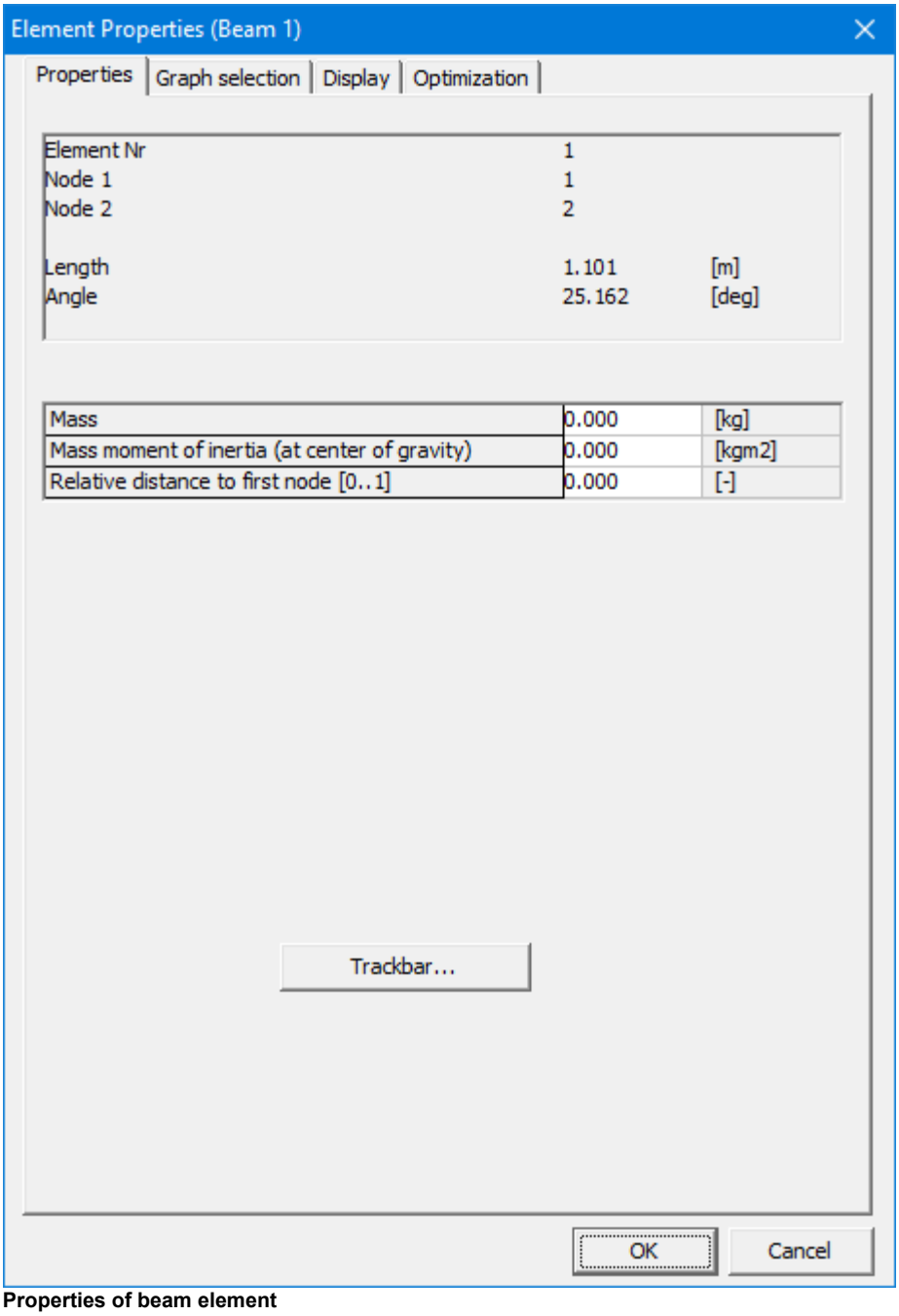
Beam element

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.

© 2024 ARTAS - Engineering Software



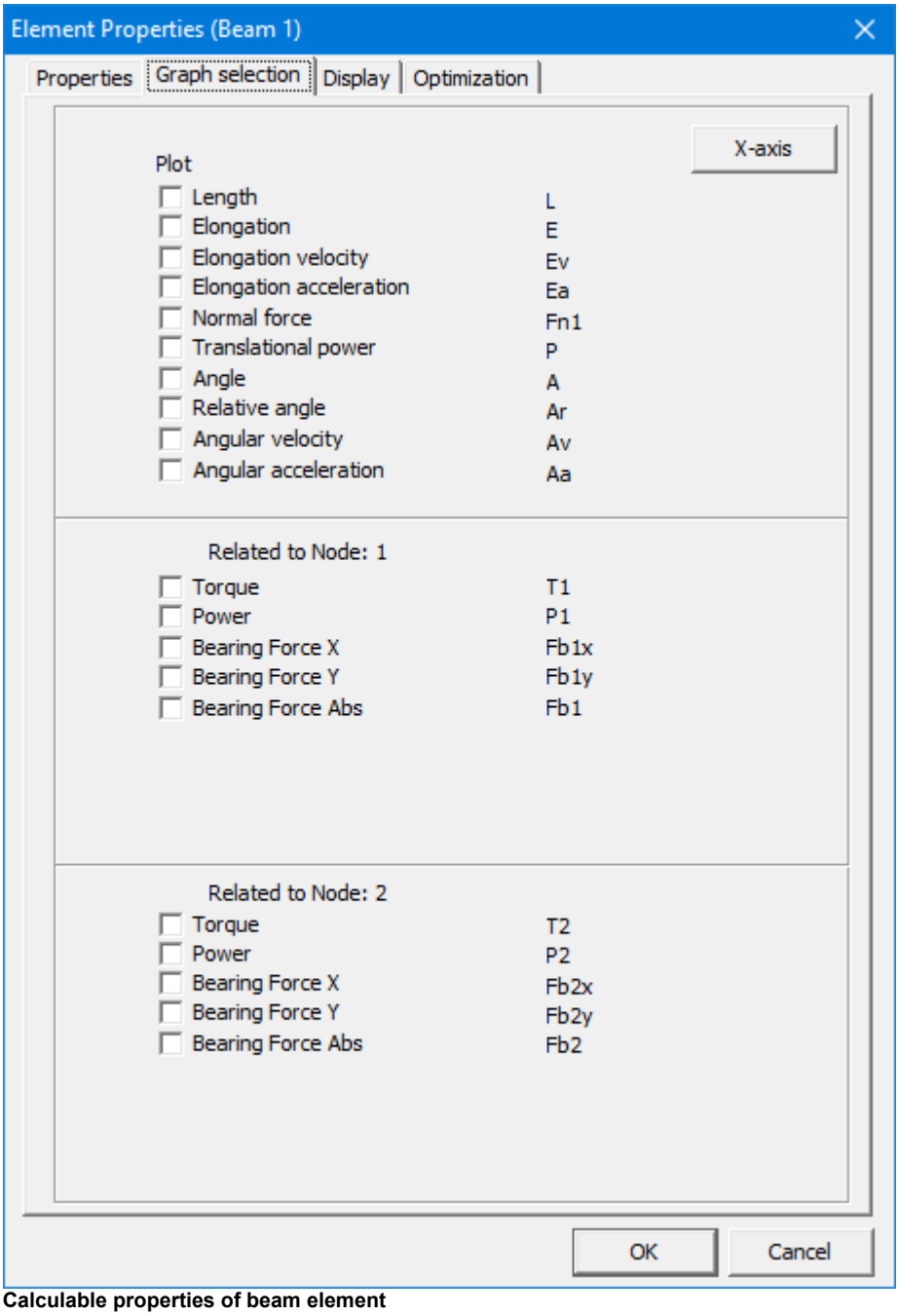
Properties of beam element

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

© 2024 ARTAS - Engineering Software

- torque and corresponding power at both nodes
- force (x, y and amplitude) that is exerted on the adjacent bearing



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.

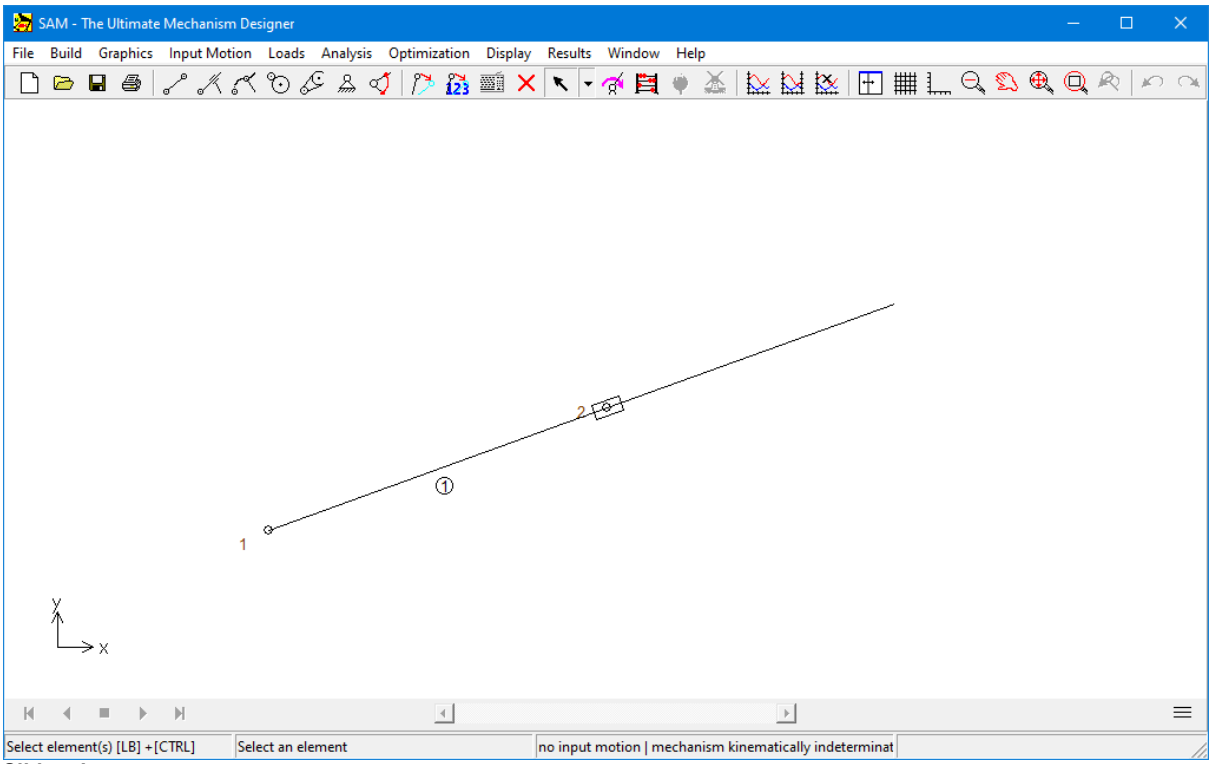
In most applications, the length of a beam element is fixed and as a result of that, the elongation and its derivatives are equal to zero. In the absence of elongation, the transmitted power in axial direction is also equal to zero (power=force*velocity).

© 2024 ARTAS - Engineering Software

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. It can also include coulomb friction, which is defined by: no-load friction + bearing force * friction coefficient. In order to achieve better graphical visualization, the user can specify the length of the slider.

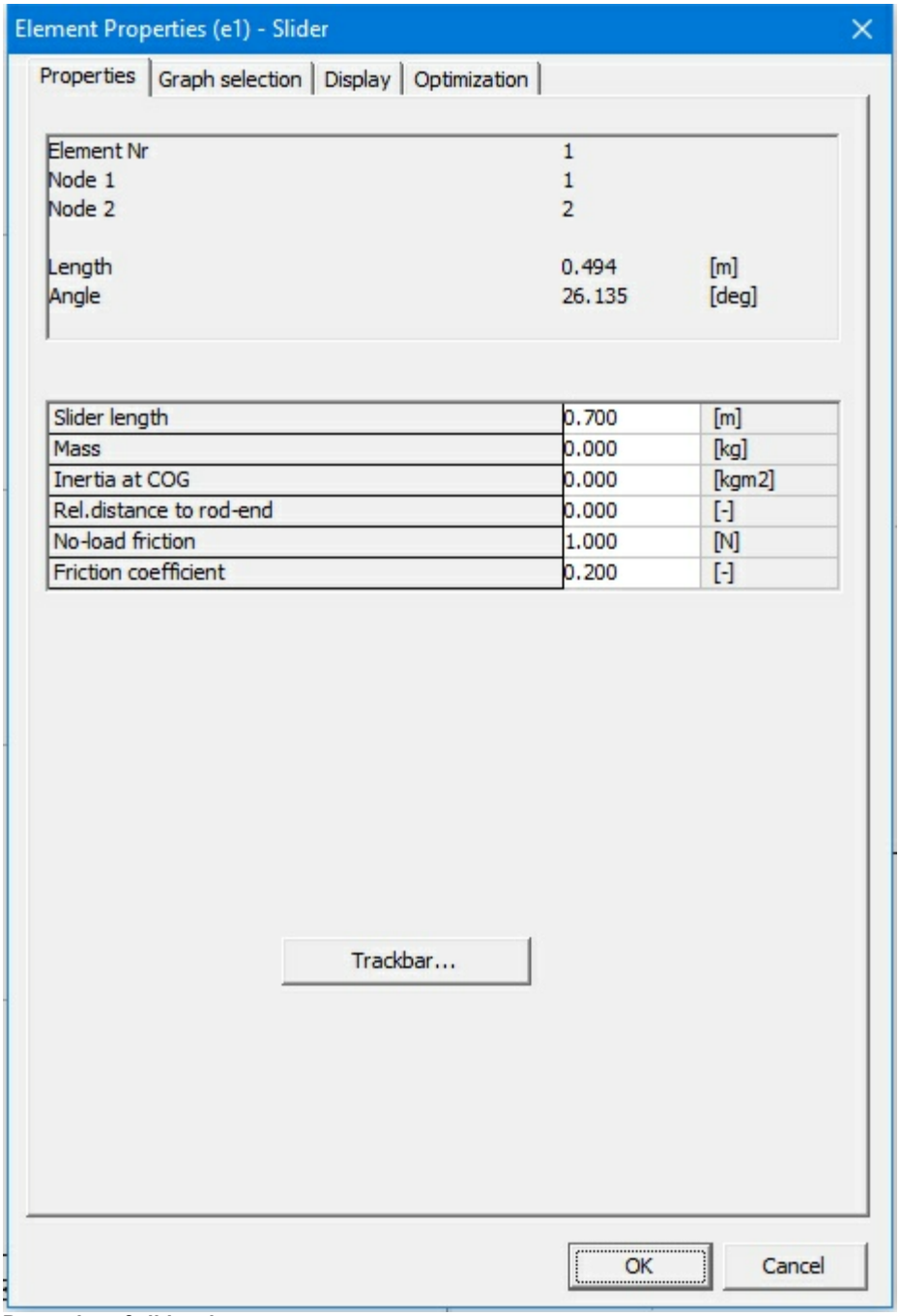


Slider element

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
- mass moment of inertia w.r.t. center of gravity
- location center of gravity (defined as relative distance w.r.t. the first node and it ranges from 0-1. The location is therefore restricted to a position on the slider.mass moment of inertia w.r.t. center of gravity
- No-load friction (= constant friction force in the absence of any bearing force)
- Friction coefficient

© 2024 ARTAS - Engineering Software

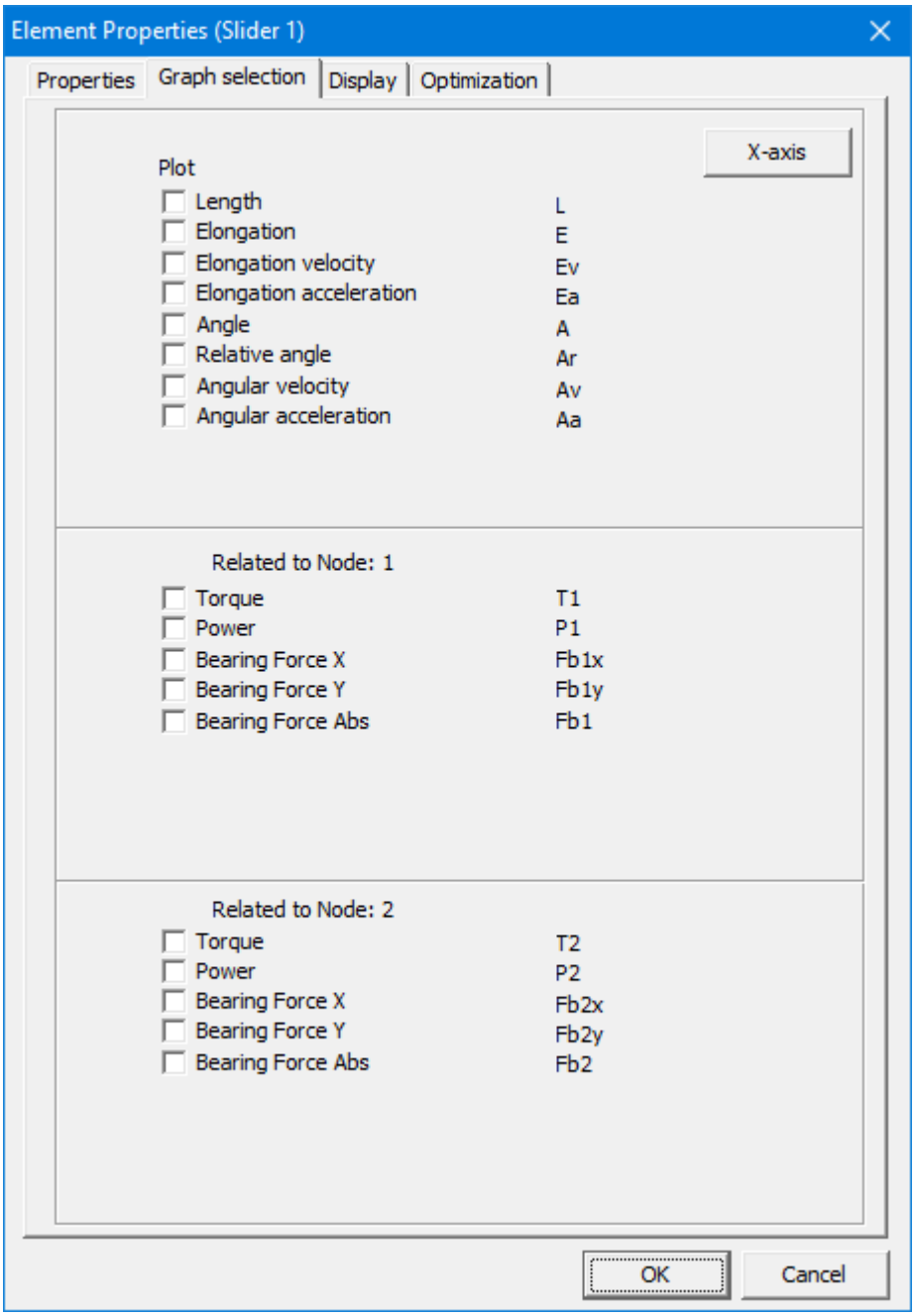


Properties of slider element

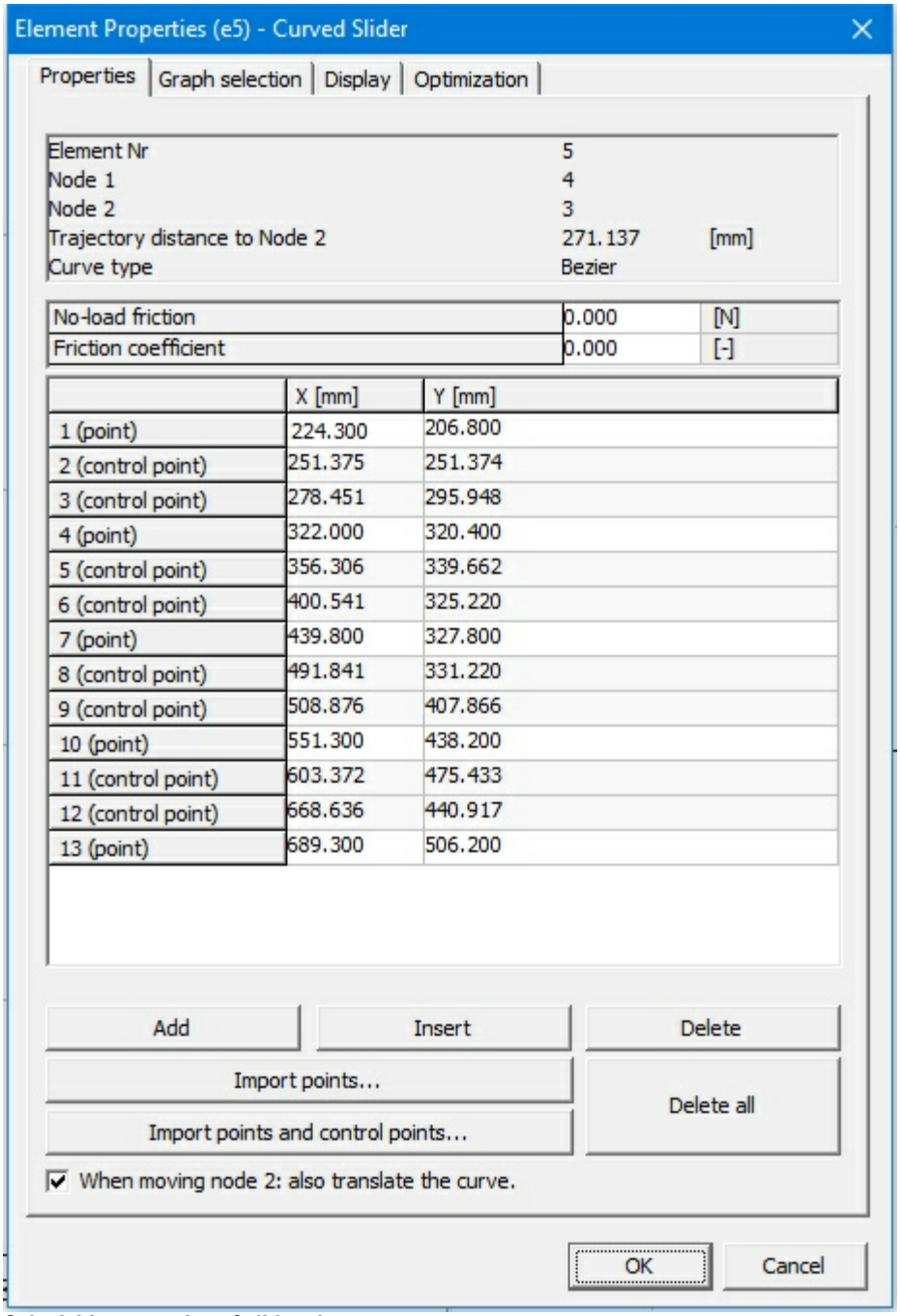
The following items can be selected as result of the analysis process :

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

- force (x,y and amplitude) that is exerted on the adjacent bearing



© 2024 ARTAS - Engineering Software



Calculable properties of slider element

The output selection dialog of the slider is equal to that of a beam element, except that the items "normal force" and "normal power" are not available. 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,

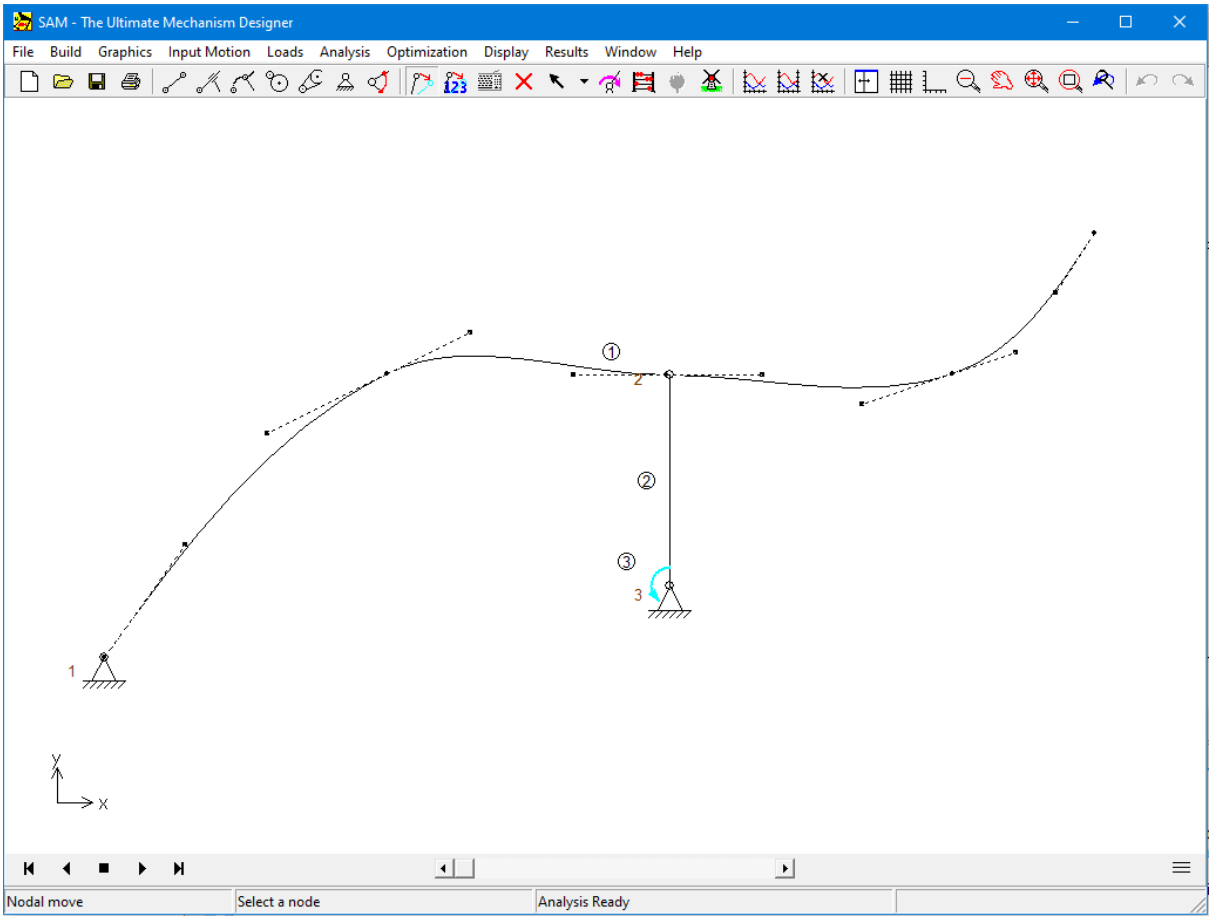
© 2024 ARTAS - Engineering Software

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. Two parameters define the friction in the direction of the movement. First a constant force, called No-load friction, that is present even in the absence of a force perpendicular to the motion direction. Second, a part that is proportional to the force perpendicular to the motion direction multiplied with the friction coefficient. Friction force = no-load friction + load * friction coefficient

The curved slider element does not have any inertia properties of it 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

© 2024 ARTAS - Engineering Software

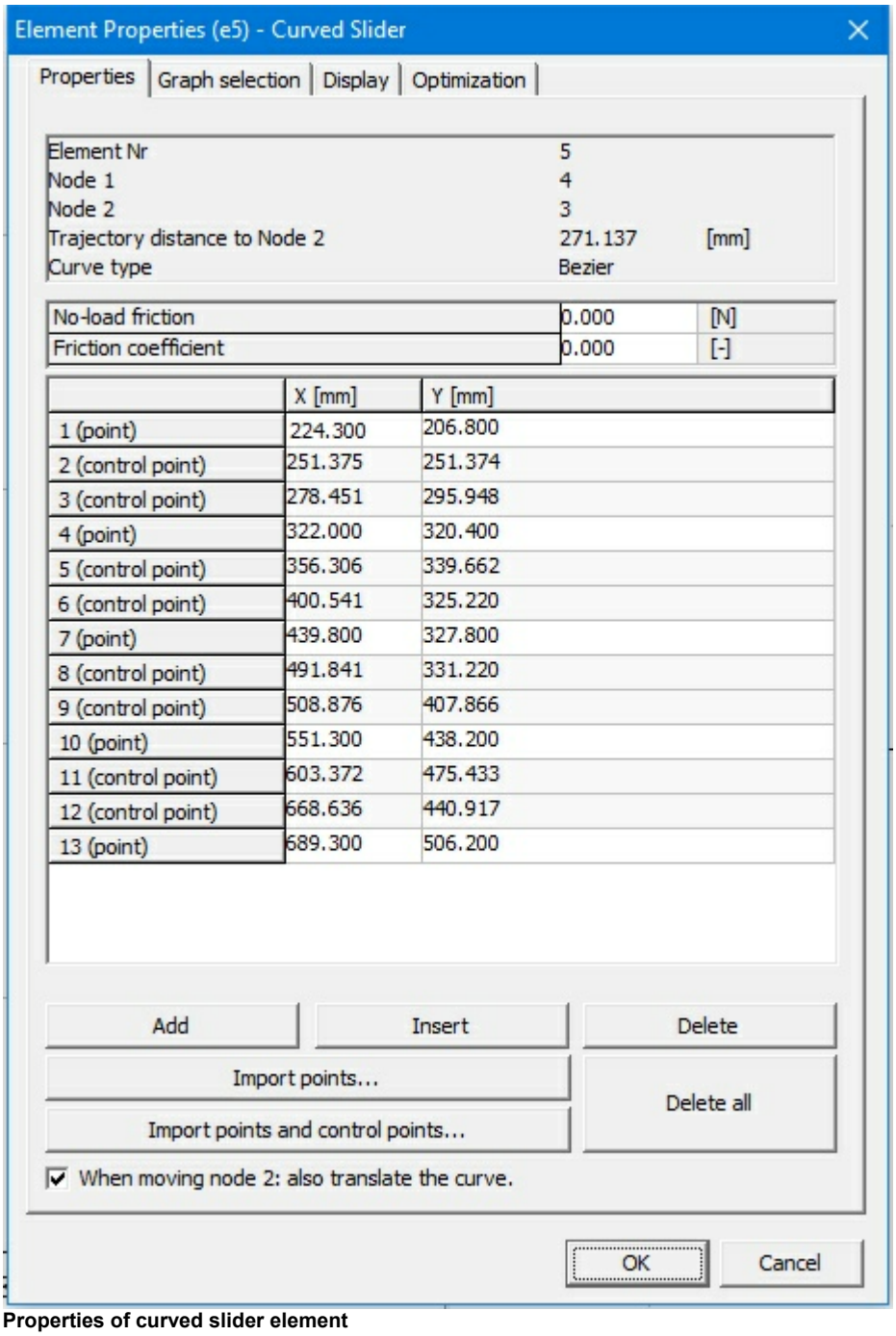
is also possible to import data from a text file that contains a list of x/y coordinates. One can choose between:

- **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.

Note 1: If the last point in an import file coincides with the first point, the curve is automatically considered a closed Bezier curve.

Note 2: The total number of points describing a curve is limited to 1000. This is including the control points that are automatically generated when Bézier points are imported. Therefore, if you only import Bézier points, the file should contain no more than 333 points.

© 2024 ARTAS - Engineering Software

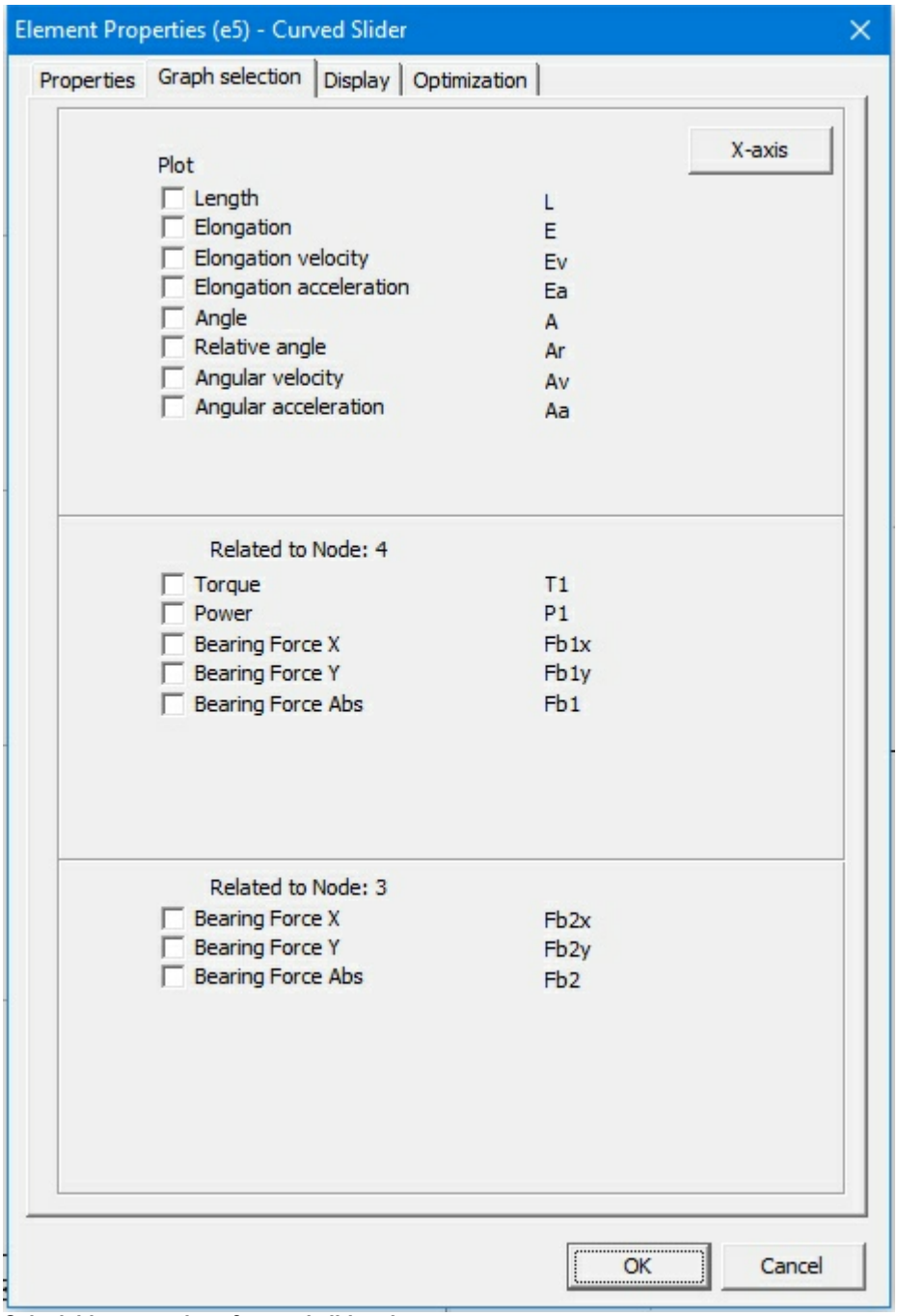


Properties of curved slider element

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
- force (x, y and amplitude) that is exerted on the adjacent bearing

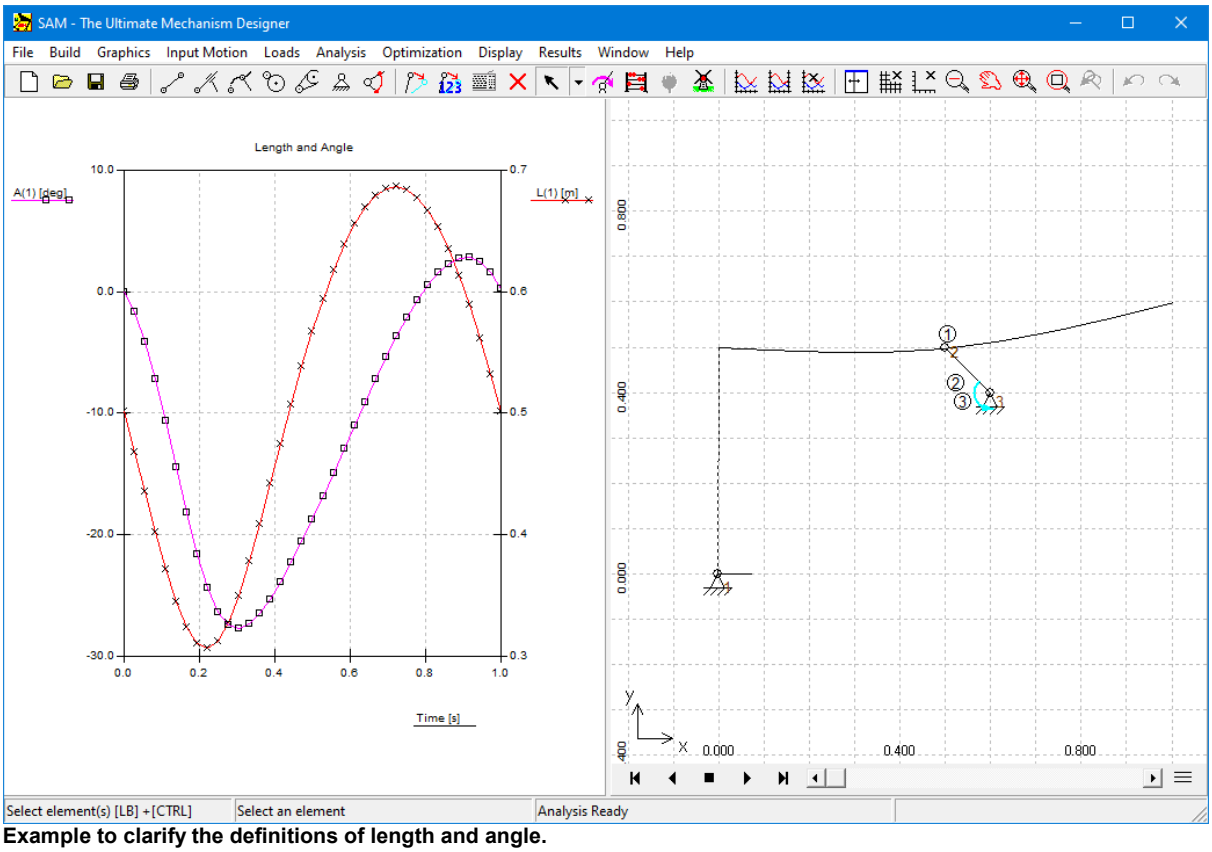
© 2024 ARTAS - Engineering Software



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].

© 2024 ARTAS - Engineering Software

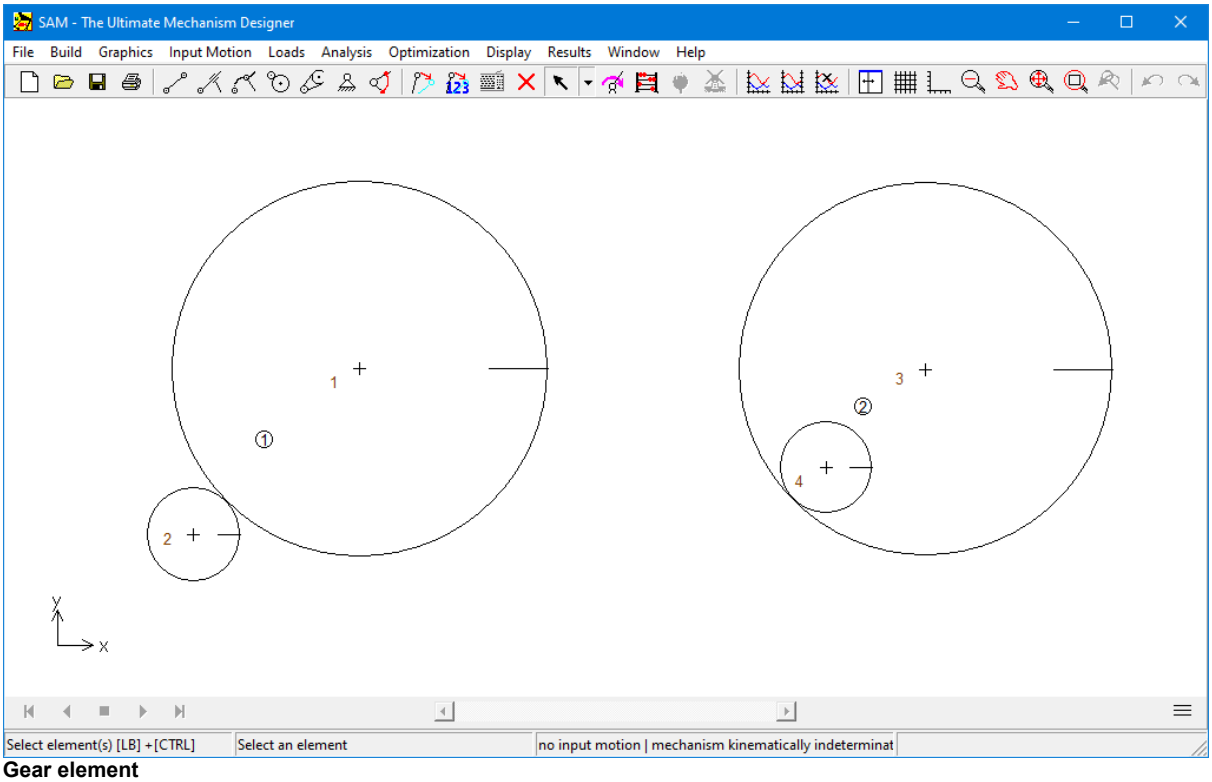


Example to clarify the definitions of length and angle.

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.

© 2024 ARTAS - Engineering Software

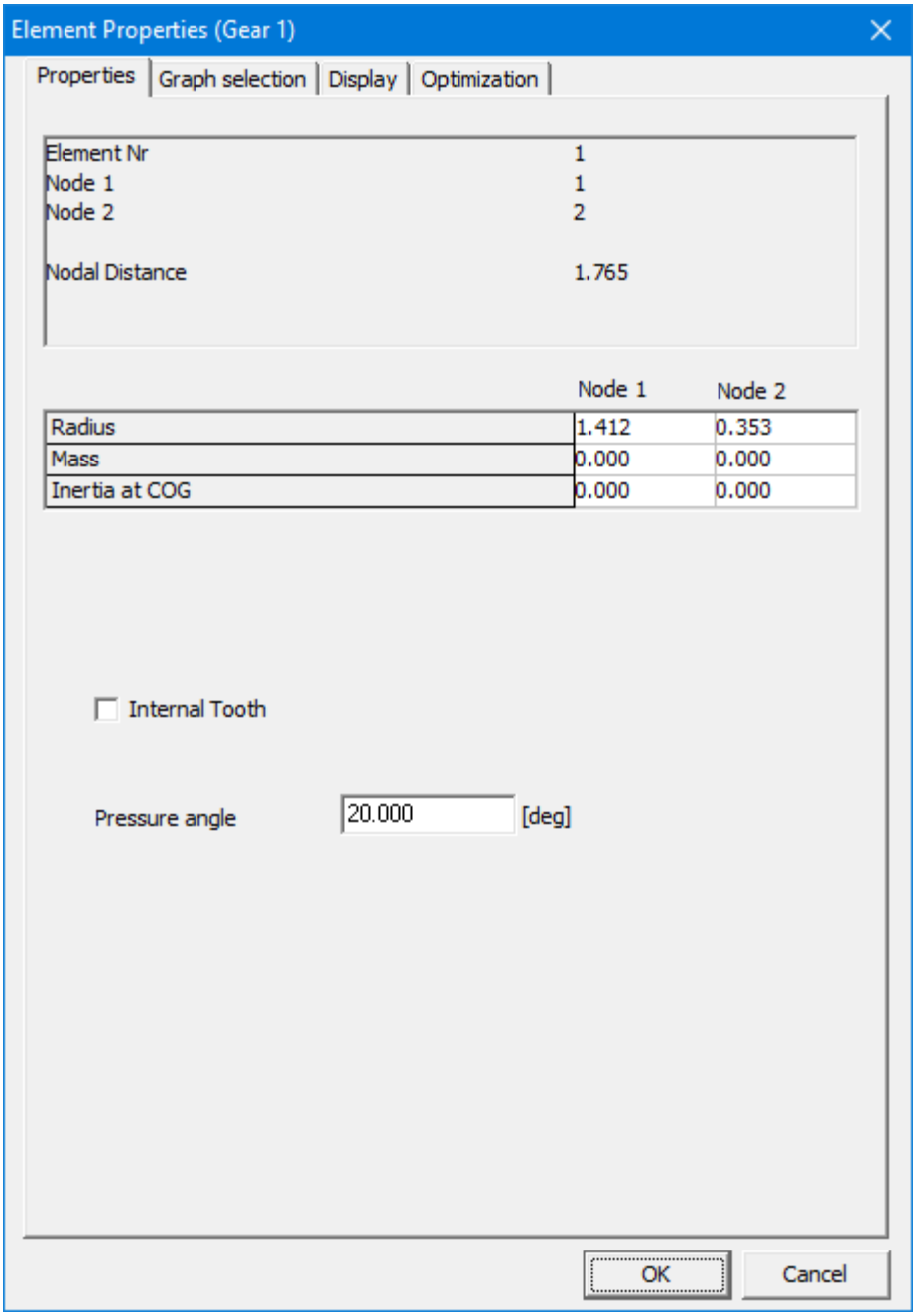


Gear element

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

© 2024 ARTAS - Engineering Software



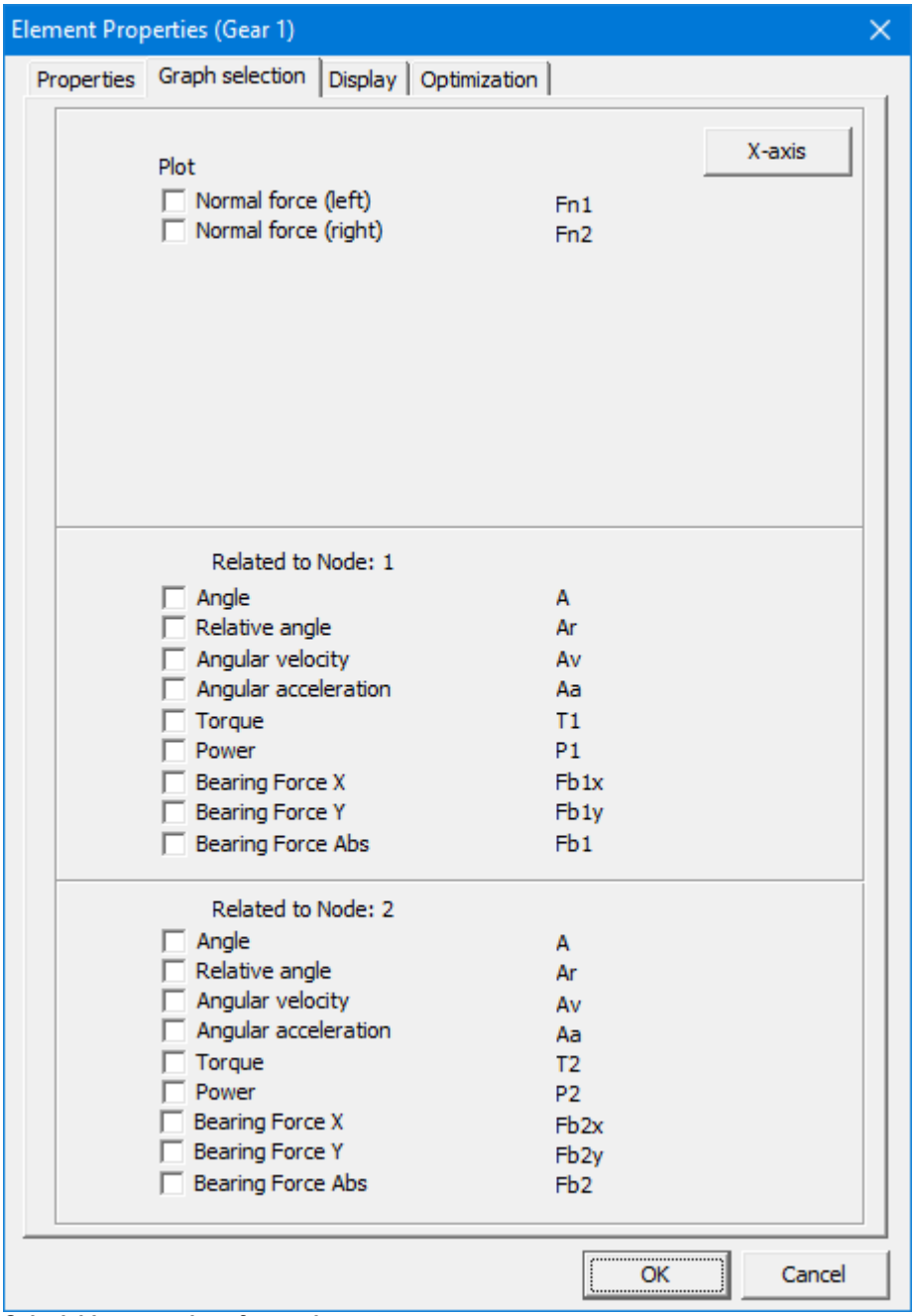
Properties of gear element

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.

© 2024 ARTAS - Engineering Software

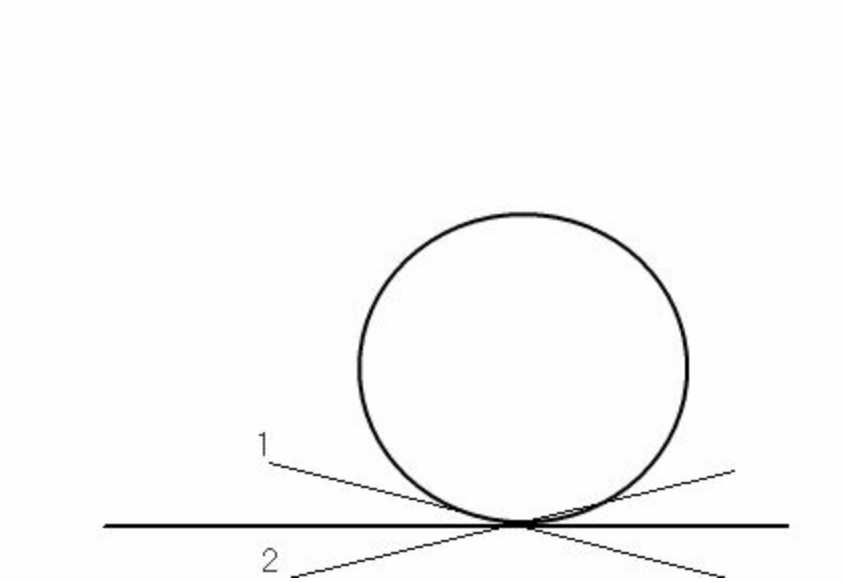
- force (x, y and amplitude) that is exerted on the adjacent bearing



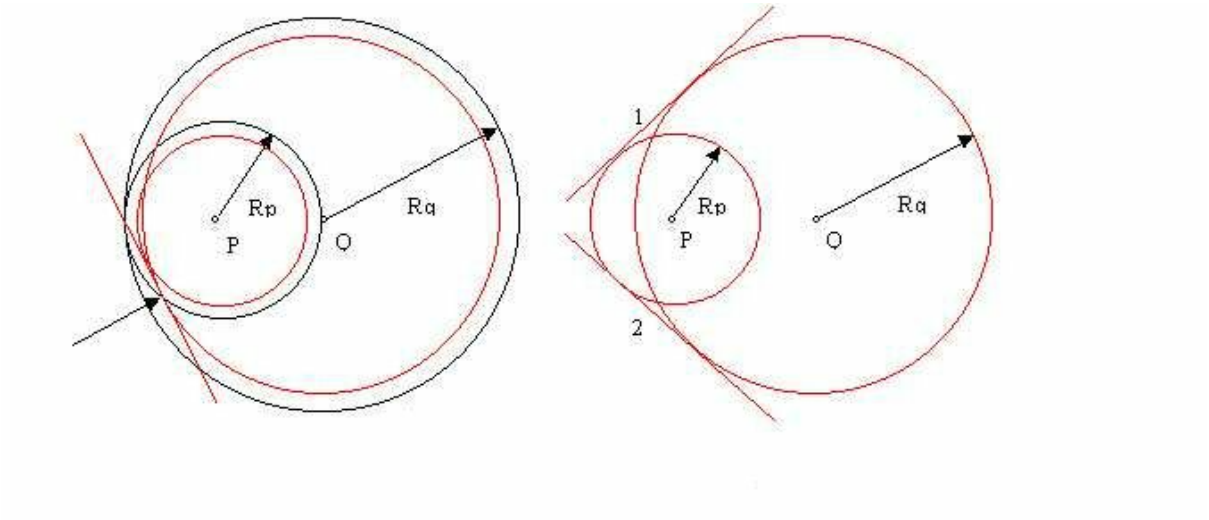
Calculable properties of gear element

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).

© 2024 ARTAS - Engineering Software



Definition of element forces Fn1 and Fn2 in a standard gear transmission



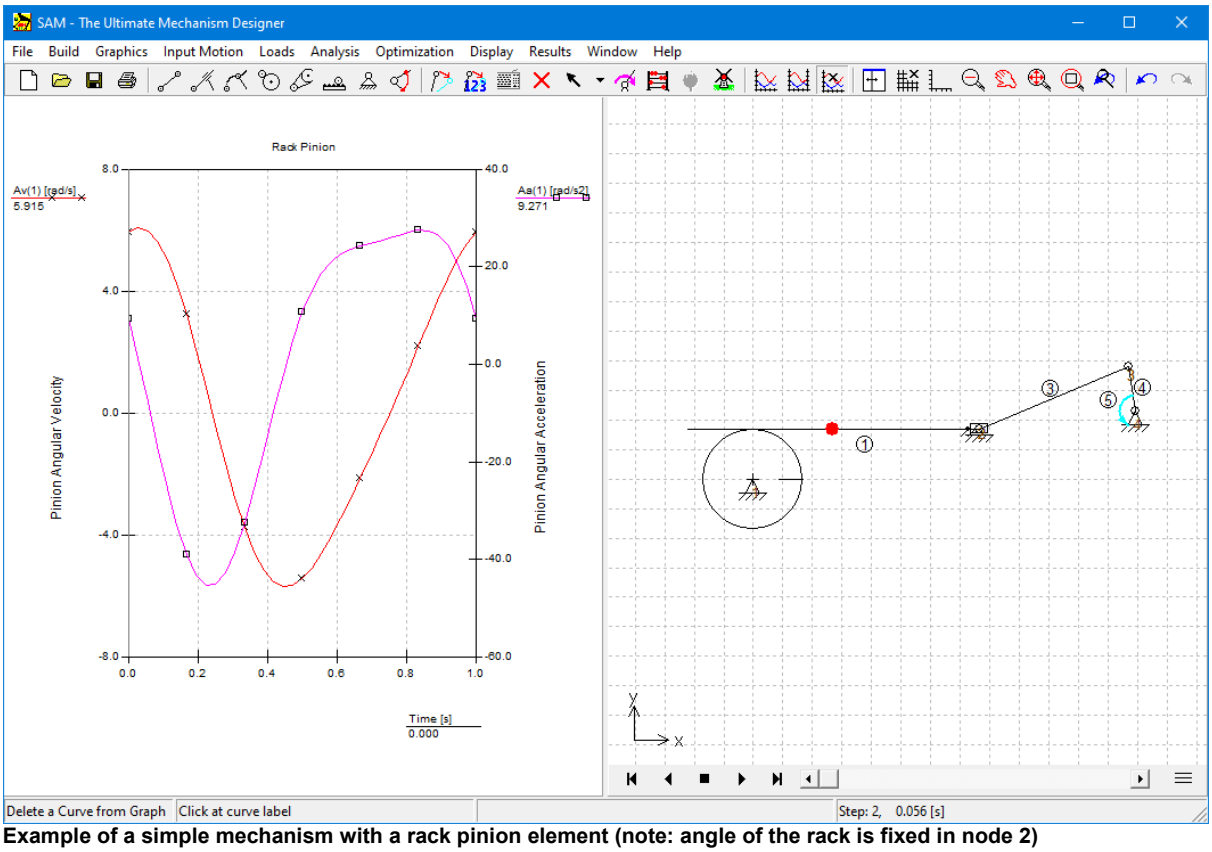
Definition of element forces Fn1 and Fn2 in a gear transmission with internal tooth

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.

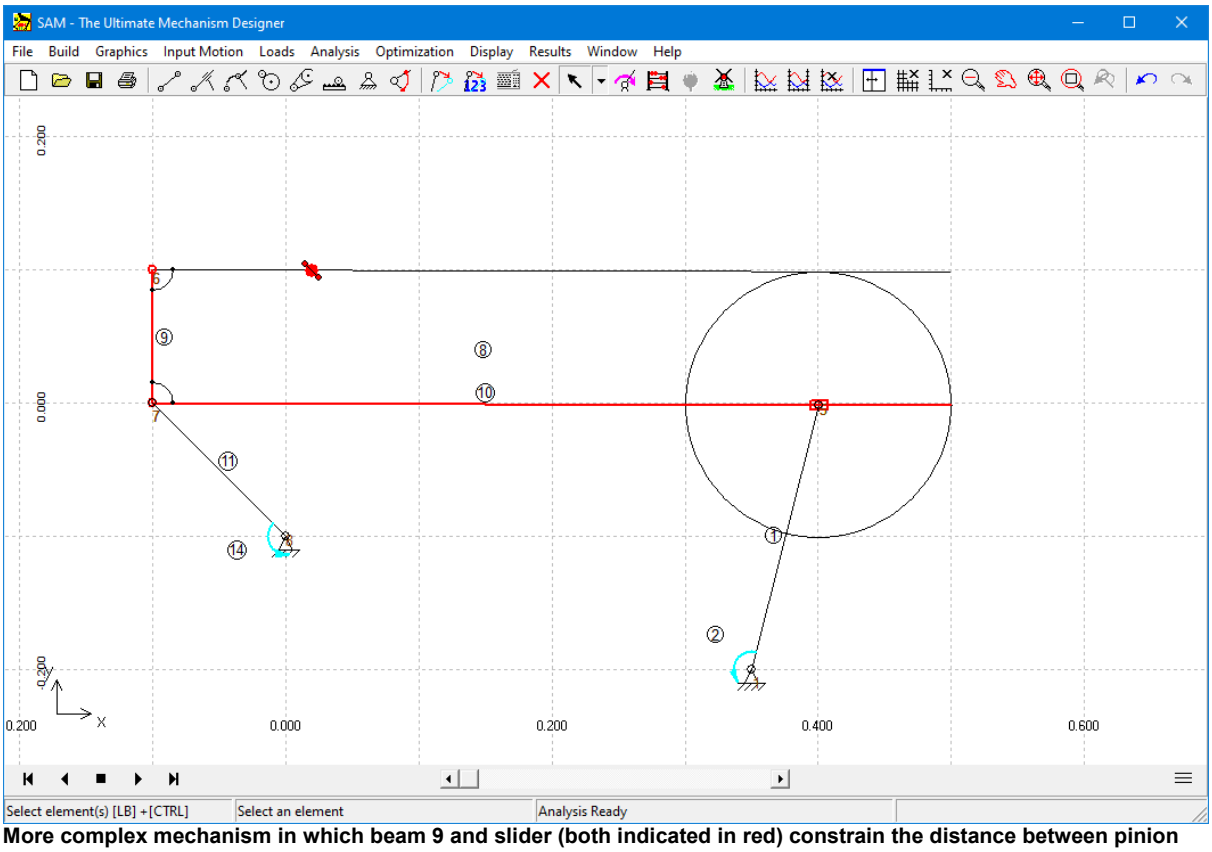
Similar to the gear pair, that only defines the rotational relation between the involved gears but not the distance between the gears, the rack pinion element imposes only one constraint. The distance from rack to the center of the pinion is not fixed and needs to be constrained by external fixation(s)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

© 2024 ARTAS - Engineering Software



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

© 2024 ARTAS - Engineering Software

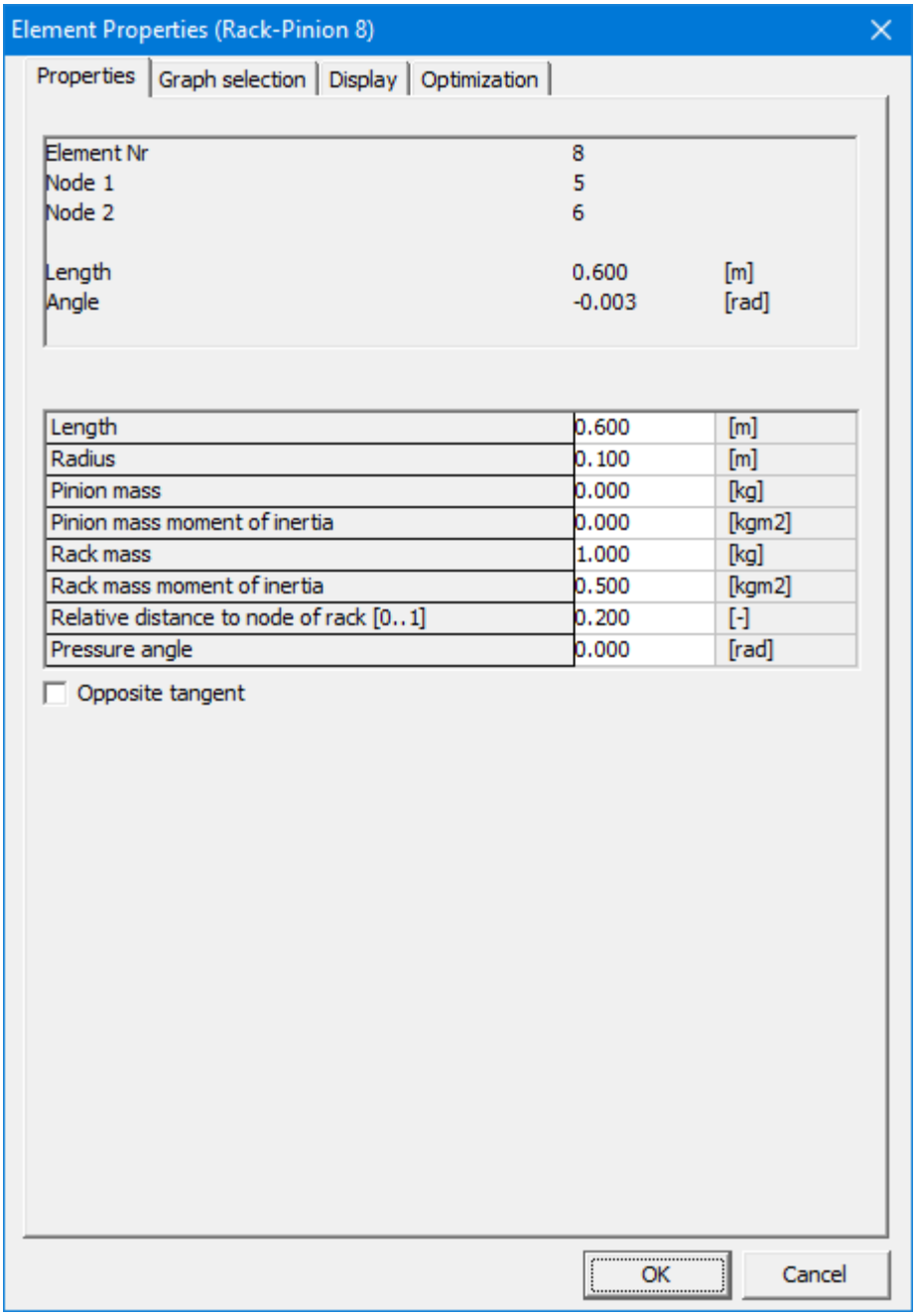


More complex mechanism in which beam 9 and slider (both indicated in red) constrain the distance between pinion and rack.

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

© 2024 ARTAS - Engineering Software



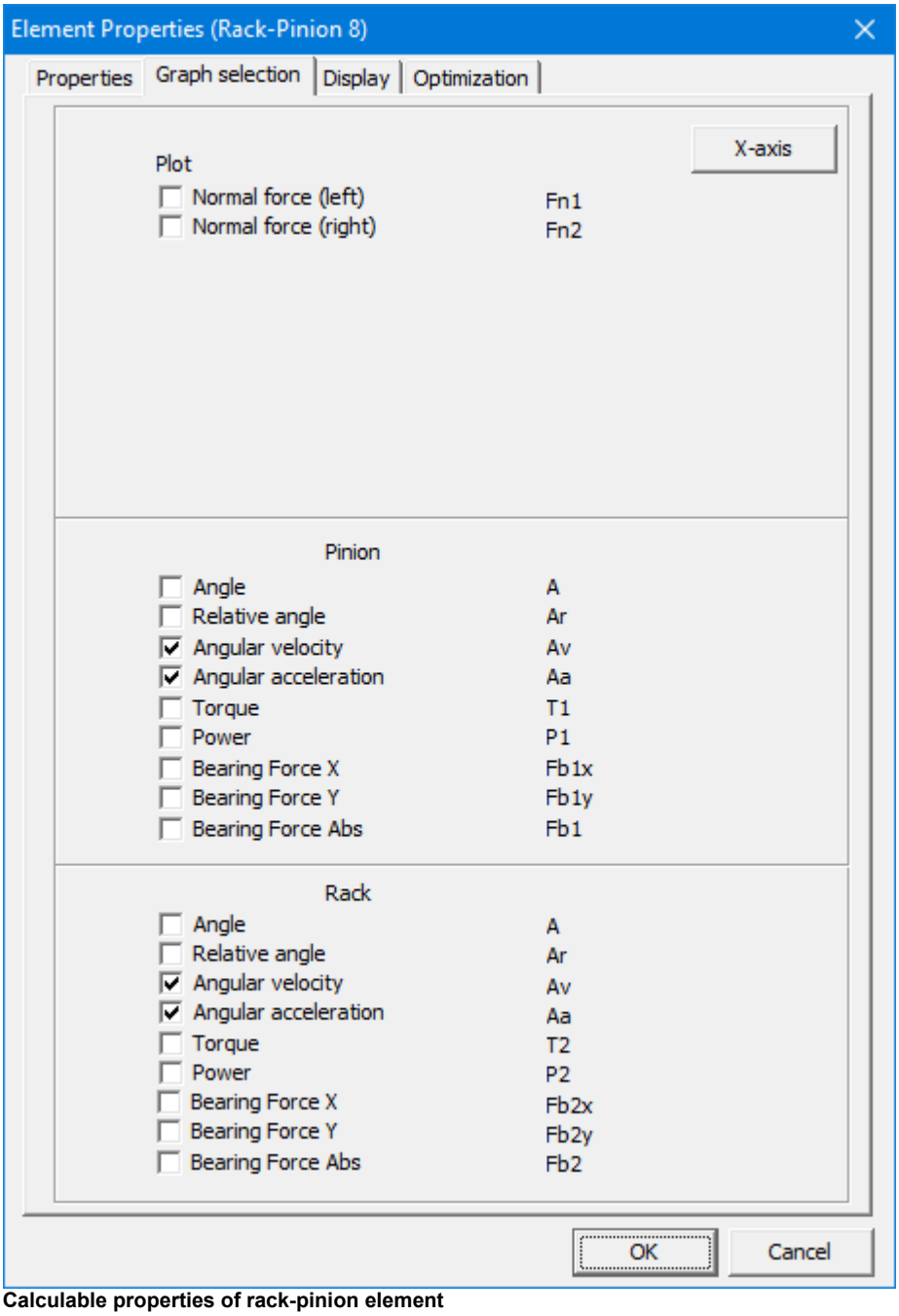
Properties of rack-pinion element

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.

© 2024 ARTAS - Engineering Software

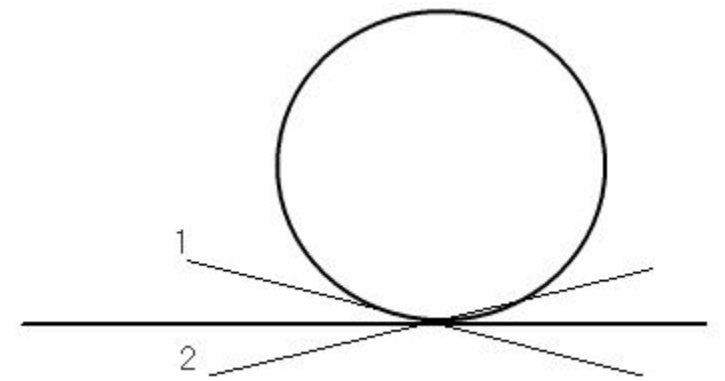
- force (x, y and amplitude) that is exerted on the adjacent bearing



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.

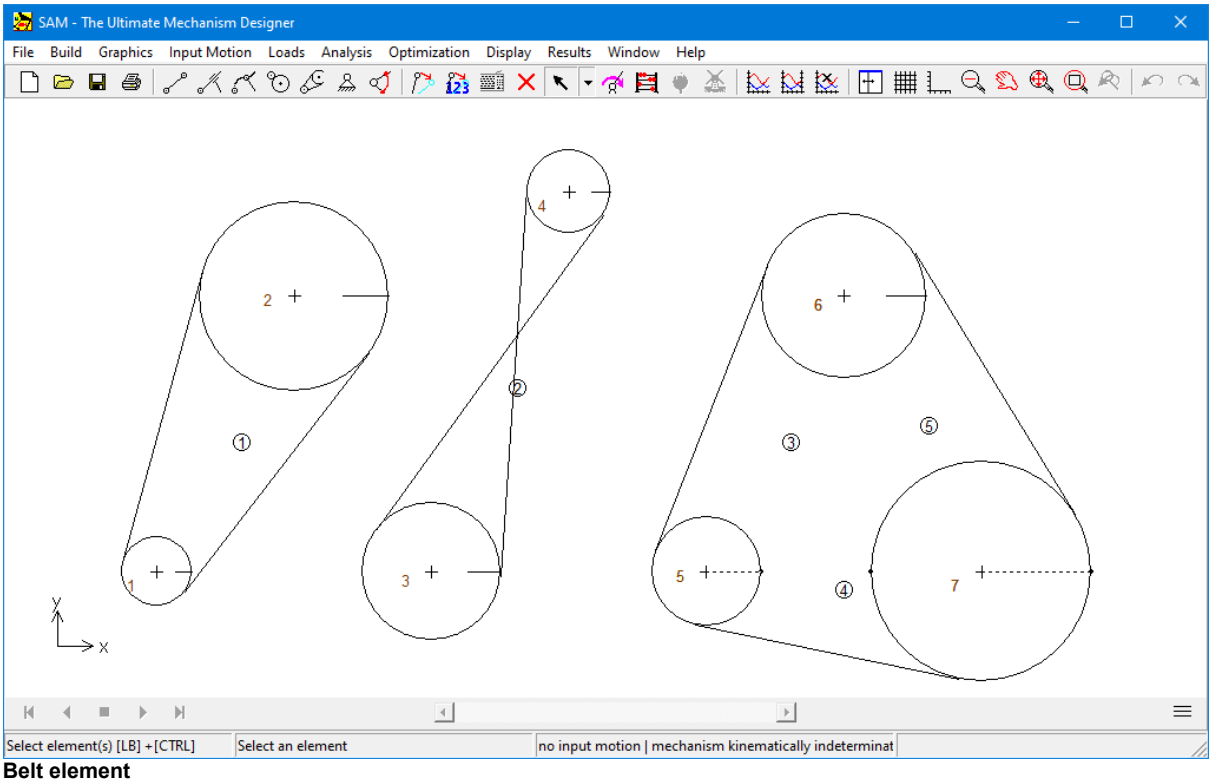
© 2024 ARTAS - Engineering Software



Definition of element forces Fn1 and Fn2 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.

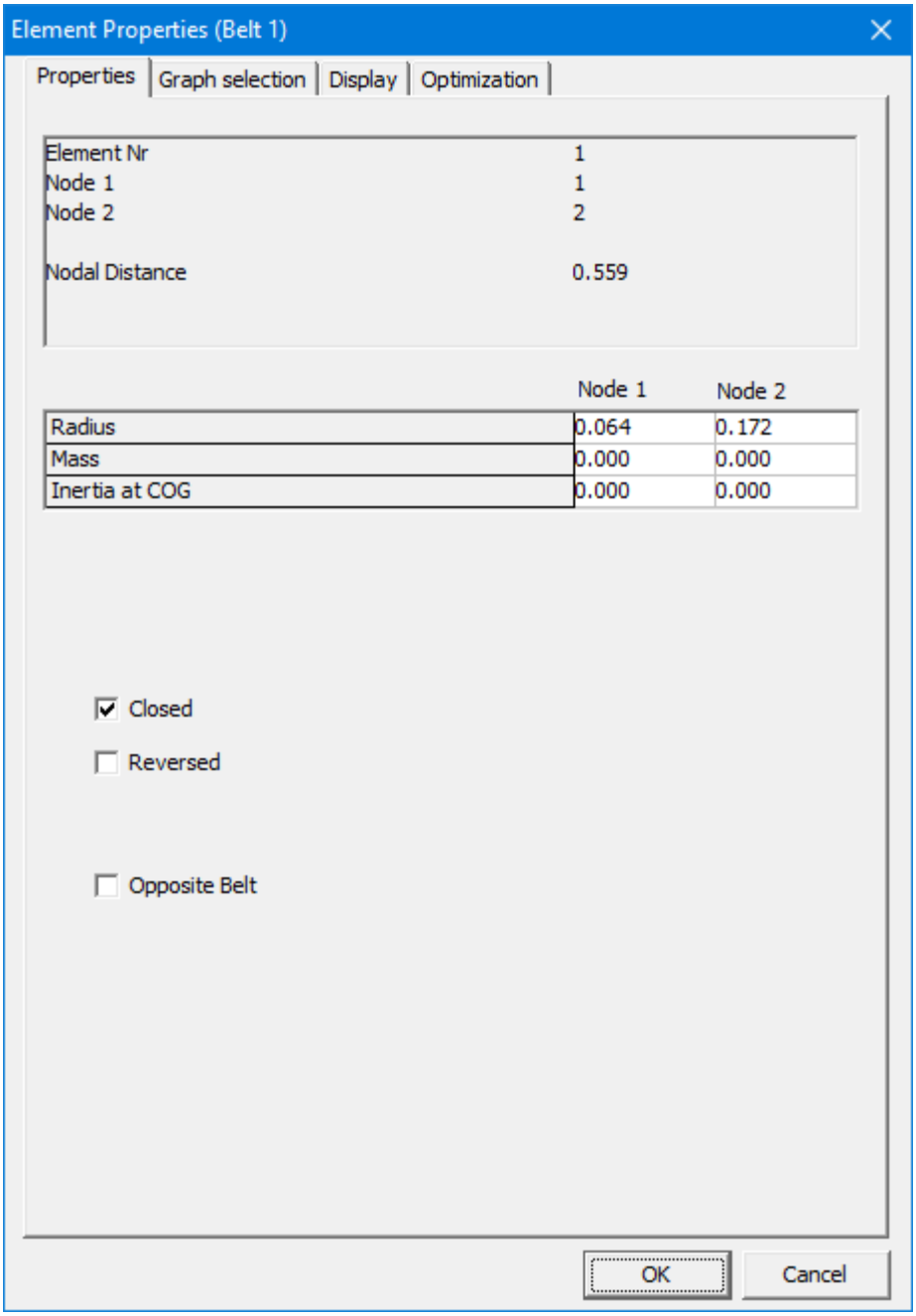


Belt element

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 :

© 2024 ARTAS - Engineering Software

- radius
- mass
- mass moment of inertia



Properties of belt element

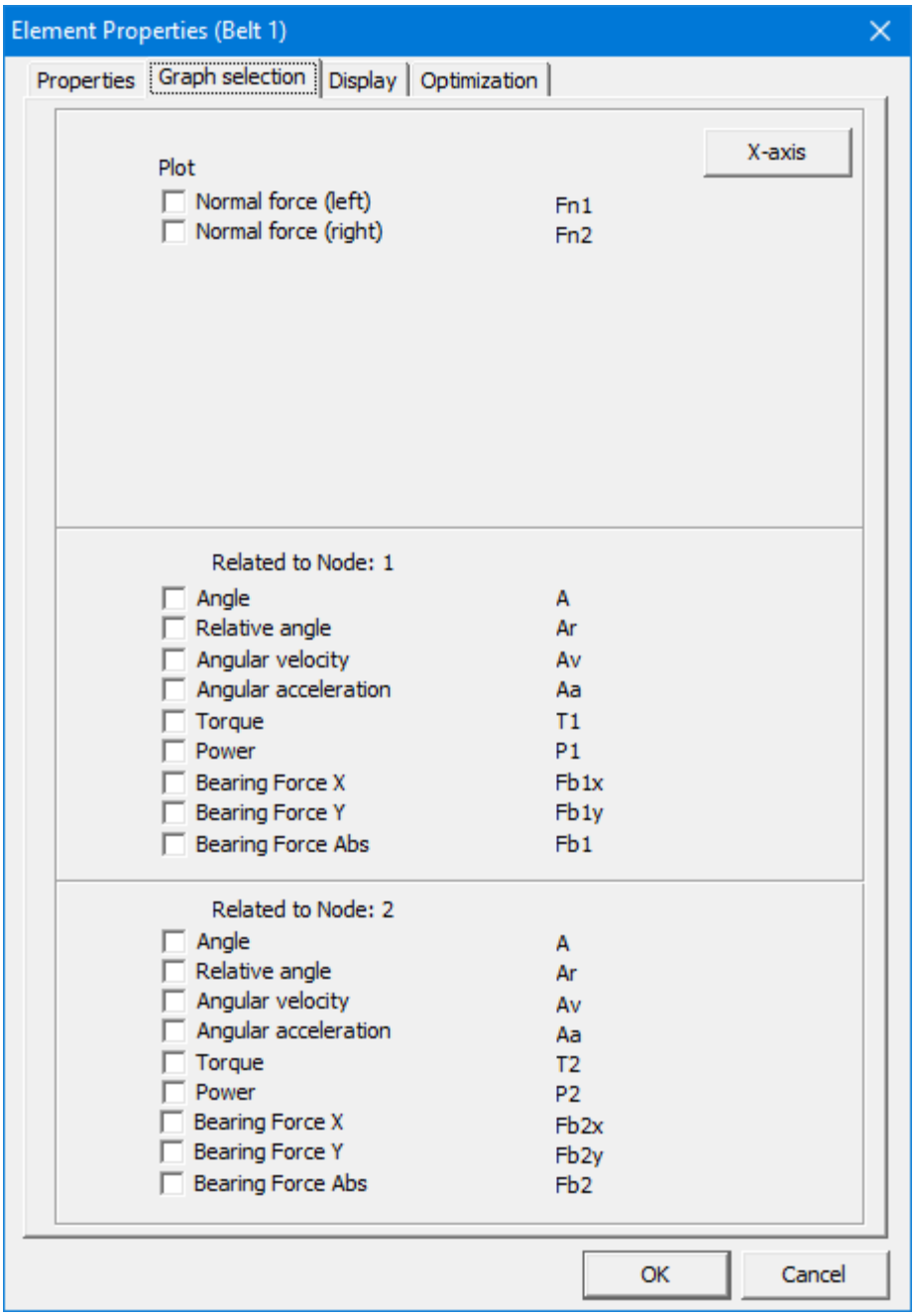
The following items can be selected as result of the analysis process :

- absolute and relative angle, angular velocity, angular acceleration

© 2024 ARTAS - Engineering Software

- 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.

- force (x, y and amplitude) that is exerted on the adjacent bearing



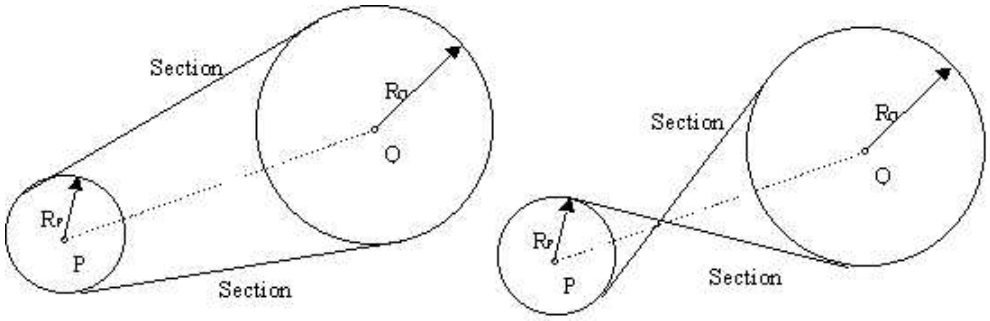
Calculable properties of belt element

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

© 2024 ARTAS - Engineering Software

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 a able to transmit both positive and negative forces).



Definition of element forces Fn1 and Fn2 in a belt transmission

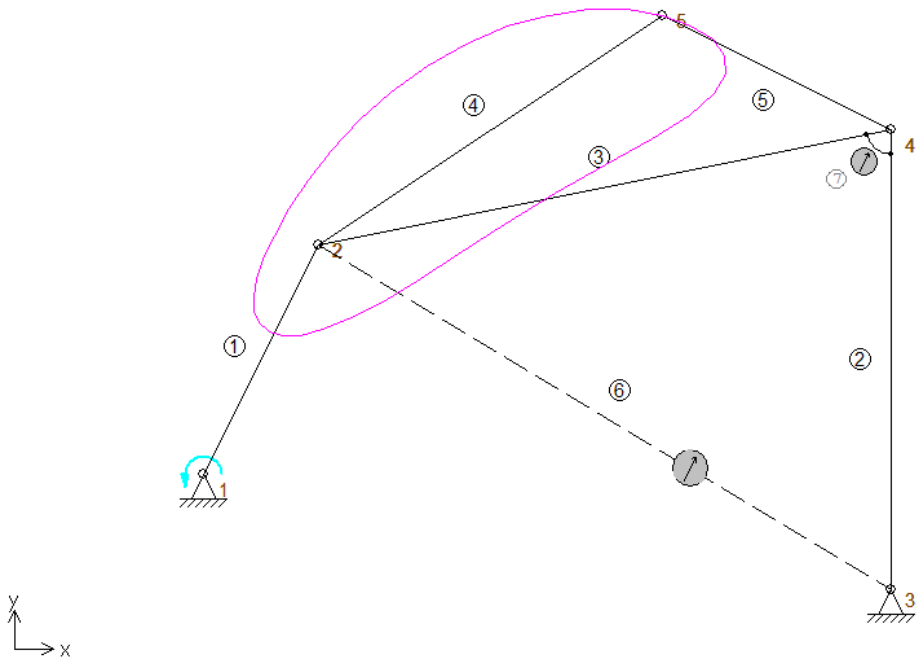
11.7 Sensor

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.

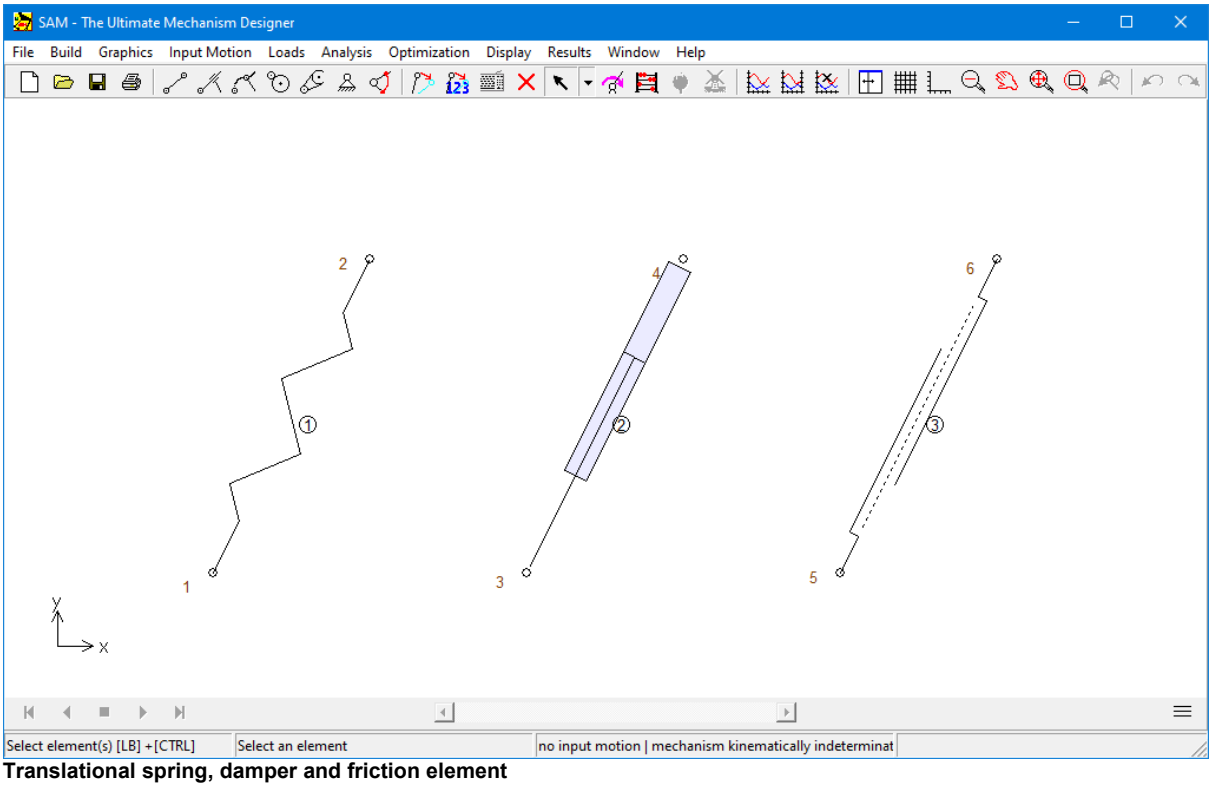
The result of the sensor depends on the order in which the involved elements/nodes are picked. The positive direction of the sensor is always from the first element/node to the second element/node. The rotational sensor measures the angle between two elements at a certain node while considering both 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.

© 2024 ARTAS - Engineering Software



11.8 Spring, Damper, Friction



Translational spring, damper and friction element

© 2024 ARTAS - Engineering Software

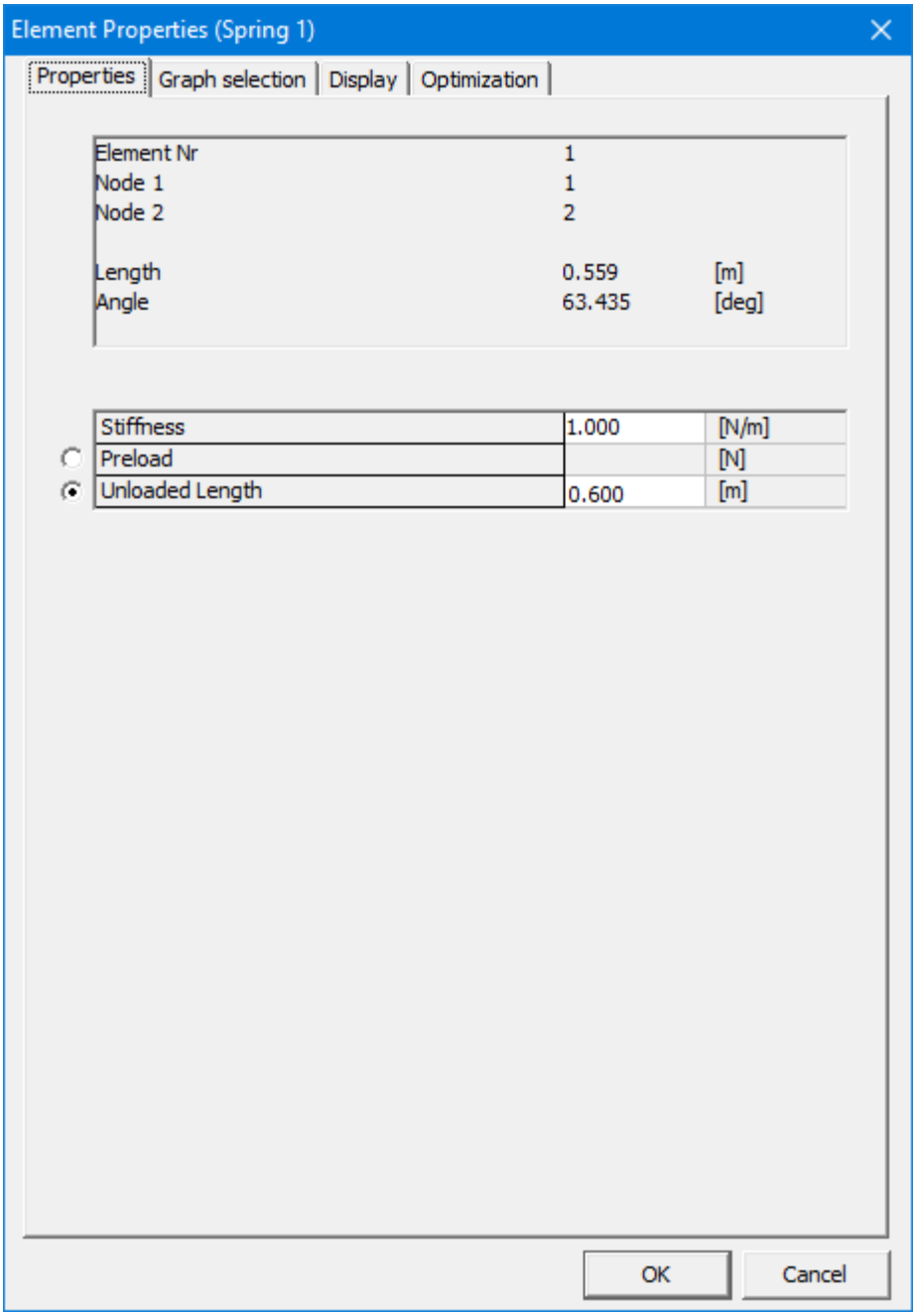
The spring, damper and friction element are closely related. Each of these elements has no influence on the motion of the mechanism and is only relevant in the kinetostatics analysis to model forces that result from springs, damper or friction. The shape of each element is defined by the position of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

© 2024 ARTAS - Engineering Software

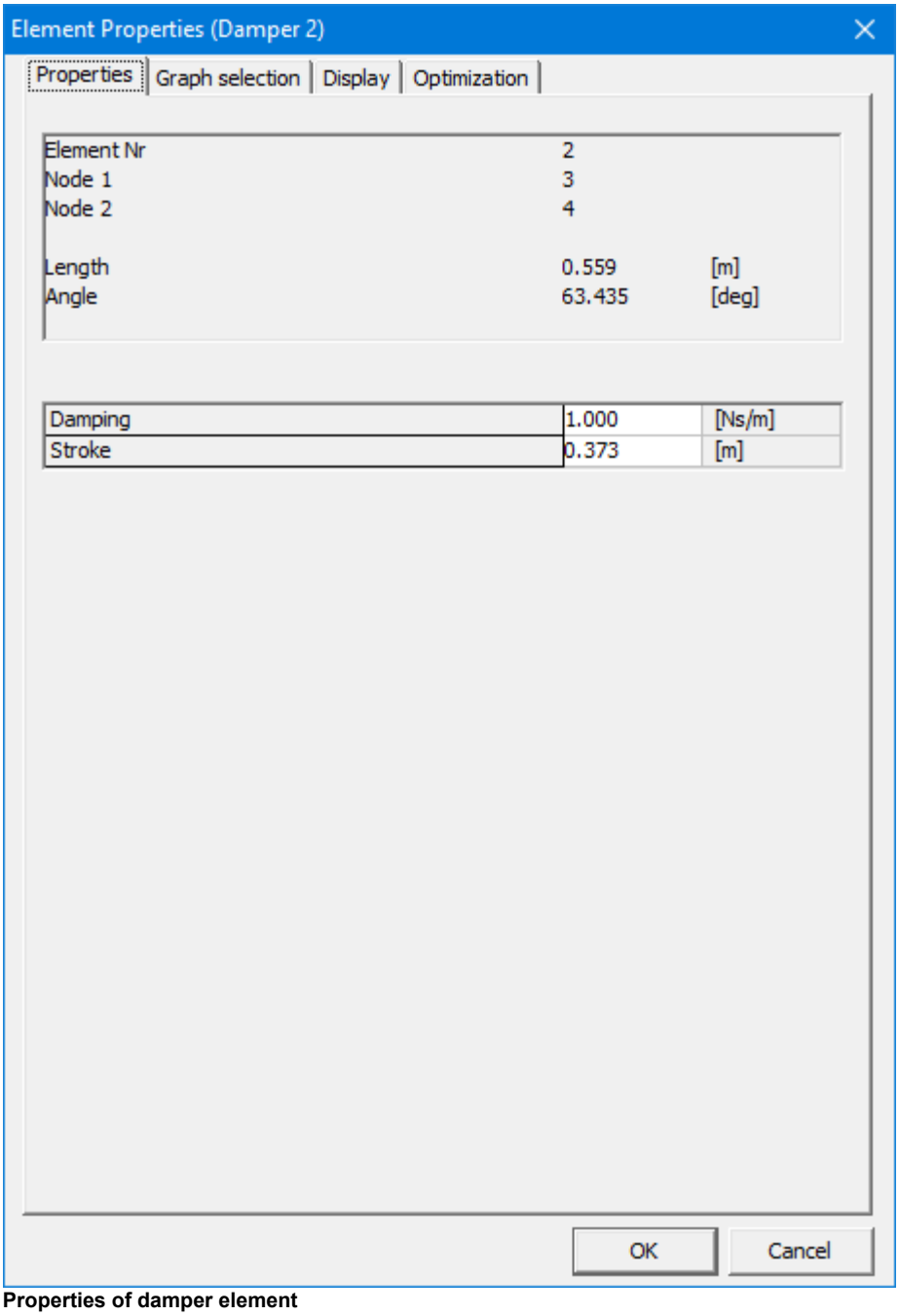


Properties of spring element

The damper element possesses two input properties :

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

© 2024 ARTAS - Engineering Software

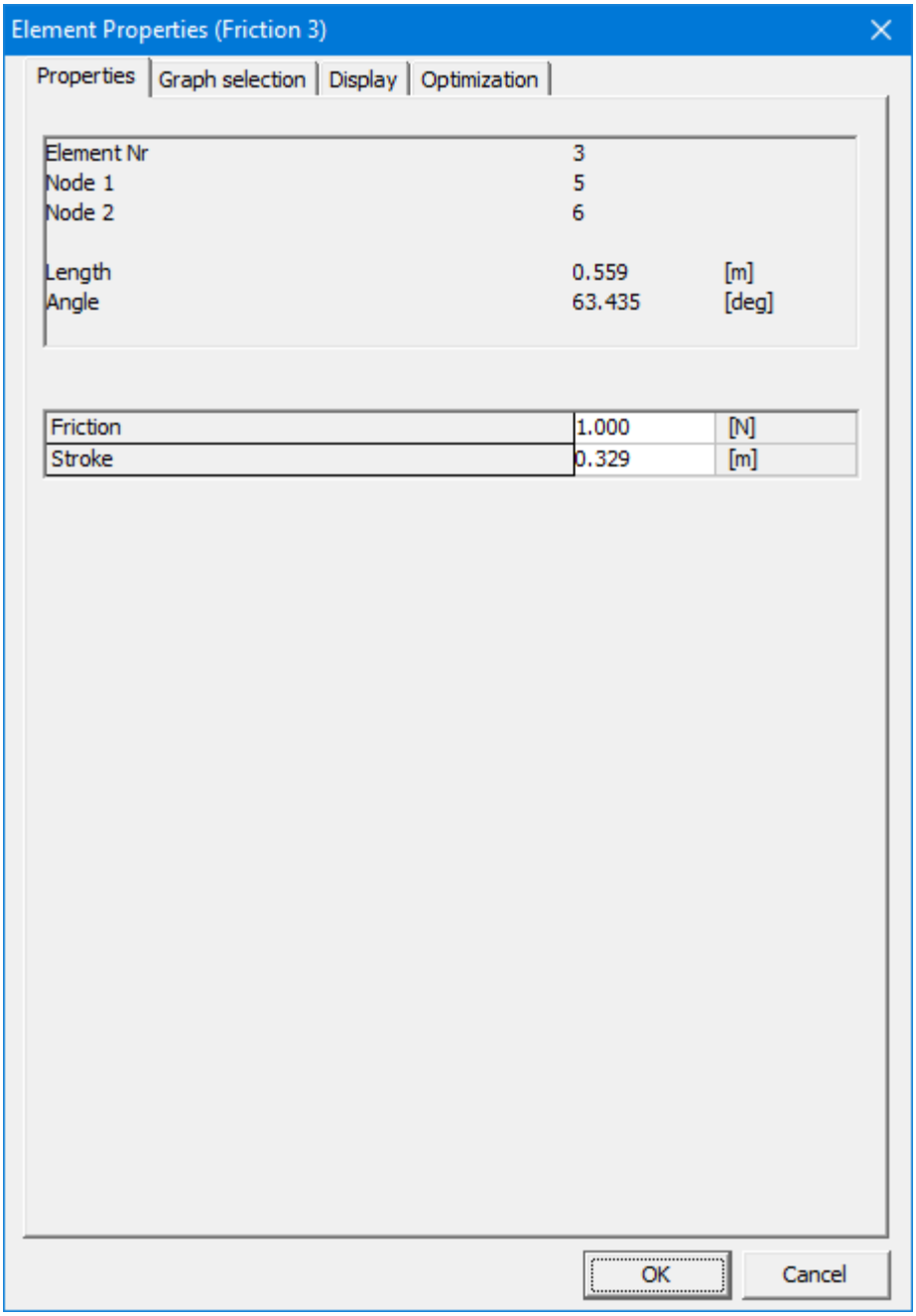


Properties of damper element

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

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

© 2024 ARTAS - Engineering Software

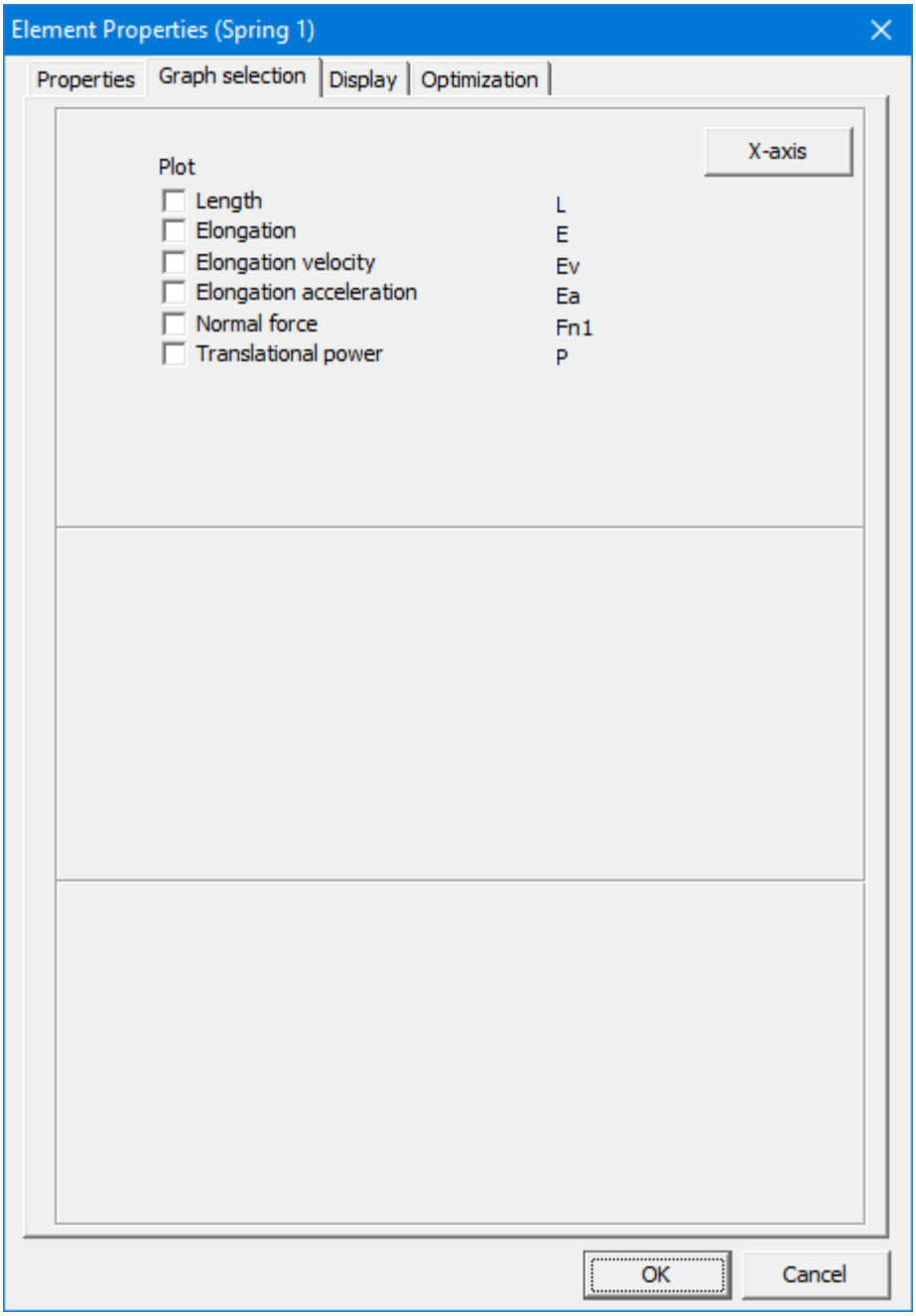


Properties of friction element

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

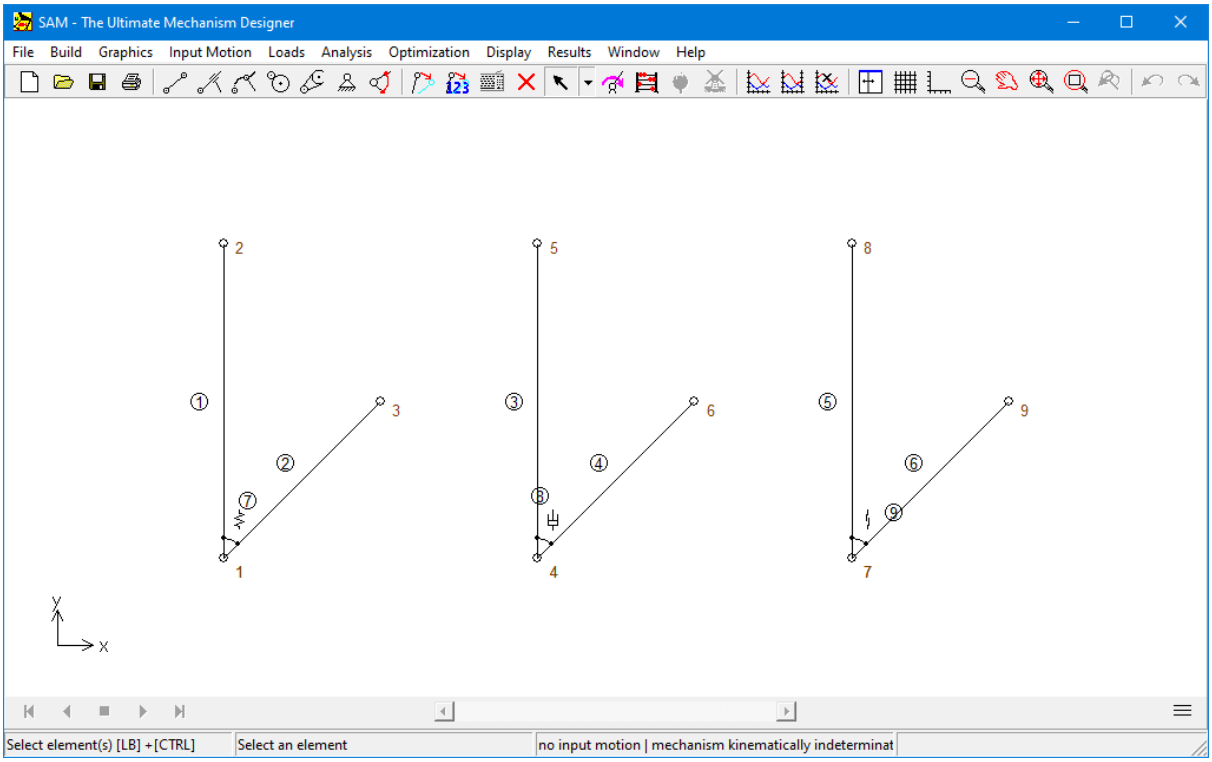
© 2024 ARTAS - Engineering Software



Calculable properties of spring, damper and friction element are identical

© 2024 ARTAS - Engineering Software

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.

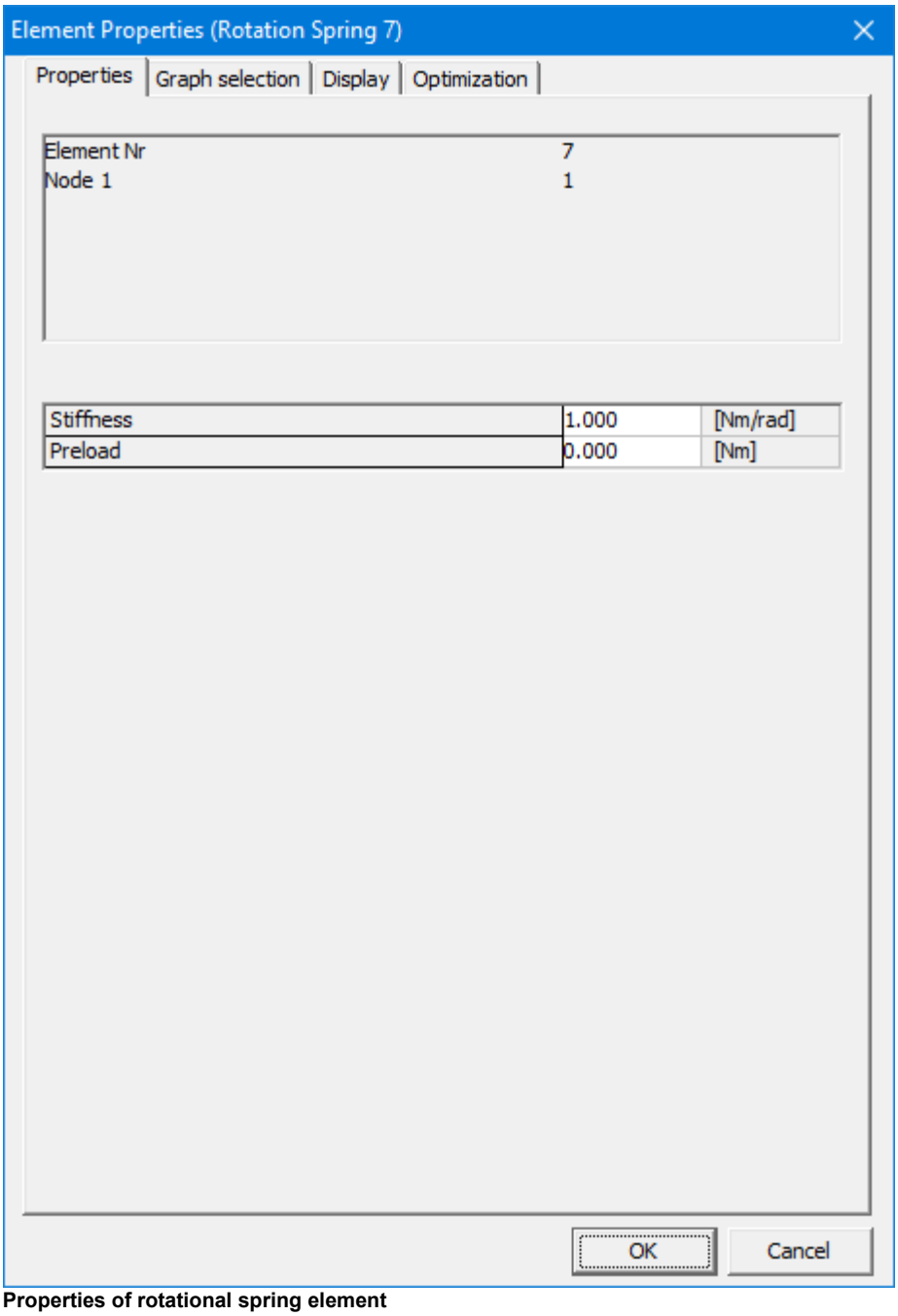
The properties of each of these elements, along with the element and node identification can be viewed and edited in the property dialog. Since each of them has different mechanical properties, there are three different property dialogs.

The rotational spring element has two mechanical properties :

- rotational stiffness

© 2024 ARTAS - Engineering Software

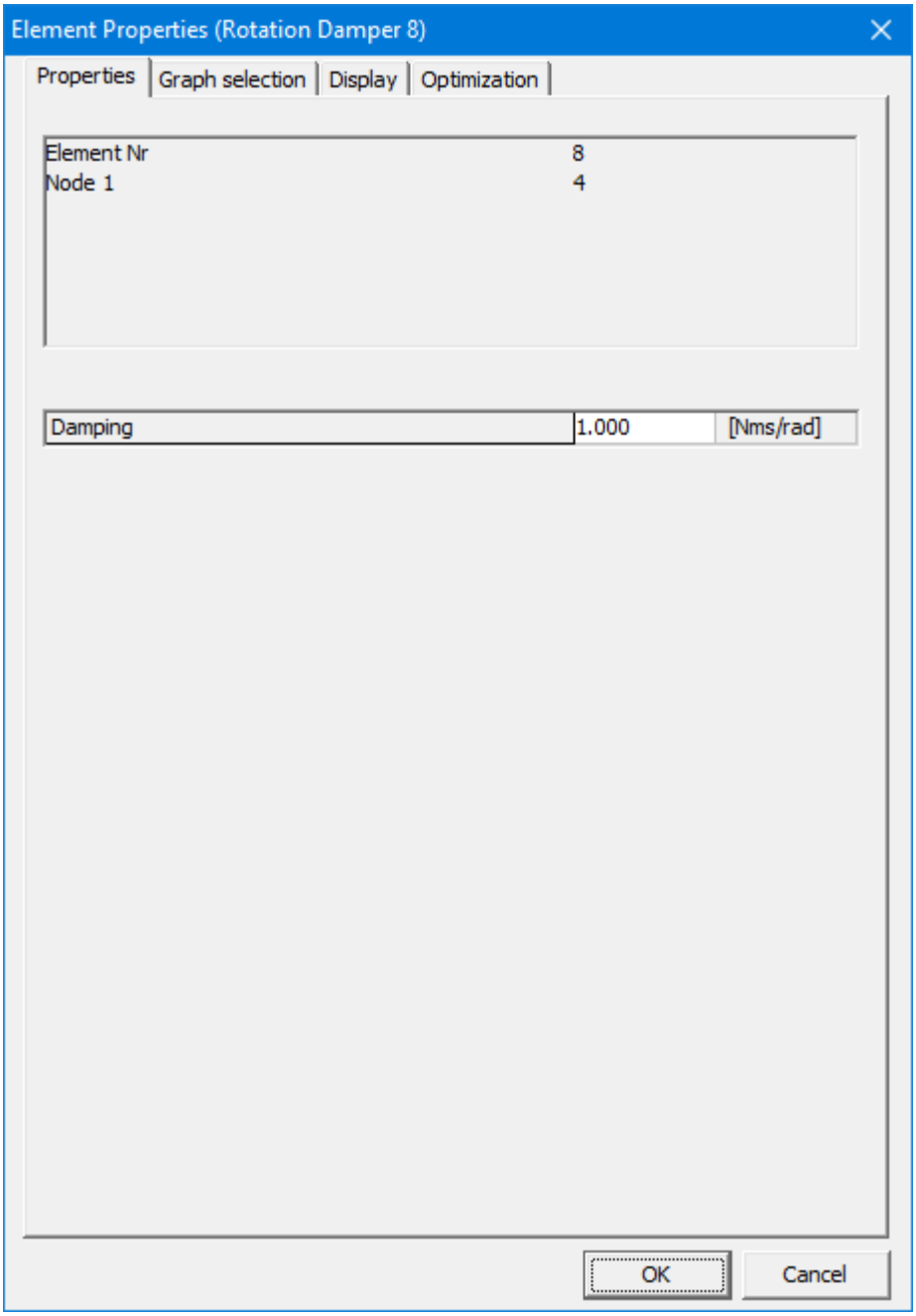
- preload in the initial position of the mechanism.



Properties of rotational spring element

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

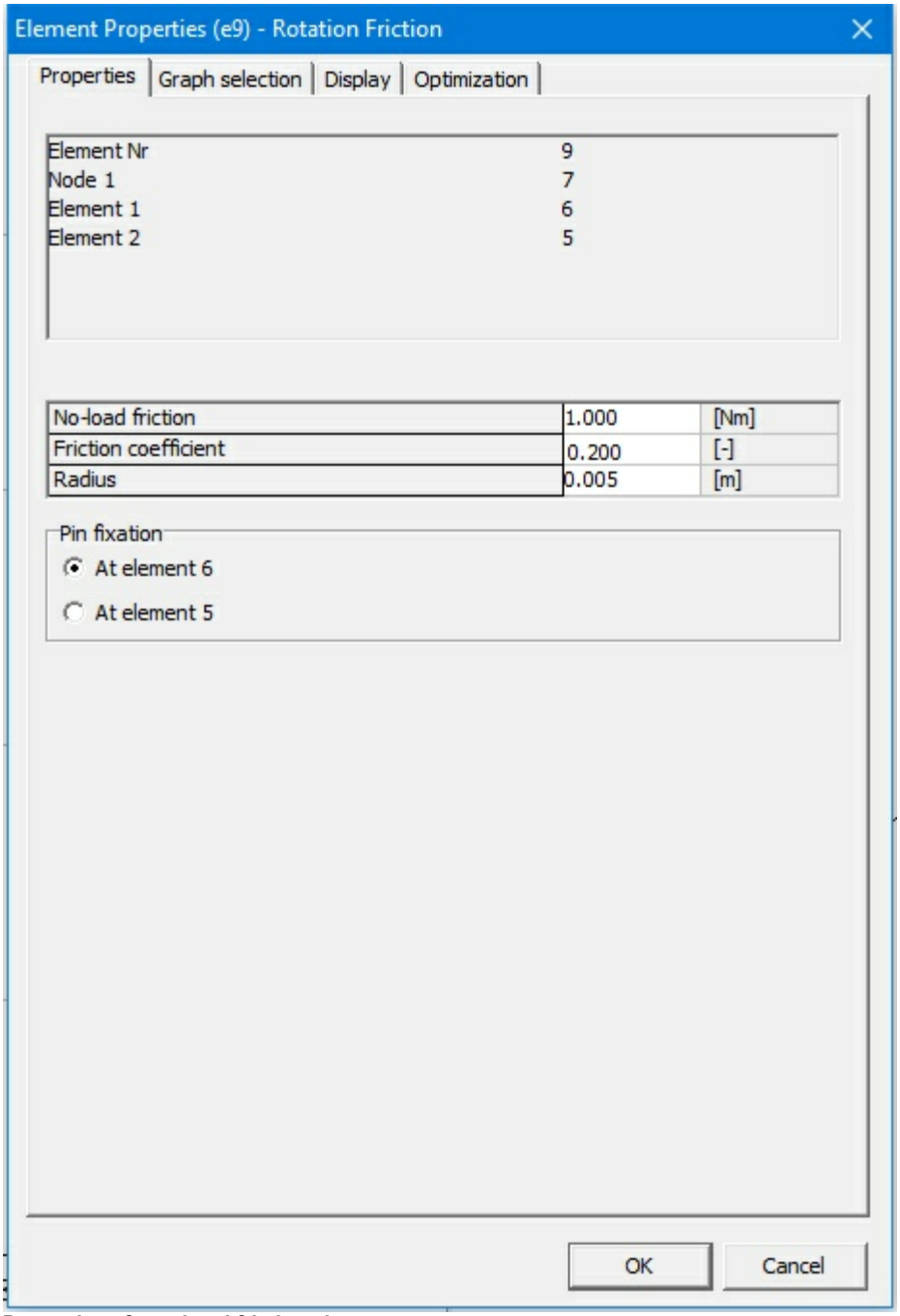
© 2024 ARTAS - Engineering Software



Properties of rotational damper element

The rotation friction element is used to model coulomb friction, which is defined by the value of a constant friction torque (no-load friction) and a load dependent component which depends on the bearing force in combination with the friction coefficient and the radius. The friction torque = no-load friction + bearing force * friction coefficient * radius. In order to define all situations unambiguously, the user had to indicate which element holds the pin of the connection.

© 2024 ARTAS - Engineering Software

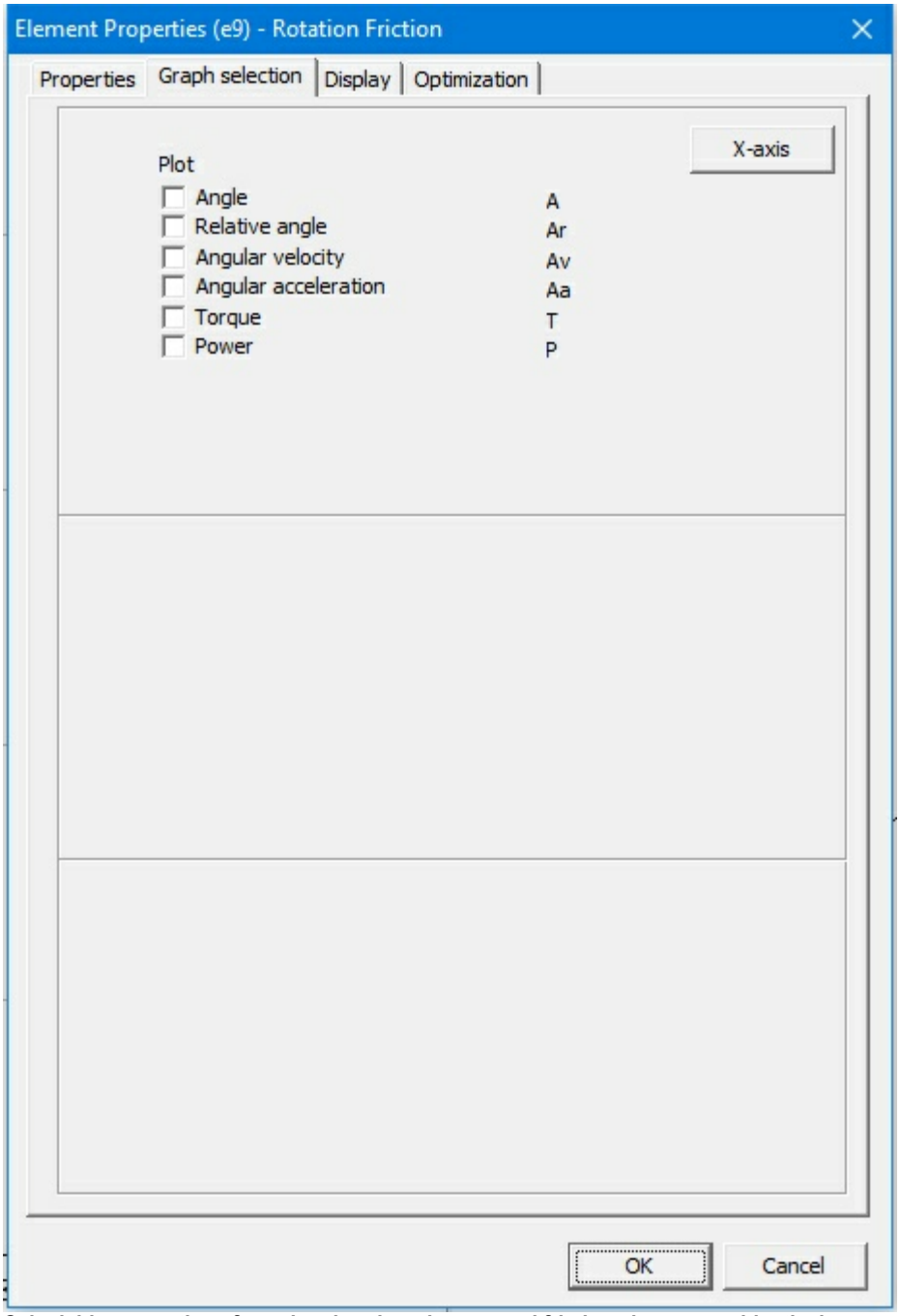


Properties of rotational friction element

The output choices of each of these elements is identical :

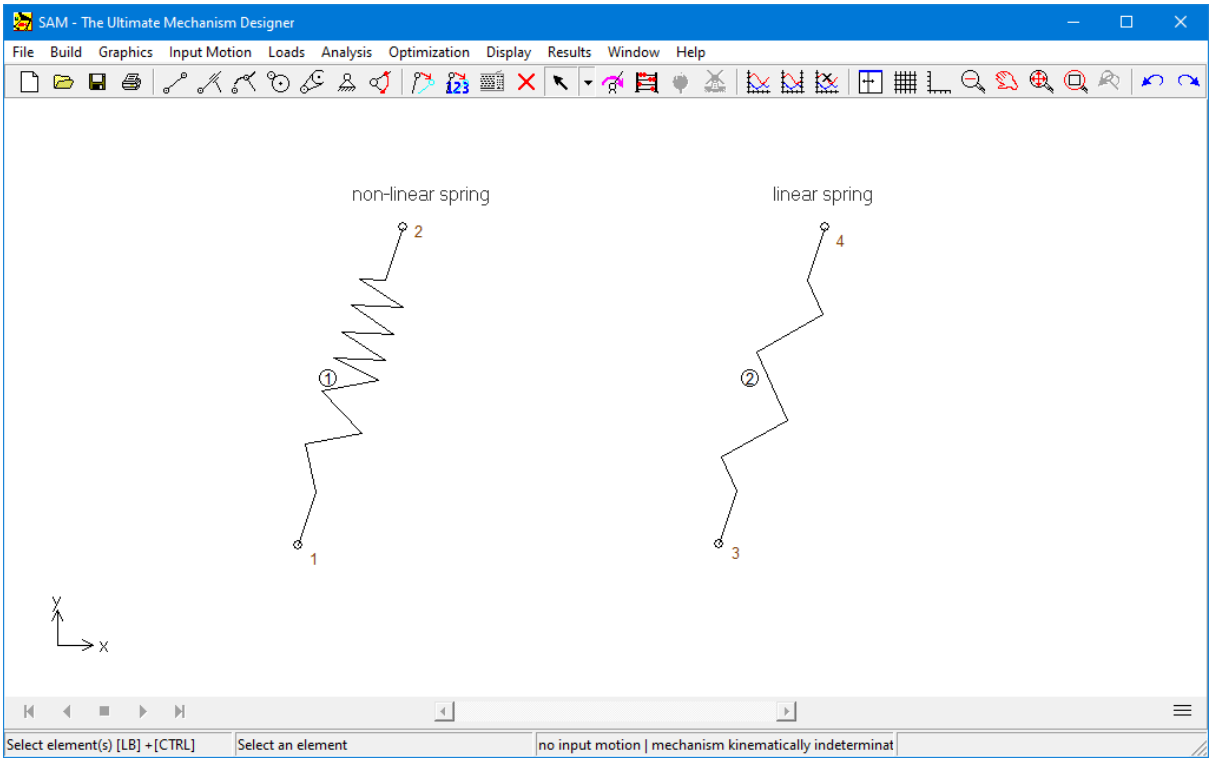
- relative angle and its derivatives
- torque and corresponding power

© 2024 ARTAS - Engineering Software



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

11.10 Non-linear spring



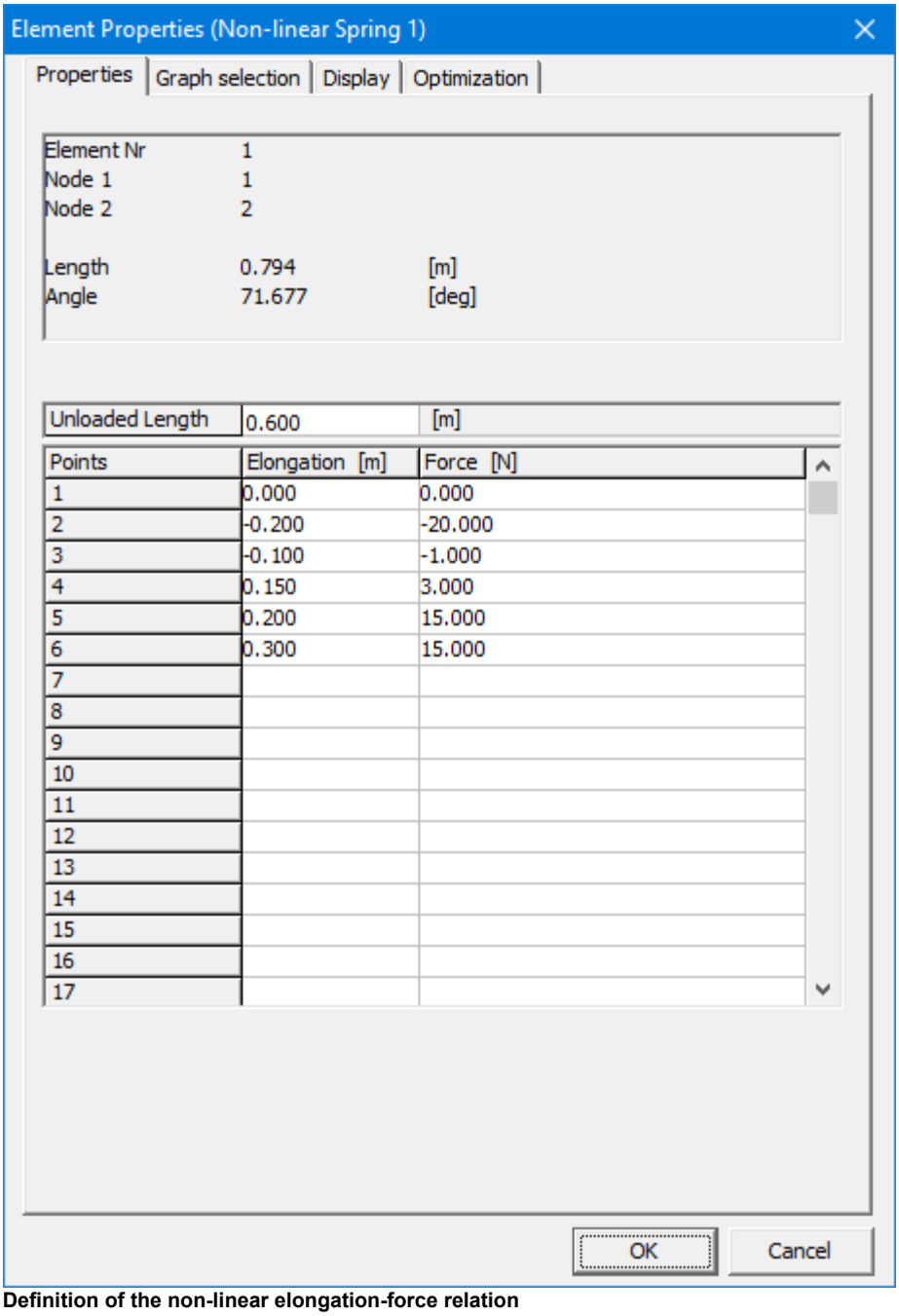
Non-linear spring (and a standard linear spring)

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.

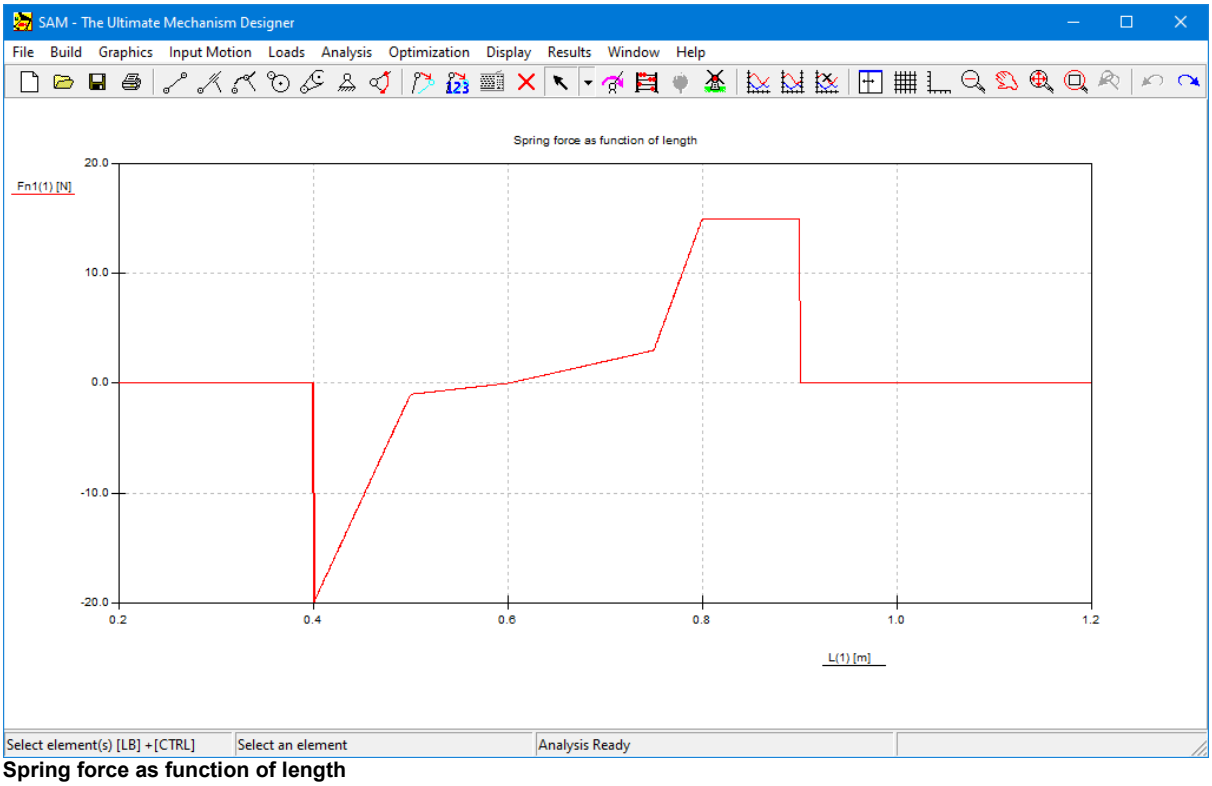
Please note that (in line with all other elements) the elongation result of the non-linear spring is calculated relative to the initial position of the mechanism and NOT relative to the unloaded length of the spring. This difference is important in case of a pre-load of the spring in the initial position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

© 2024 ARTAS - Engineering Software



Definition of the non-linear elongation-force relation

© 2024 ARTAS - Engineering Software

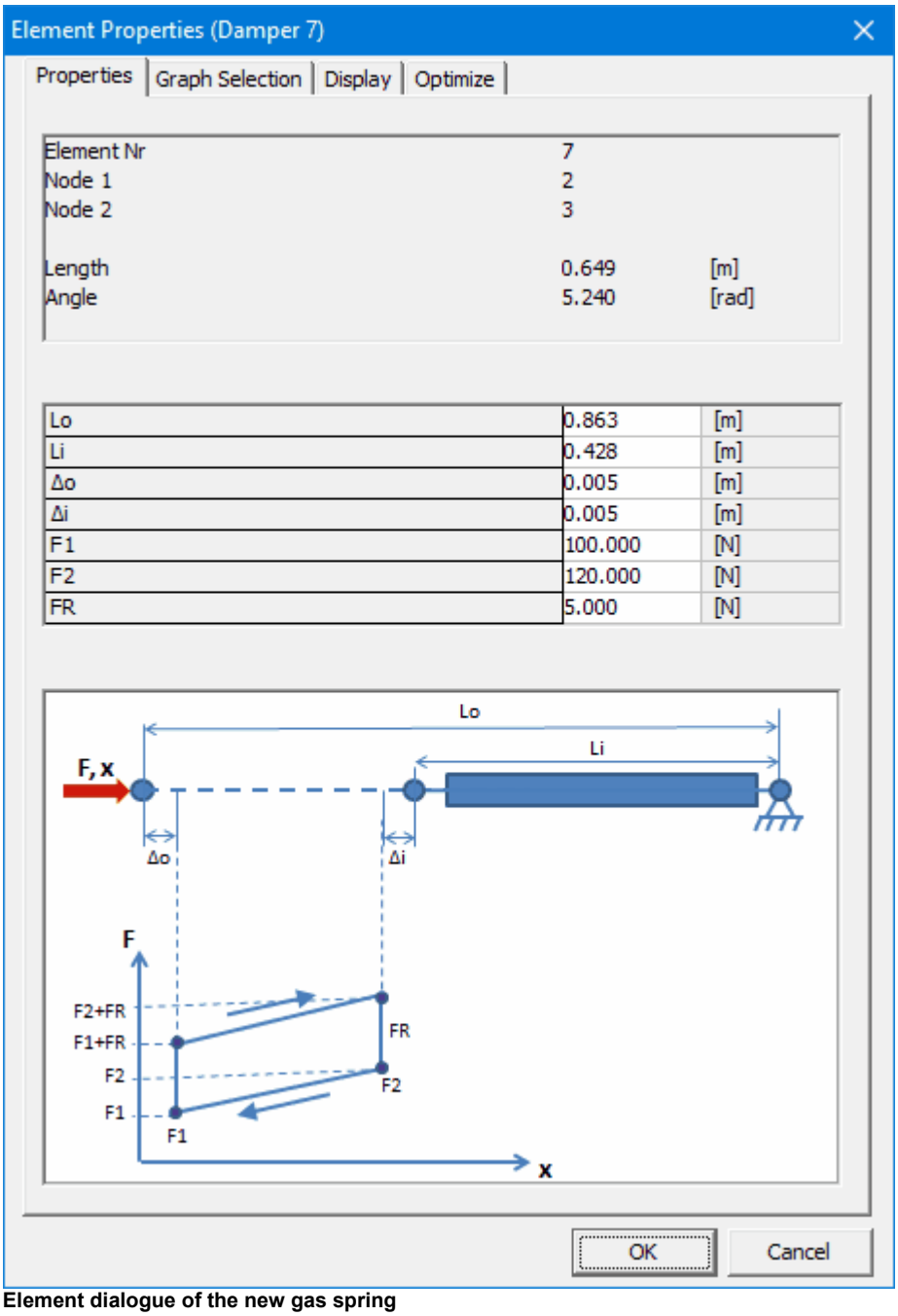


Spring force as function of length

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.

© 2024 ARTAS - Engineering Software



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

© 2024 ARTAS - Engineering Software

- 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.

© 2024 ARTAS - Engineering Software

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).

Middle Mouse Button (press)
When the middle mouse button is pressed the a mouse movement leads to a pan action in the mechanism window.

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 (Reft 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, carthesian/polar or you can even "construct" intersections.

<DEL>
Pressing the <DEL> 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
If a shape is selected it will be copied to the clipboard and can be pasted (Ctrl+V). If no shape is selected the graph window will be copied to the clipboard.

© 2024 ARTAS - Engineering Software

<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>+Q
The mechanism window will be copied to the clipboard.

<Ctrl>+mouse move
Horizontal of 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

<F11>
Shortcut for User Defined Results ...

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.

© 2024 ARTAS - Engineering Software

 **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** Select any object, element, shape, ...

 **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.

© 2024 ARTAS - Engineering Software

 **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 (alternatively one can press the middle mouse button and move the mouse)

 **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

© 2024 ARTAS - Engineering Software

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, Rack and Pinion
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 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

© 2024 ARTAS - Engineering Software

- 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 create and manipulate shapes that are 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

© 2024 ARTAS - Engineering Software

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).

© 2024 ARTAS - Engineering Software

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 :

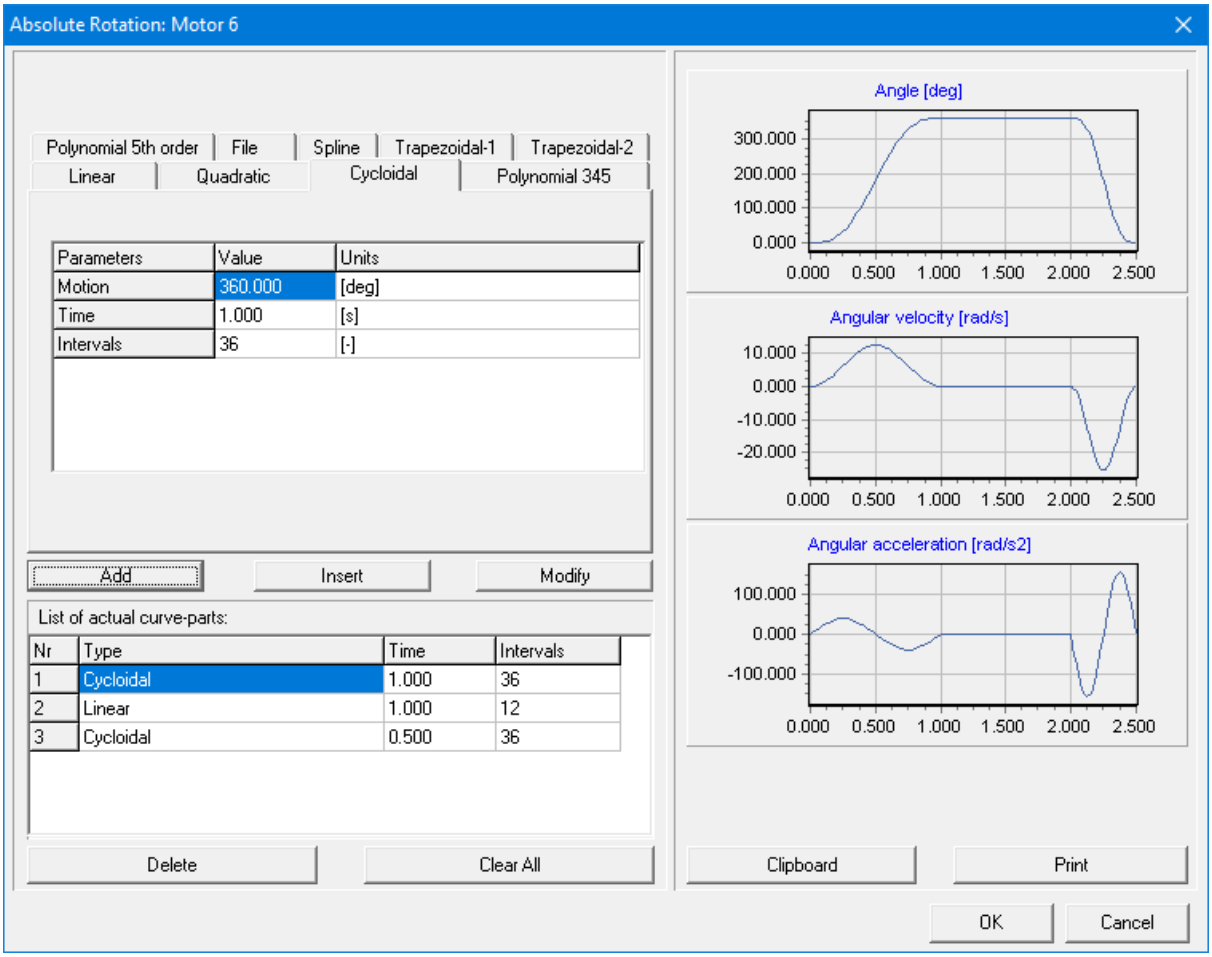
- Linear: motion with constant velocity
- Sinusoidal: sinusoidal motion

© 2024 ARTAS - Engineering Software

- 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.

© 2024 ARTAS - Engineering Software

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 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:

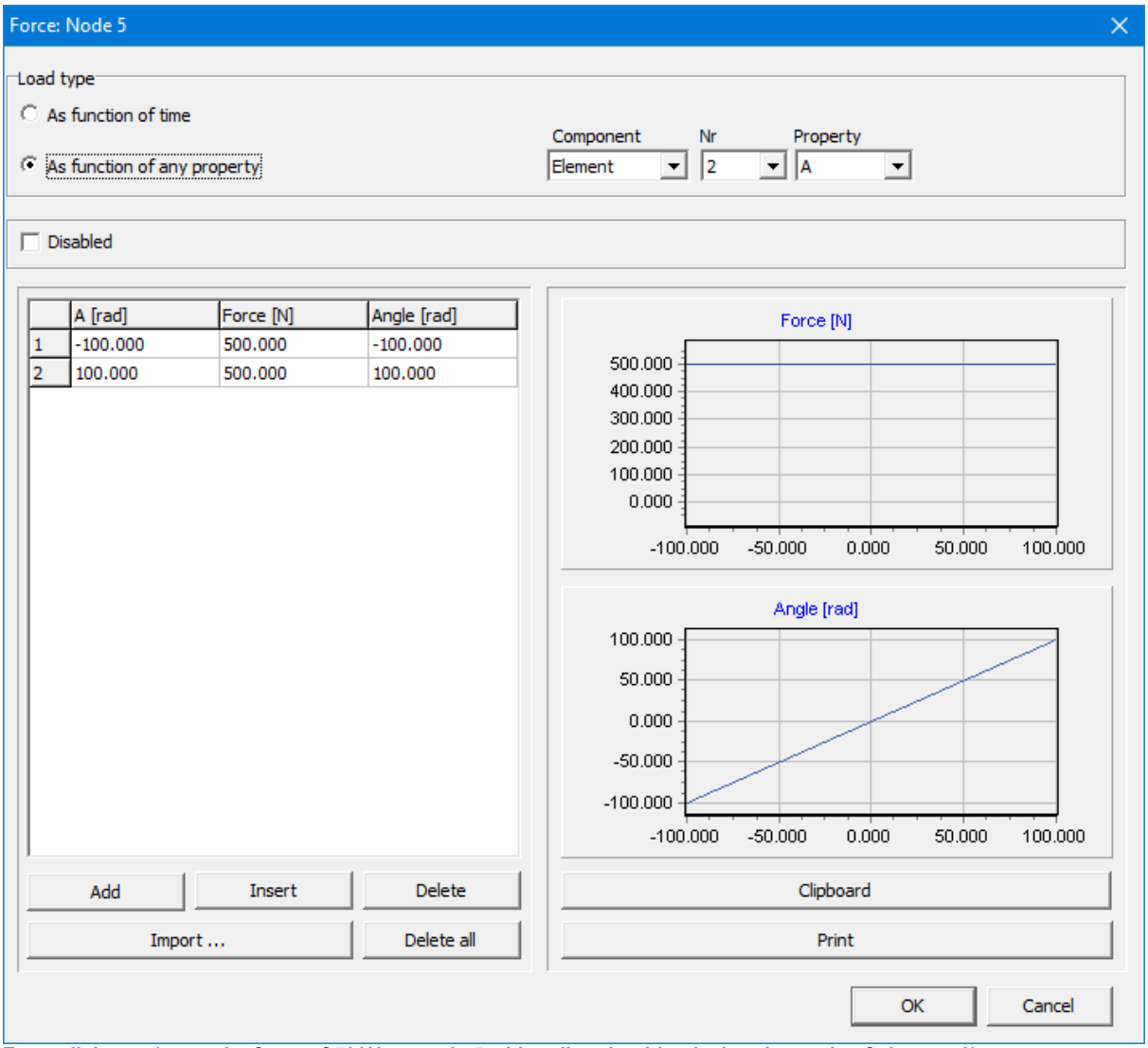
Force/Torque Dialog Box

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.

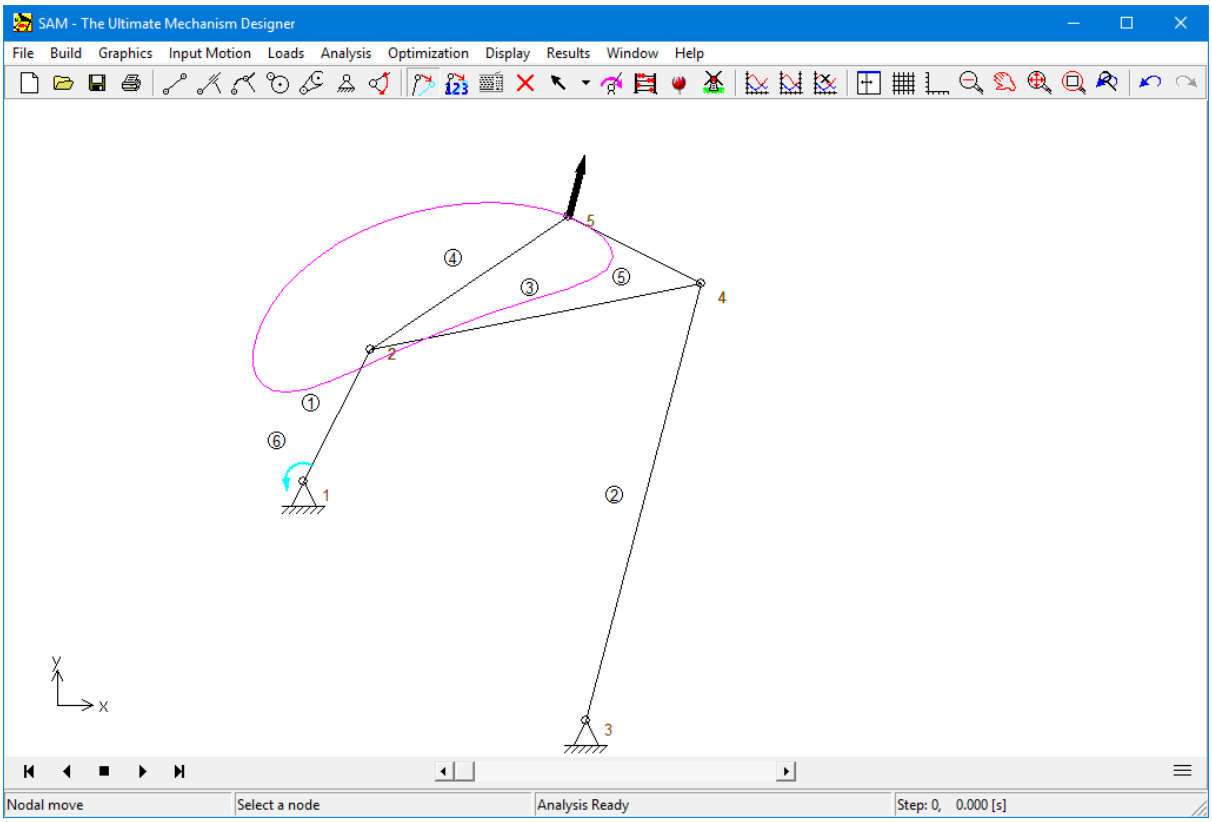
In the dialogue below a force on node 5 is defined (see title of dialogue). It has a constant value of 500N and its angle is always identical to the angle/direction of element 2.

© 2024 ARTAS - Engineering Software



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

© 2024 ARTAS - Engineering Software



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.

© 2024 ARTAS - Engineering Software

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.

Mechanism > Clipboard Ctrl+Q
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.

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.

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.

Copy > Clipboard Ctrl+C

© 2024 ARTAS - Engineering Software

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.

© 2024 ARTAS - Engineering Software

Chapter 13

13 Literature

[1] 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

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

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

[4] 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

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

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

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

[8] 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.

[9] 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.

© 2024 ARTAS - Engineering Software

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.

© 2024 ARTAS - Engineering Software

- 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 seperate 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.

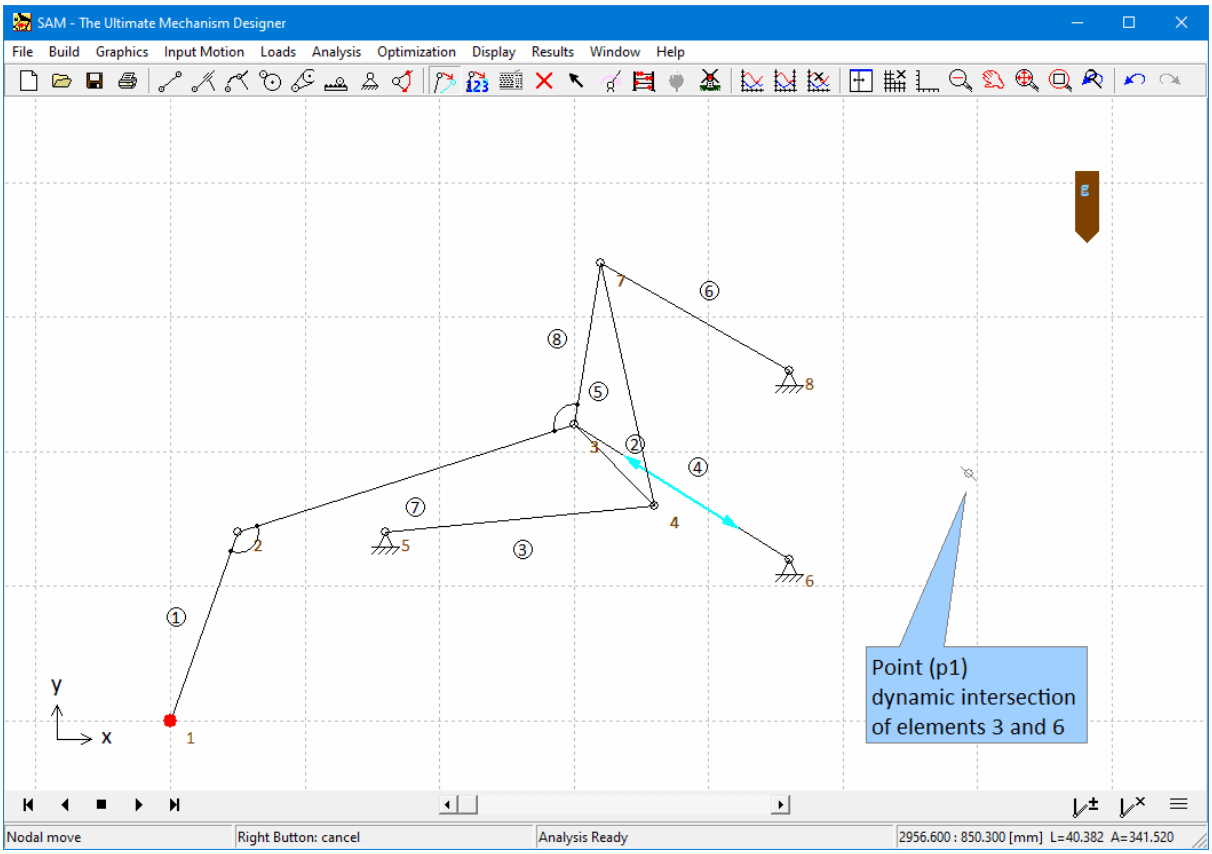
A template file is a DXF-file that contains exactly one point, one line, one circle, one arc and one text entry (these entities must be created in exactly that order !)

14.4 What was new in SAM 8.3

Dynamic Points
The possibility to construct points via the intersection of lines and even the intersection of centerlines has become dynamic. When moving the objects used in the creation of a point, the xy coordinates of that point are adapted automatically. That holds for the situation that an object is move in the reference position, but also when performing an animation.

In this way one can for example create point p1 as the intersection of beam 3 and beam 6, which constitutes the momentary center of rotation of the triangle formed by the triangular coupler. When the mechanism is animated and moves from one position to the next, the location of point p1 is adapted automatically.

© 2024 ARTAS - Engineering Software

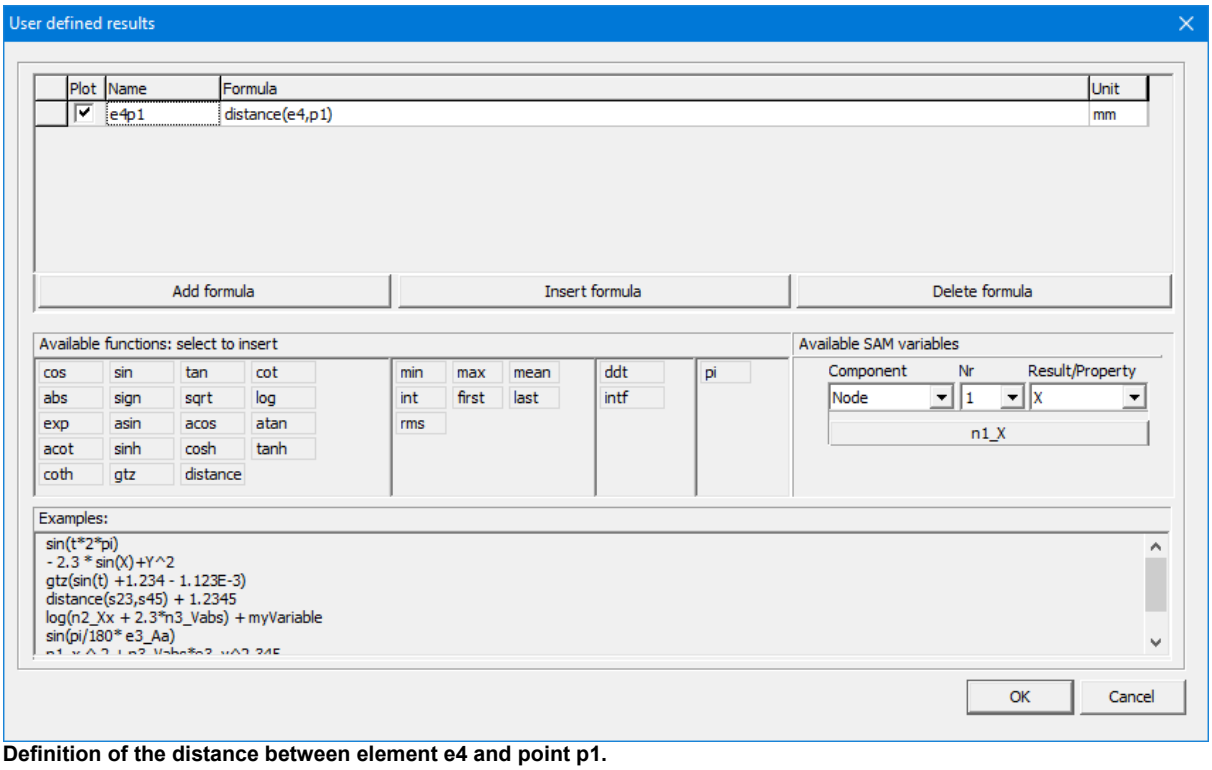


The point p1 is created as an intersection of the two (infinite) lines corresponding to the beam elements 3 and 6. The point is dynamic an moves during an animation.

Distance

The postprocessing module (Results > User Defined ...) has been extended with a new function: distance(object1,object2), in which an object can be either a node, element, point or shape.

© 2024 ARTAS - Engineering Software

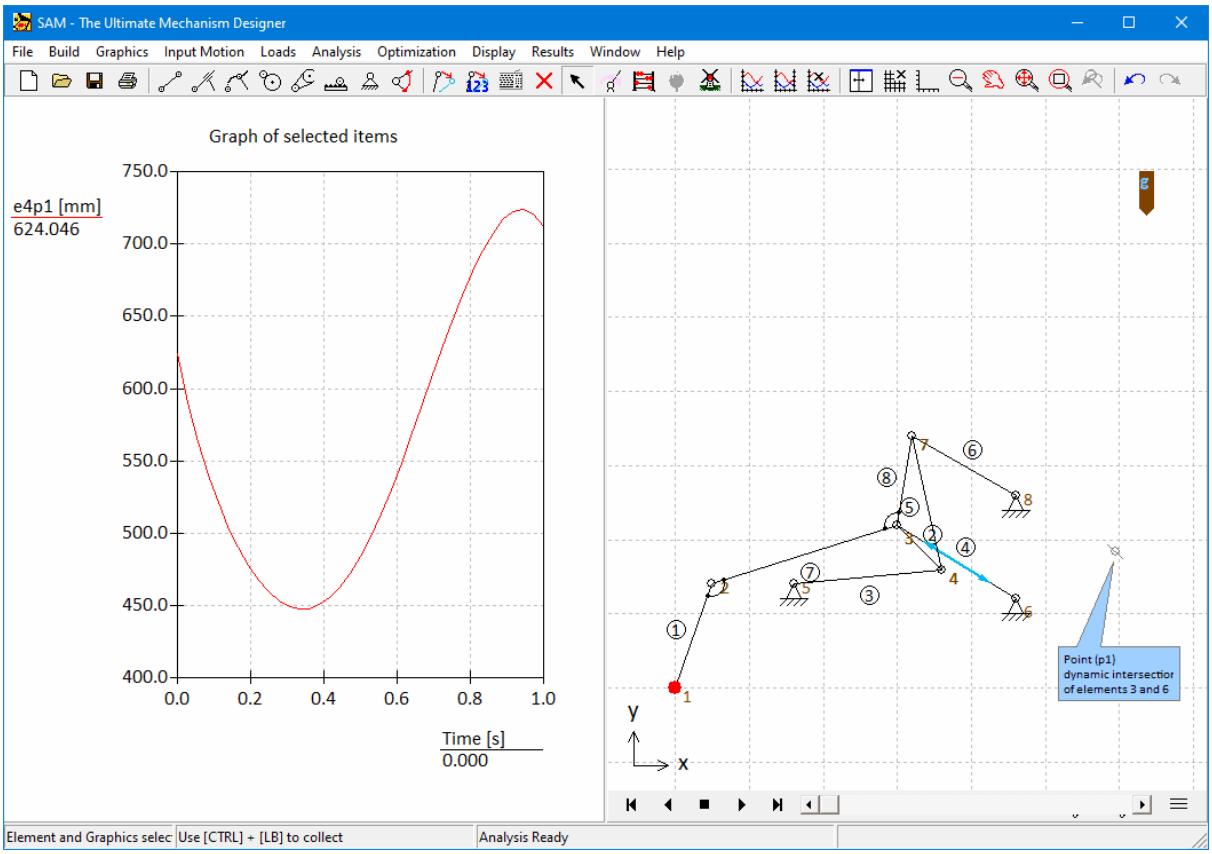


Definition of the distance between element e4 and point p1.

The use of this feature is manyfold:

1. Plot the distance between two objects as function of time. In the previous example it might be useful to plot the distance between the driving cylinder (element e4) and the dynamic point p1 in order to monitor if that distance is always positive and does not reach zero, since in that situation the mechanism would be blocked.

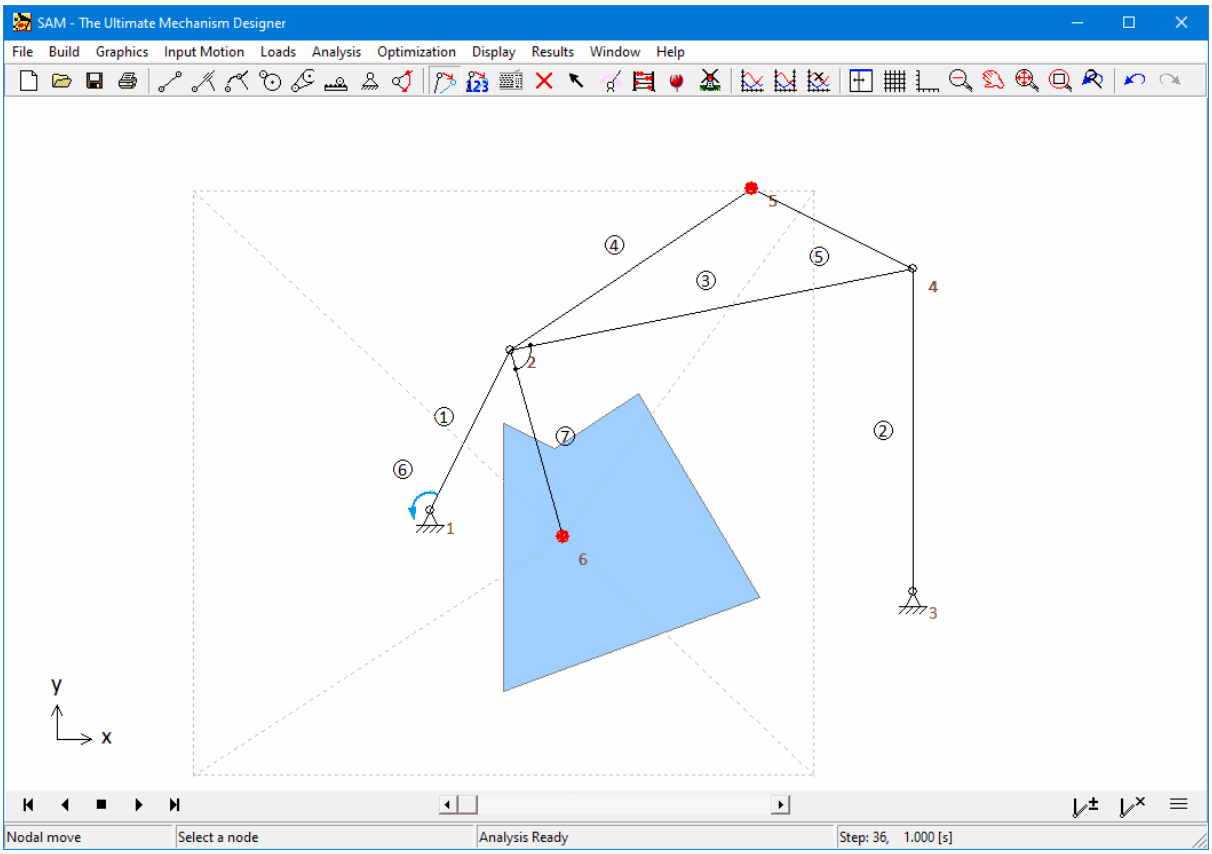
© 2024 ARTAS - Engineering Software



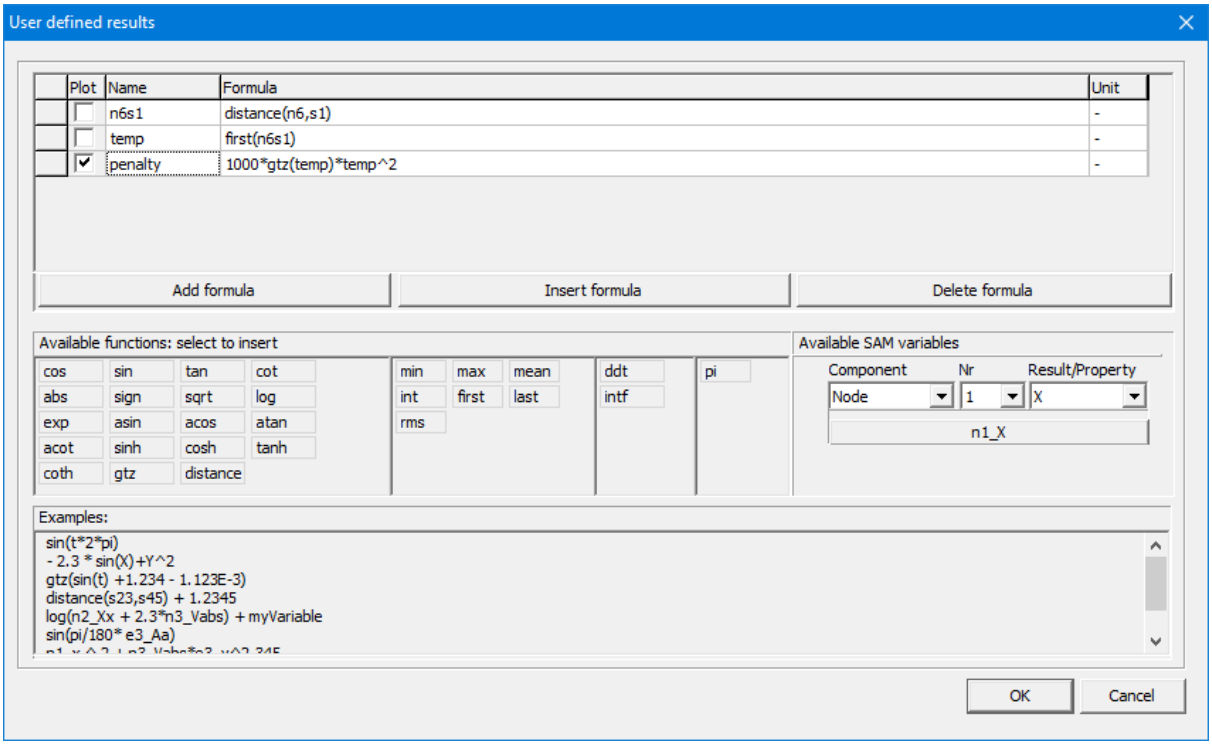
Distance between element e4 and point p1 as function of time is plotted next to the mechanism

2. During an optimization, the xy-location of a node can be an optimization parameter. Using the distance function one can define a penalty function for the optimization in order to constrain the xy-location of that node to any polygon

© 2024 ARTAS - Engineering Software



Example of an optimization in which node n6 is a design parameter but should be constrained to the area defined by the polygon (shape s1).

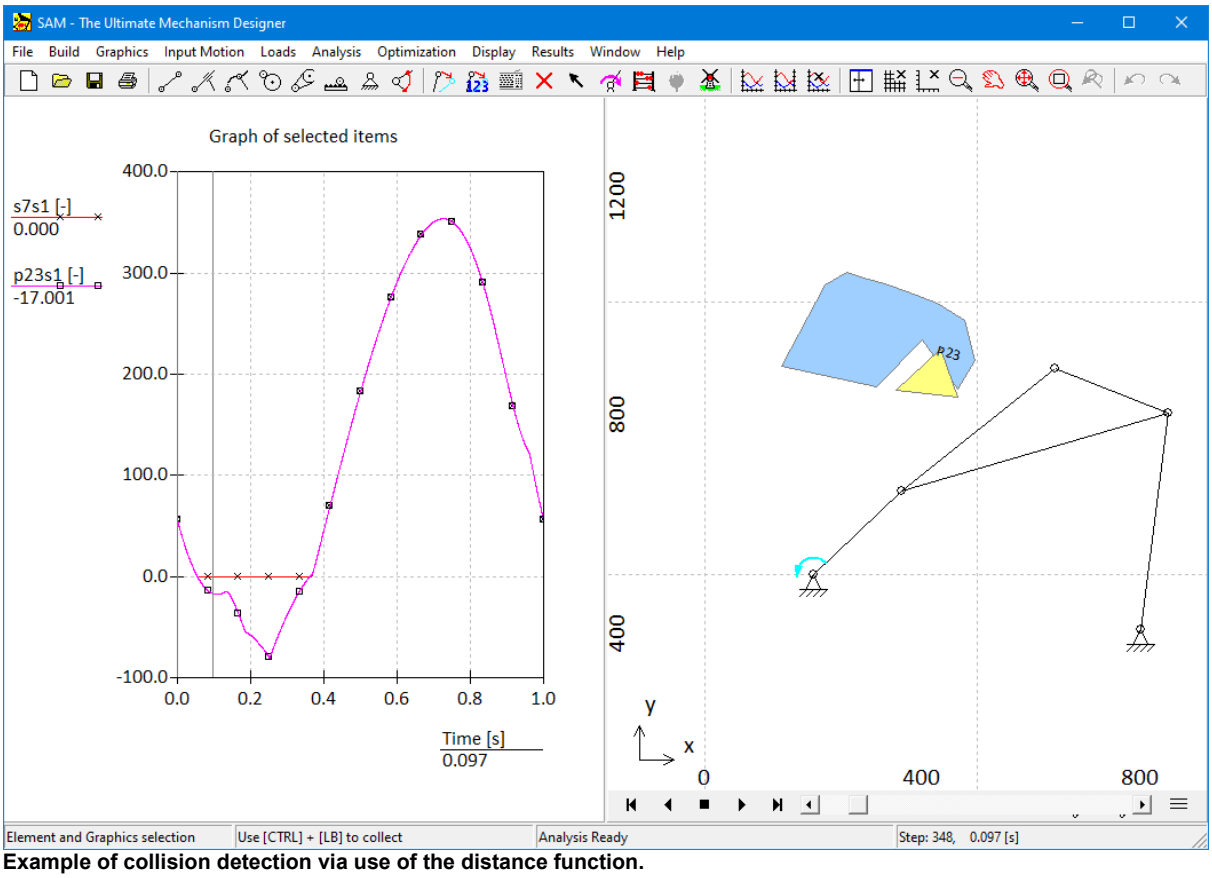


© 2024 ARTAS - Engineering Software

Definition of the penalty function to constrain node n6 to the polygon s1. As soon as node n6 lies outside of the polygon in the initial position of the mechanism, the penalty is larger than 0.

3. Collision detection between objects

The distance function can be used to detect collision between objects and depending on the type of objects even the penetration depth (or minimum retraction distance to avoid collision). In the following example the blue polygon is a stationary shape s1, whereas the yellow triangle (shape s7) is attached to the coupler. In the plot one sees the variable s7s1, which corresponds to the distance between the two shapes and the variable p23s1, which corresponds to the distance between point p23 on the triangle and the blue polygon. In case of two colliding shapes the distance is set to zero, whereas in the case of a point or node moving into a shape both the positive and negative distance (negative distance = penetration depth) is shown in the plot.



Example of collision detection via use of the distance function.

Copy/Paste (Ctrl-C/Ctrl-V) for graphics/shapes
One can now use Ctrl-C and Ctrl-V to copy and then paste any shape.

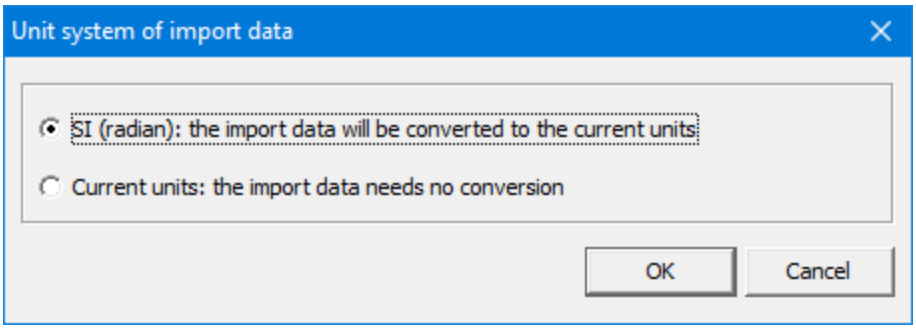
14.5 What was new in SAM 8.2

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.

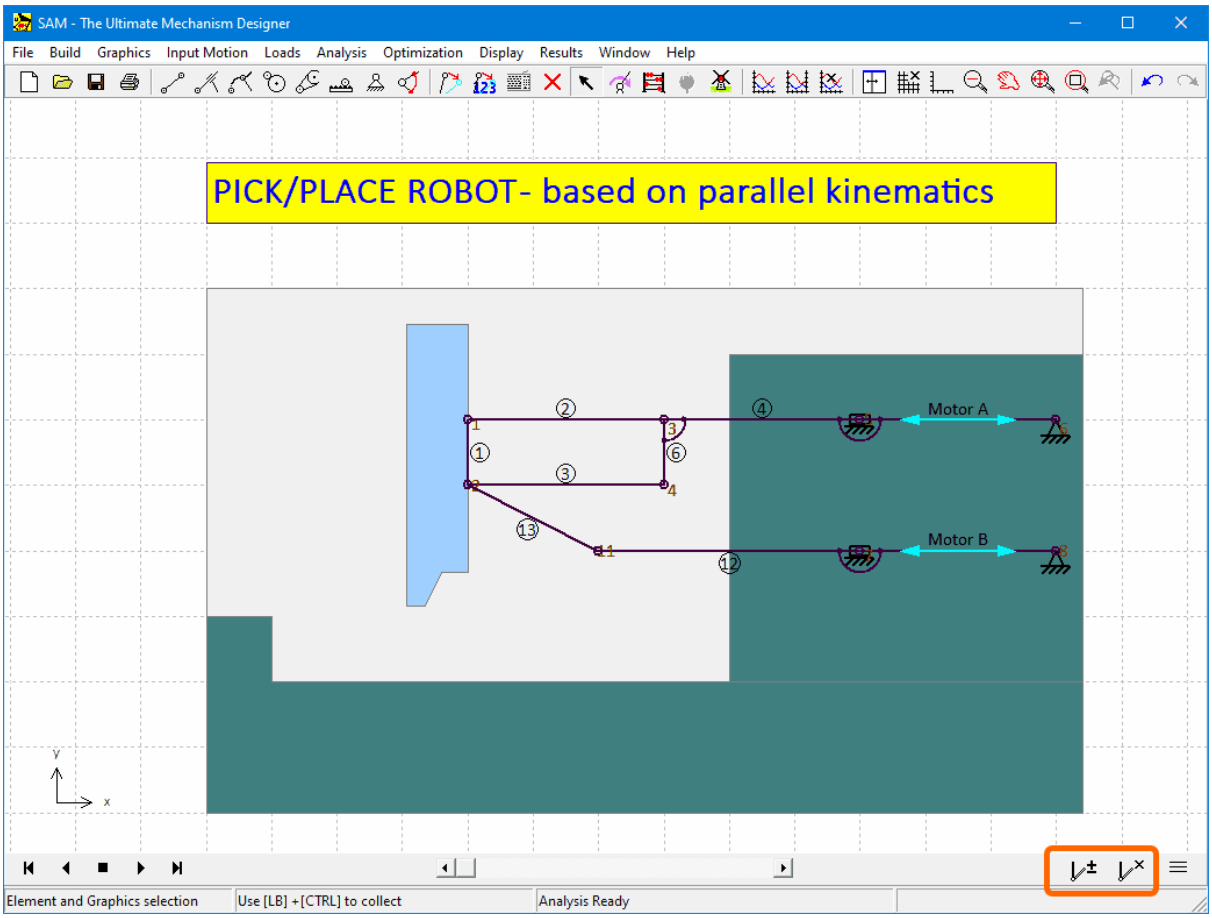
Text box including call-out

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.

© 2024 ARTAS - Engineering Software

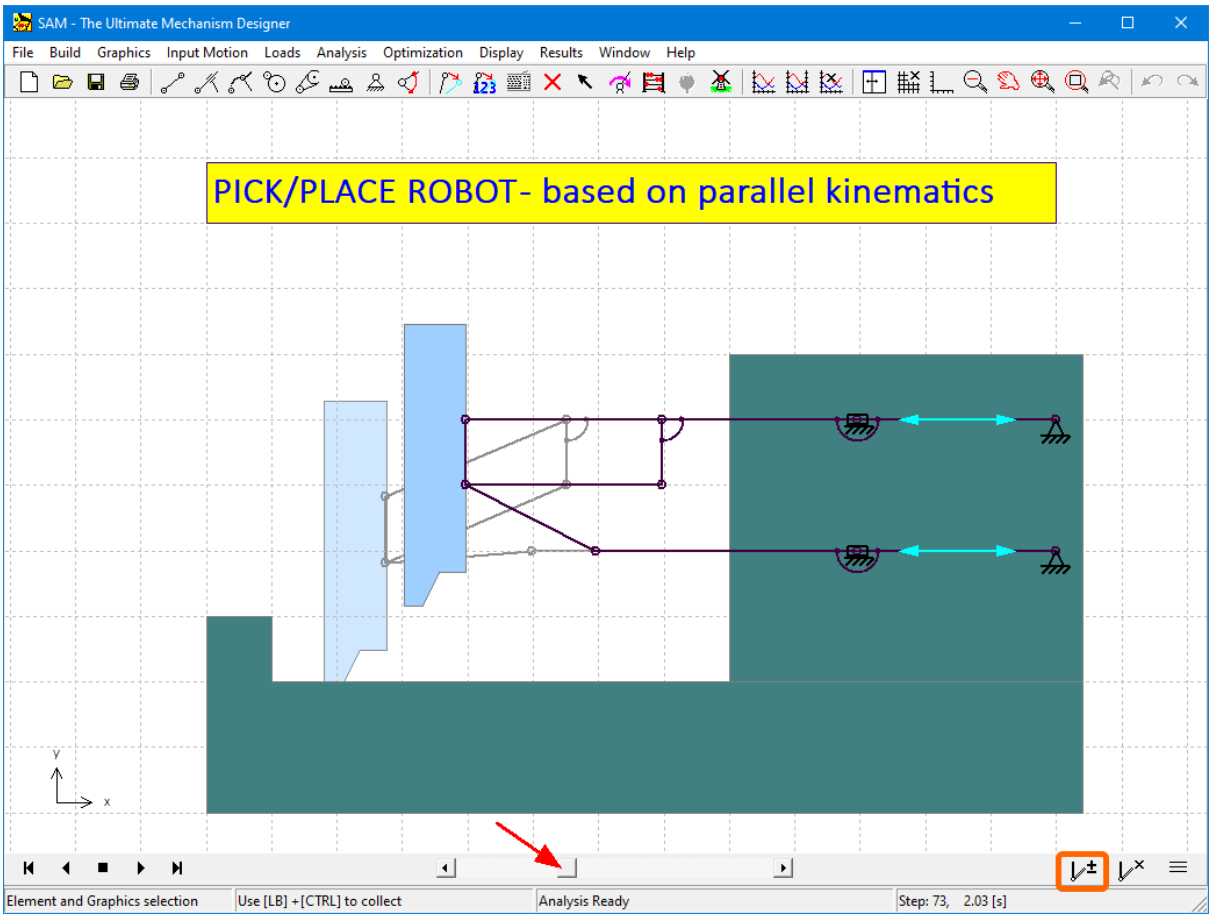


Example showing fill color and display order

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.

© 2024 ARTAS - Engineering Software



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

Cascading styles (Preferences - Project - Object)

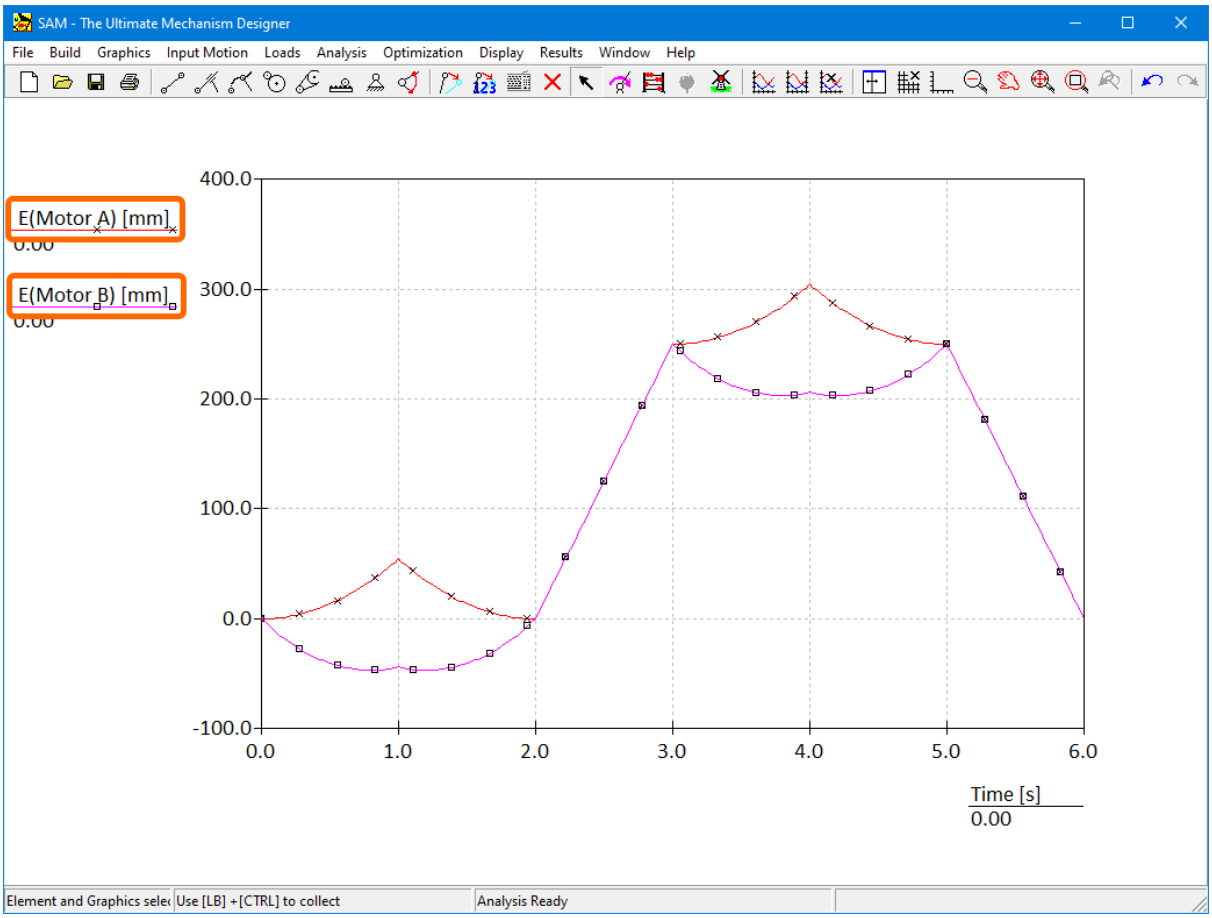
Various styles, like font size, (fill) color, ... can be defined at various levels. The settings at preferences level are the initial settings for each new project, but can be adapted at project level. Within each project, the settings at project level serve as initial values for each new object, but 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.

© 2024 ARTAS - Engineering Software

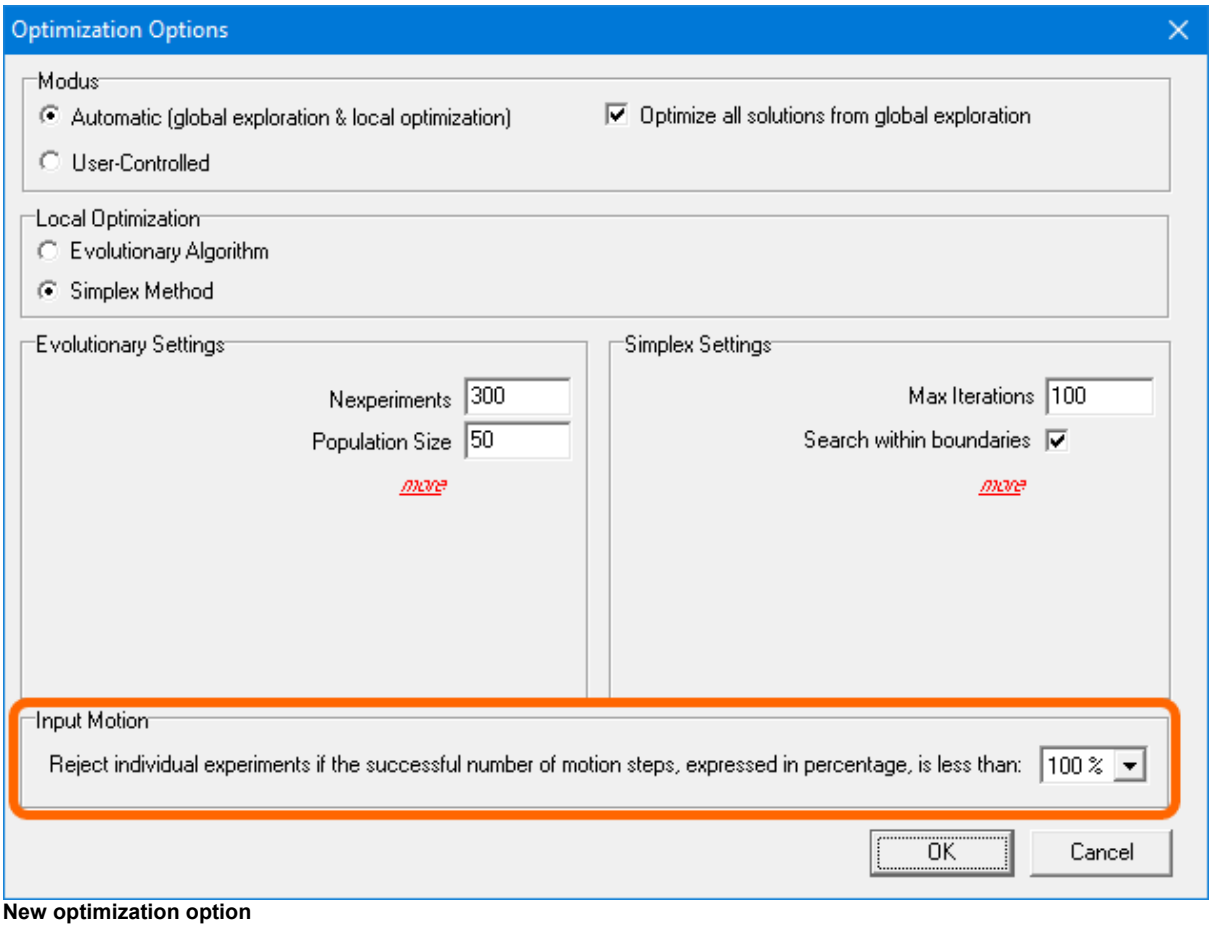


Motion (=elongation E) of the driving elements with labels 'Motor A' and 'Motor B' are shown as function of time

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%.

© 2024 ARTAS - Engineering Software

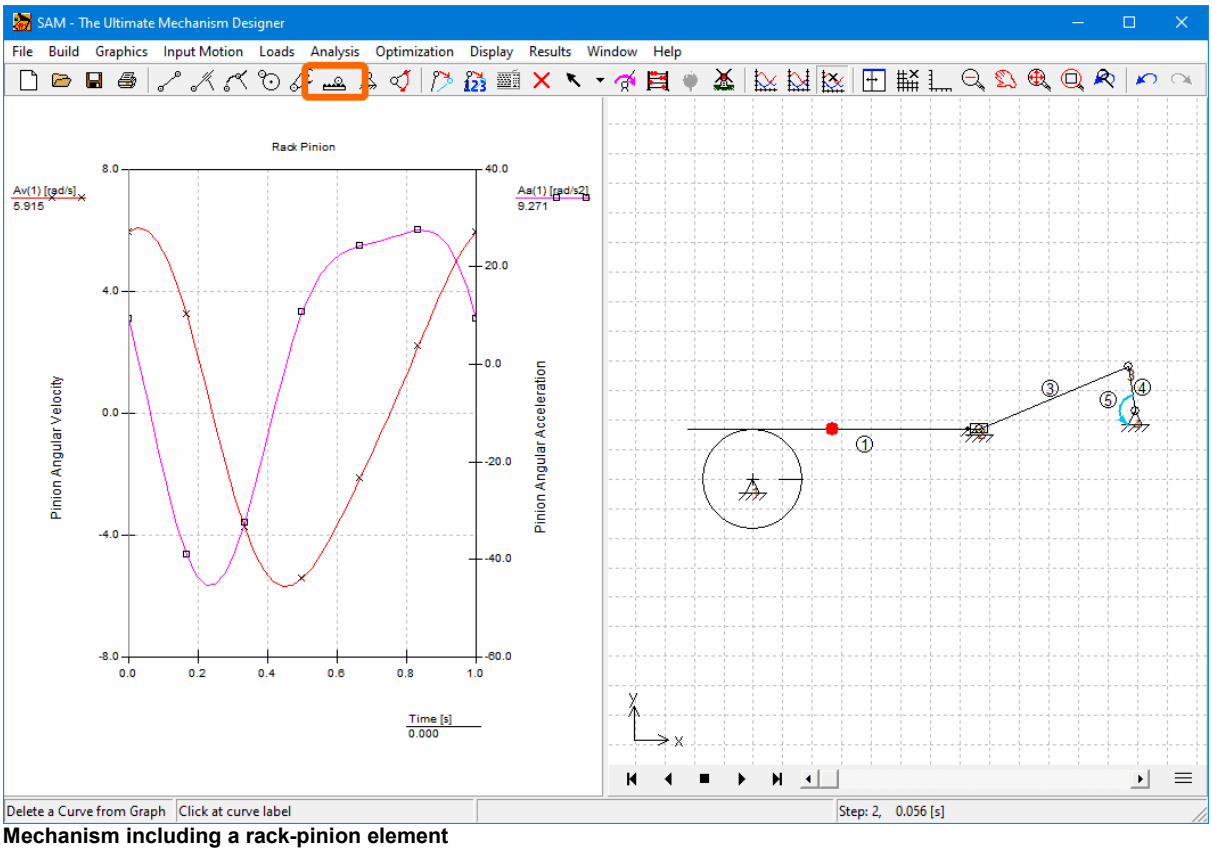


New optimization option

14.6 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.

© 2024 ARTAS - Engineering Software



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

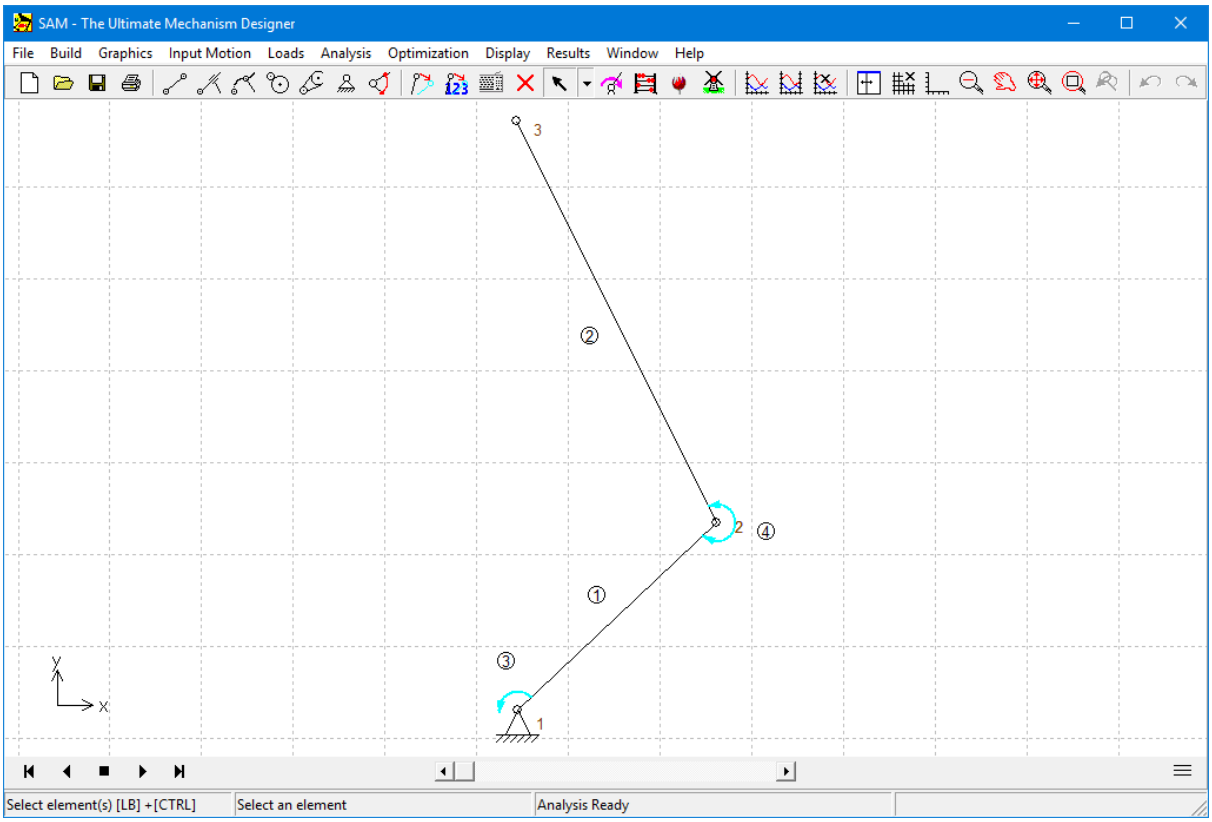
By optimizing the analysis process a huge increase of analysis/optimization speed has been realized. This manifests itself especially during an optimization, requiring typically thousands of analysis runs, in which optimization speed increases of up to a factor 4 have been realized.

14.7 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.

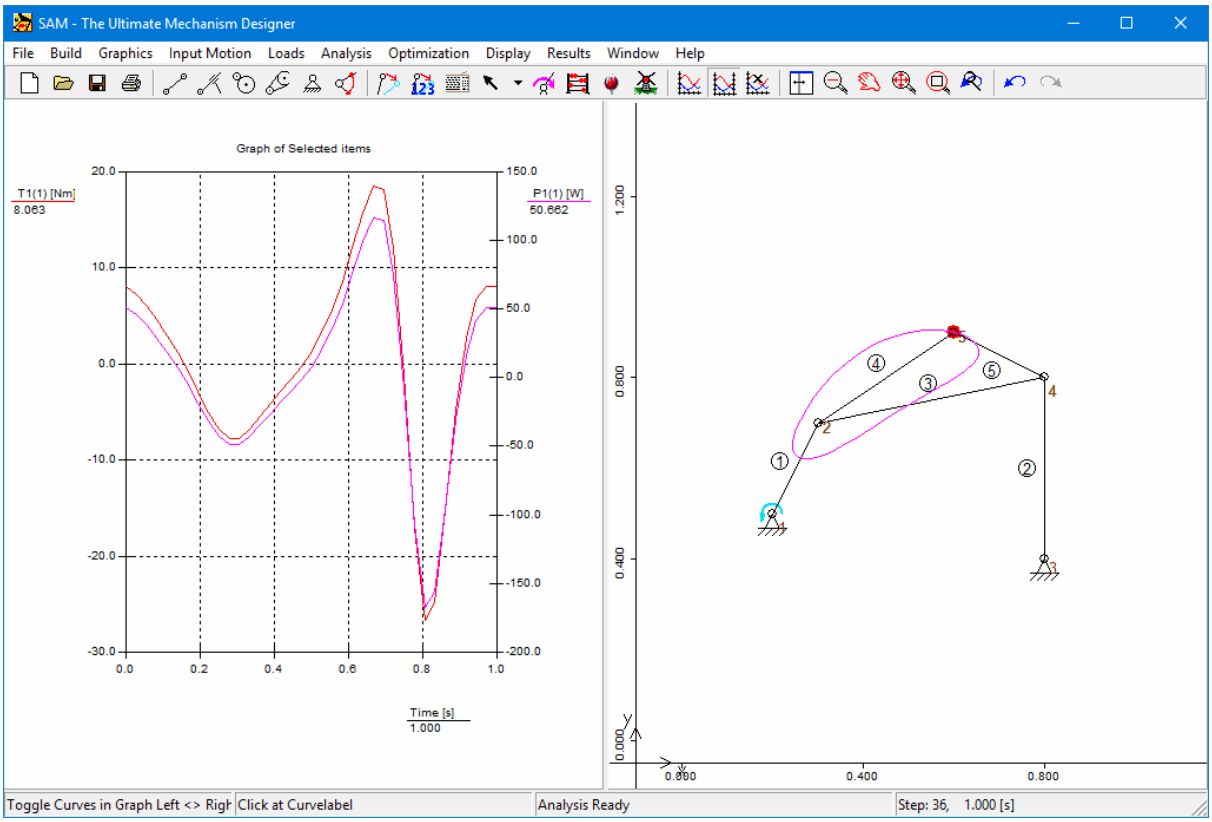
© 2024 ARTAS - Engineering Software



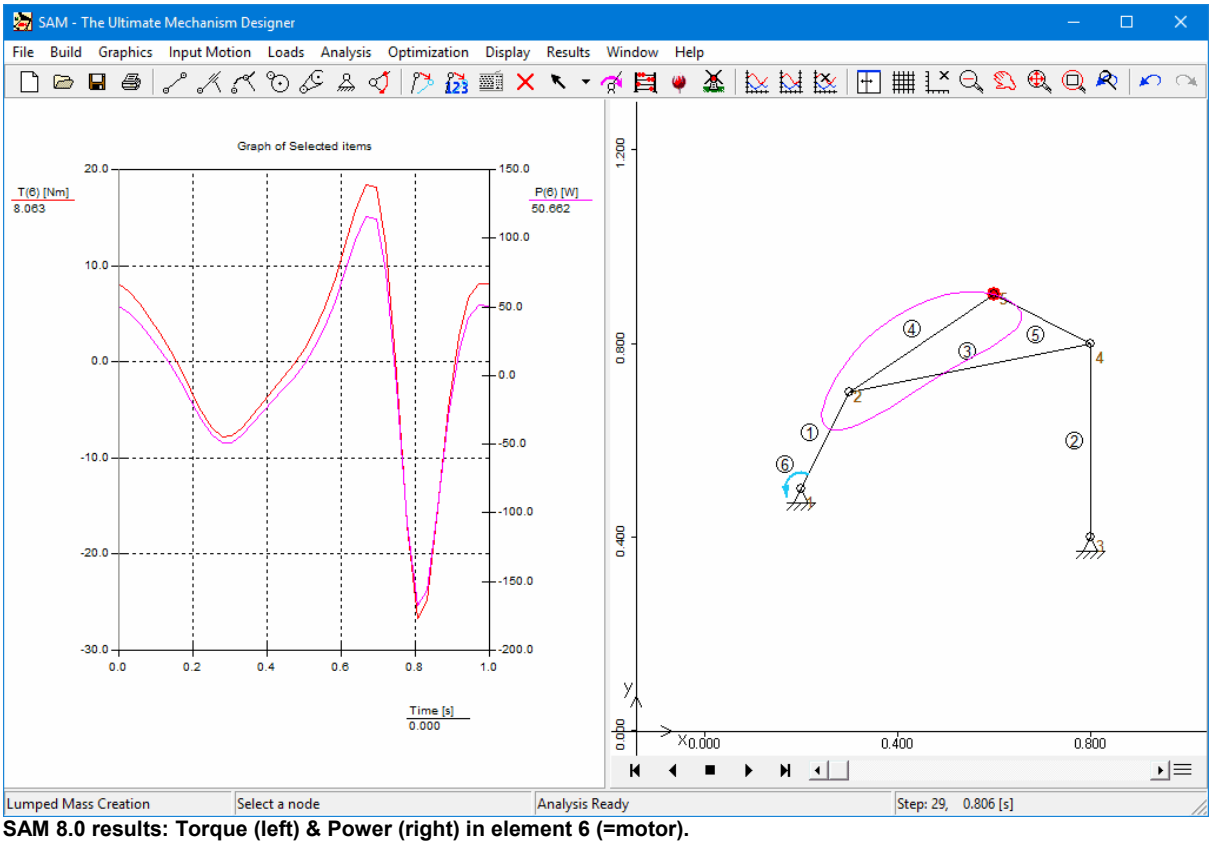
Example robot arm with two motor elements (3) and (4)

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

© 2024 ARTAS - Engineering Software



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

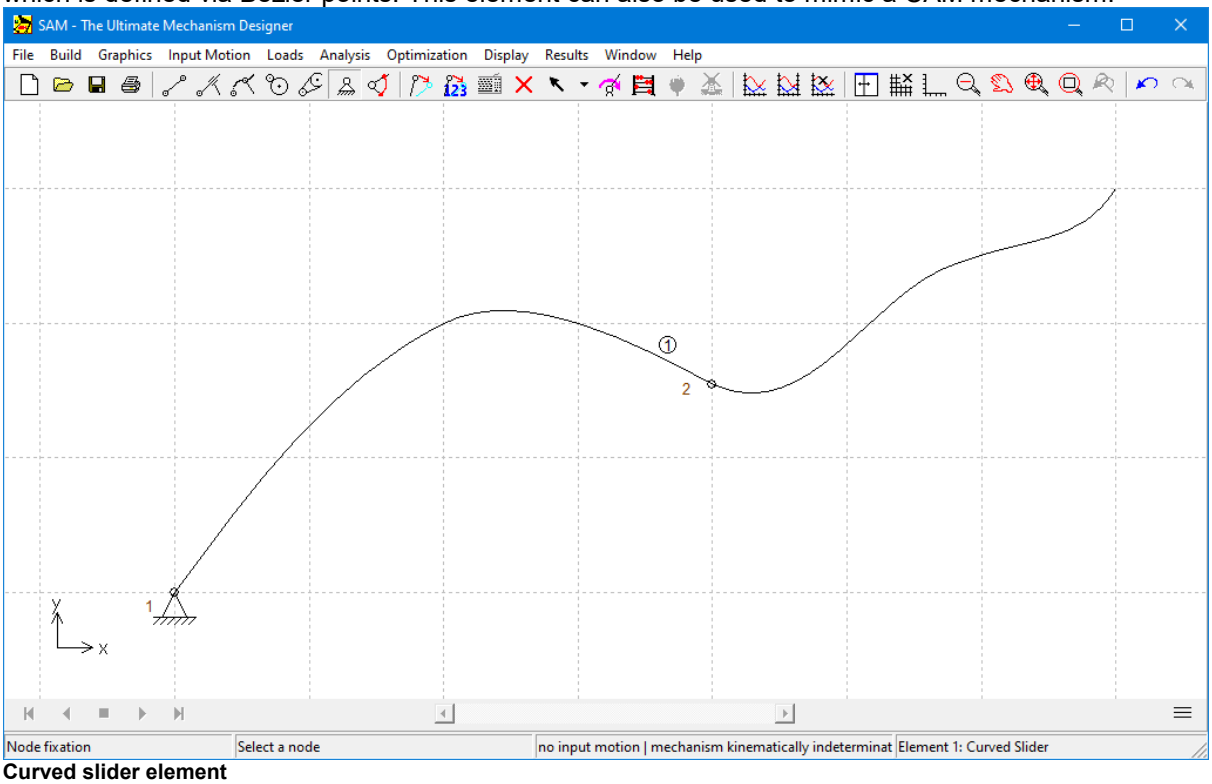


SAM 8.0 results: Torque (left) & Power (right) in element 6 (=motor).

© 2024 ARTAS - Engineering Software

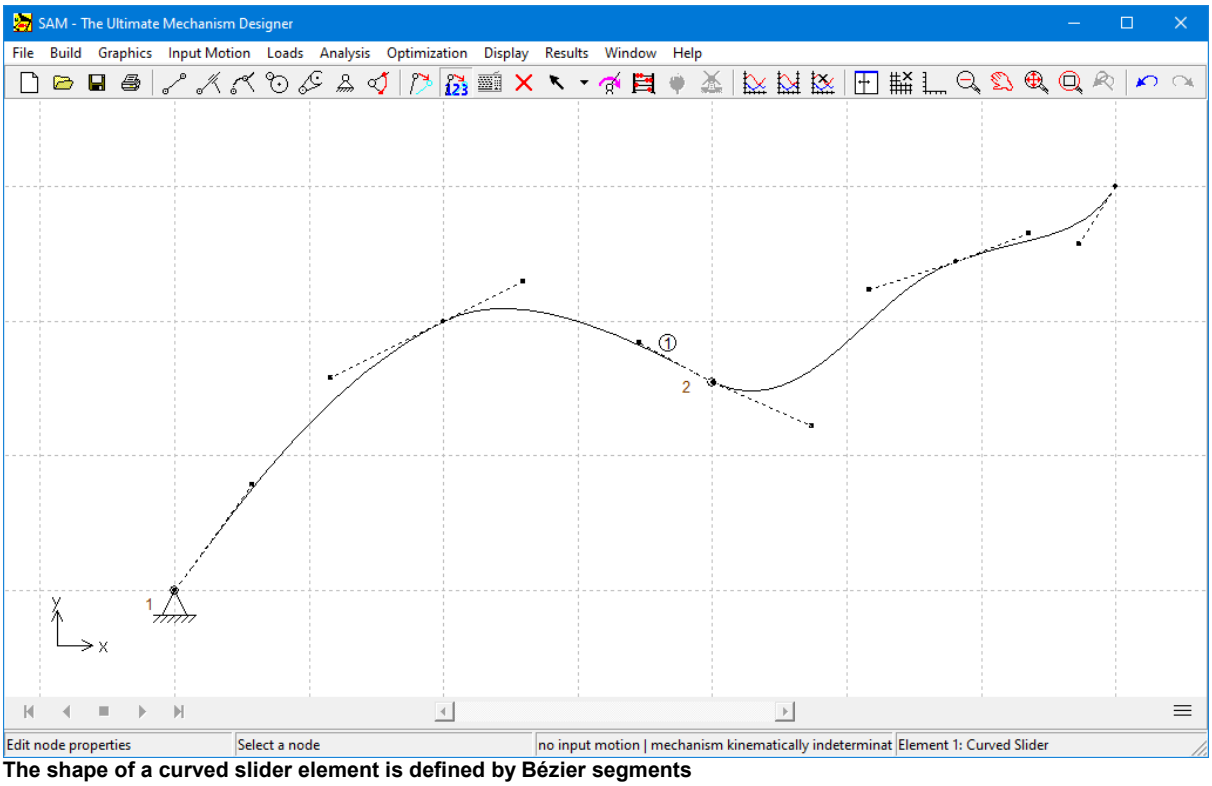
Curved Slider

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.

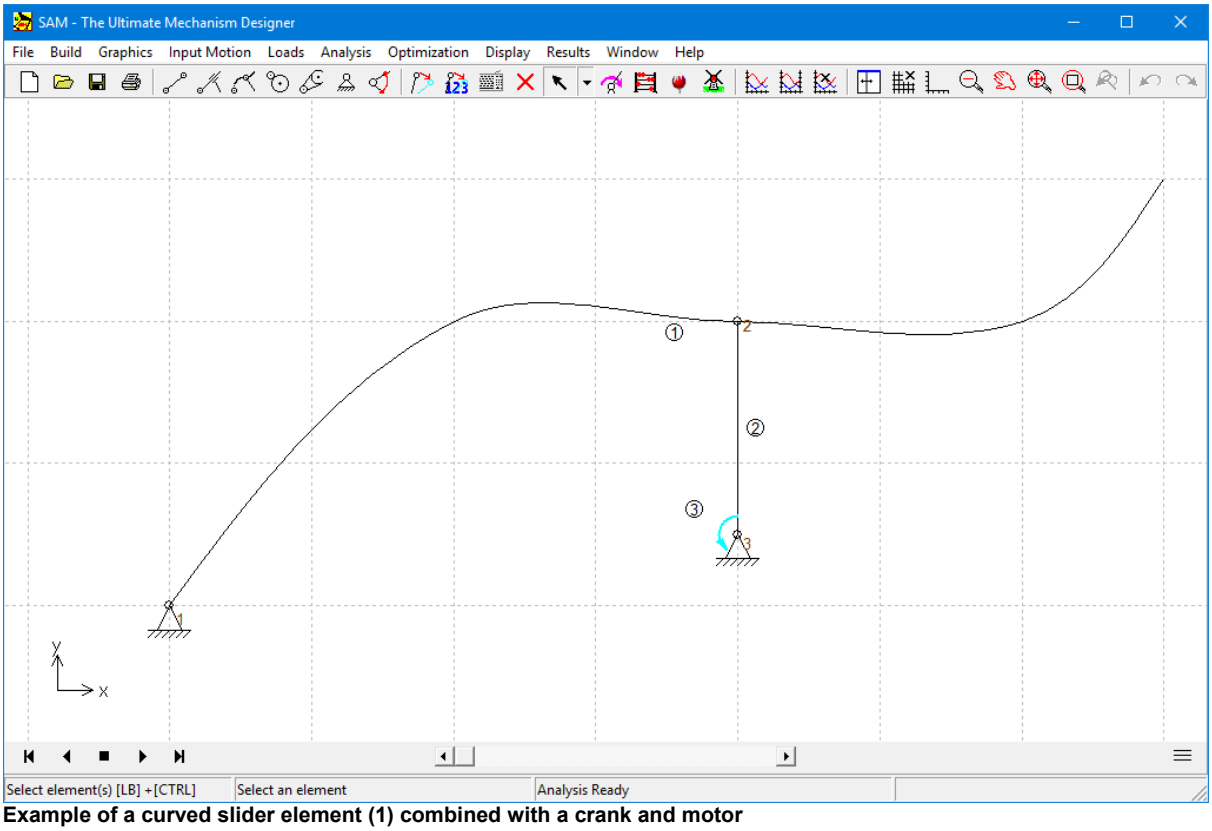


Curved slider element

© 2024 ARTAS - Engineering Software



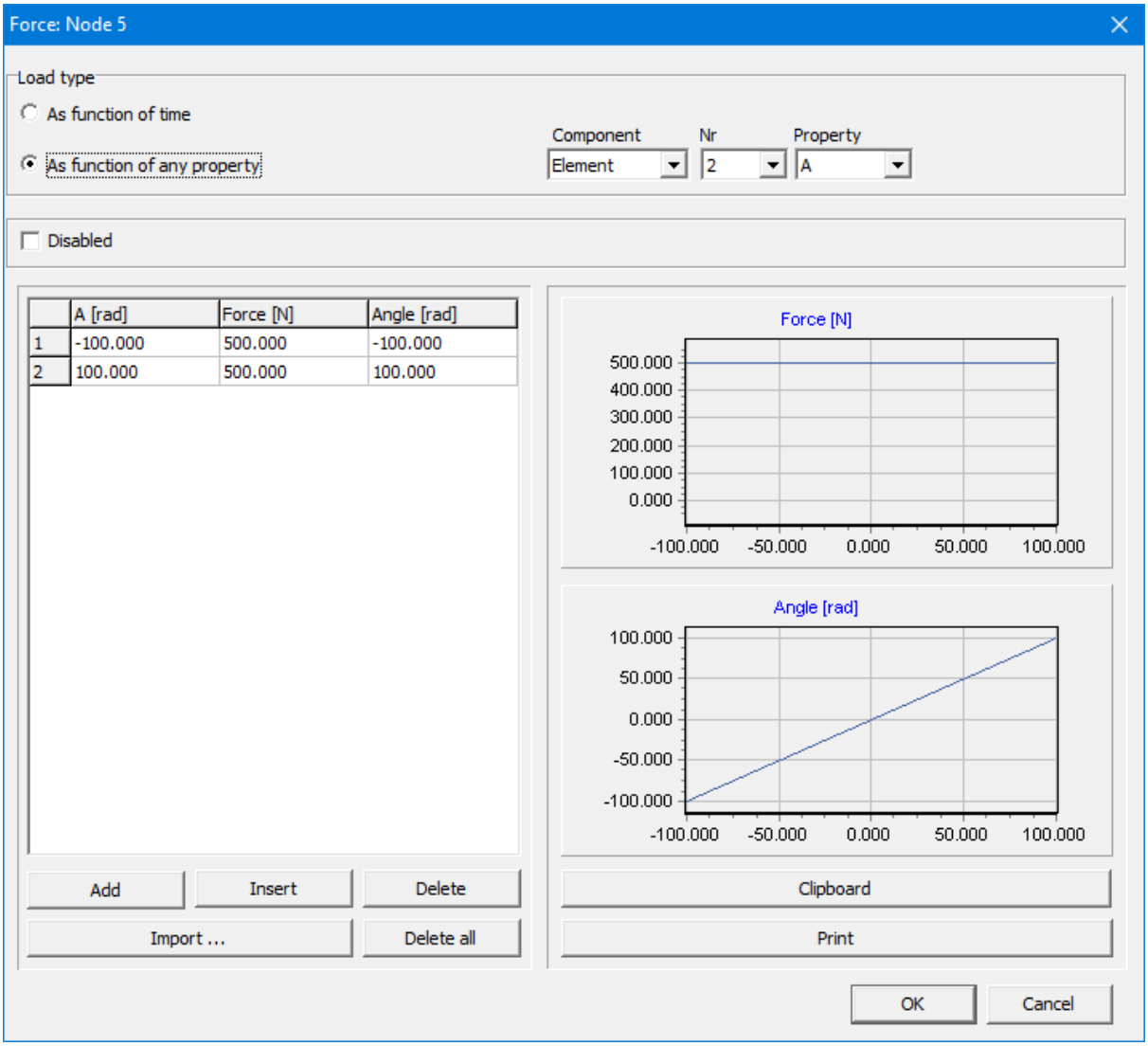
The shape of a curved slider element is defined by Bézier segments



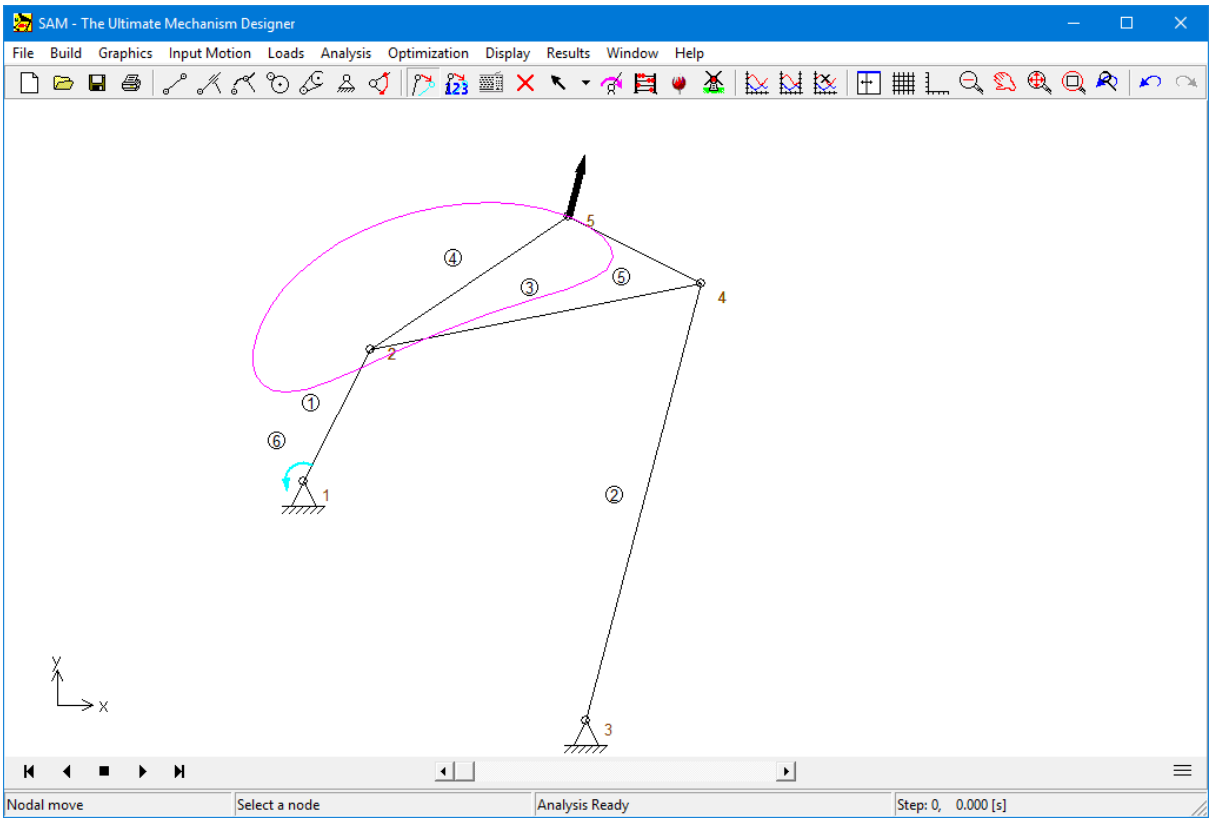
Example of a curved slider element (1) combined with a crank and motor

© 2024 ARTAS - Engineering Software

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

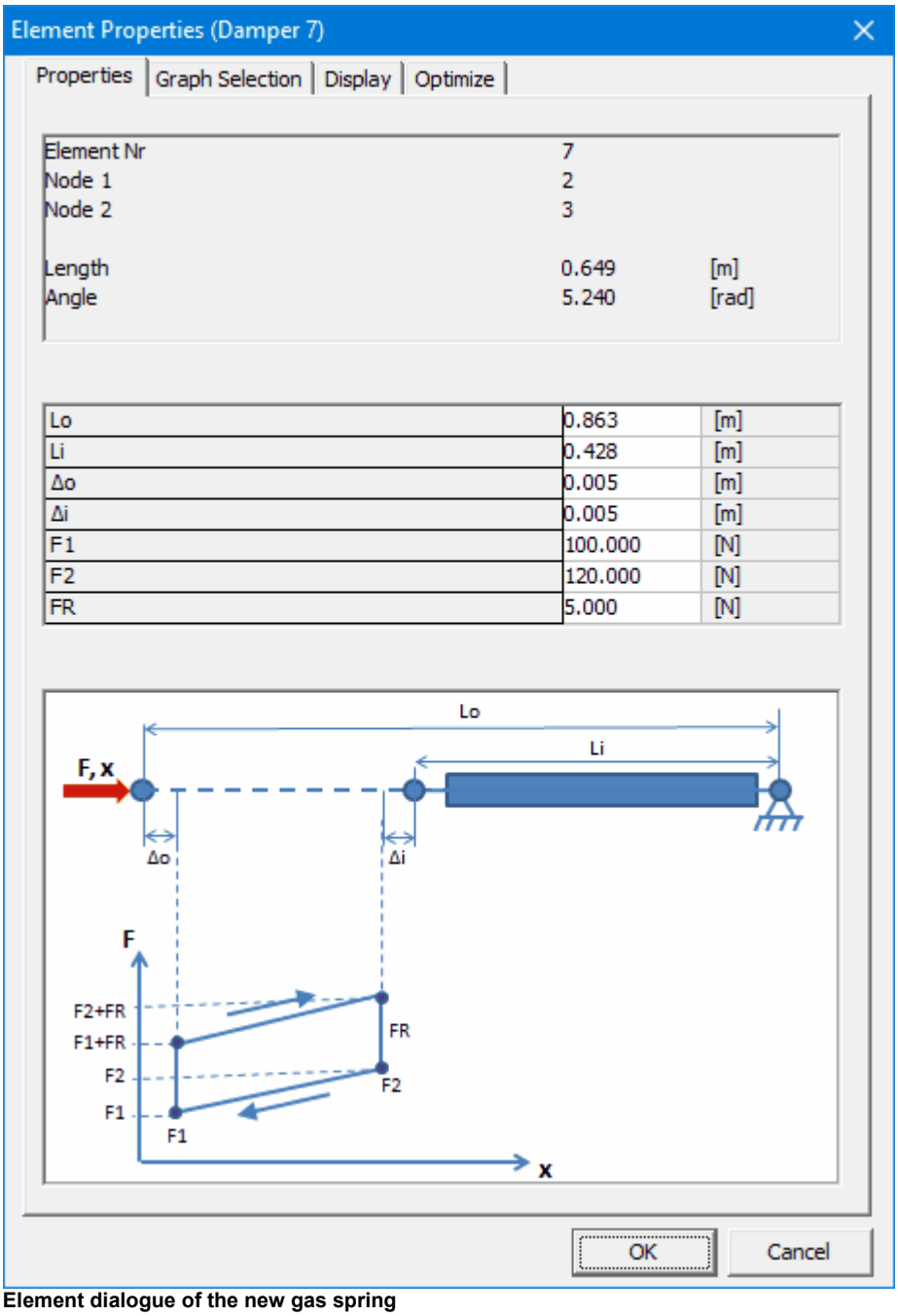


Example: force of 500N on node 5 with a direction identical to the angle of element 2)

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.

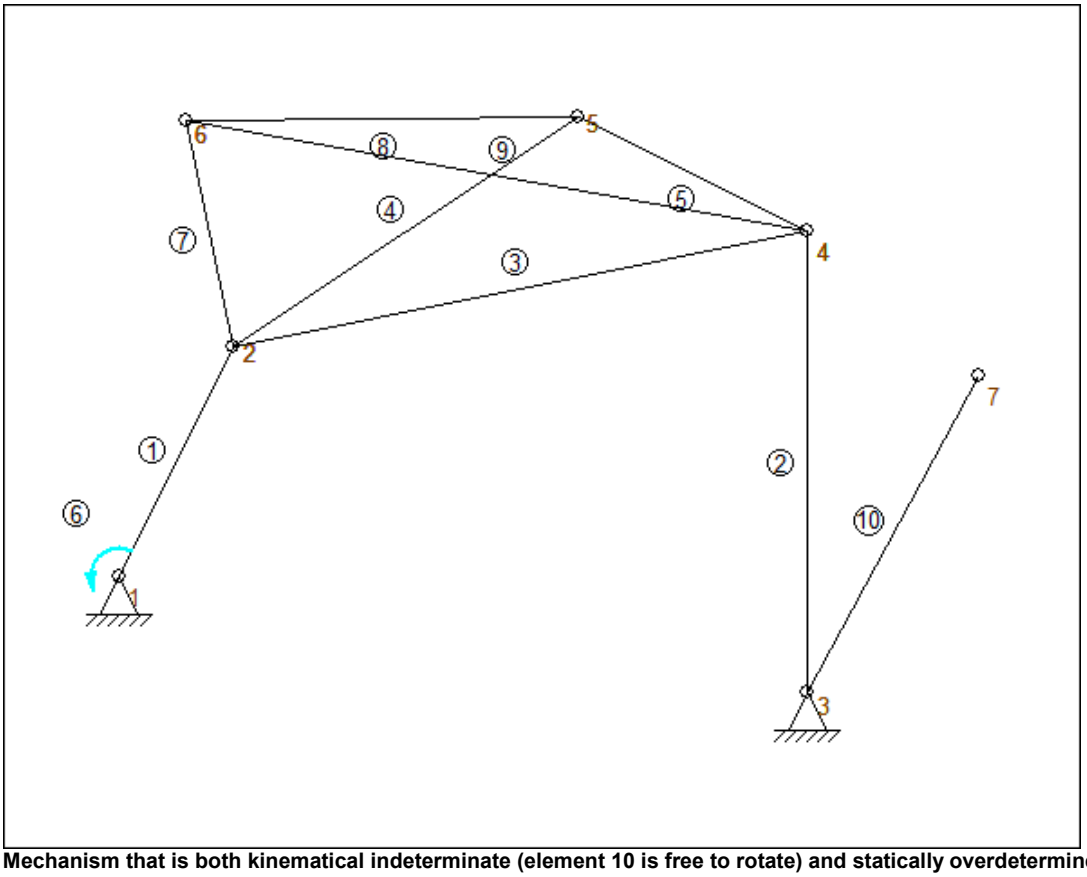
© 2024 ARTAS - Engineering Software



Element dialogue of the new gas spring

© 2024 ARTAS - Engineering Software

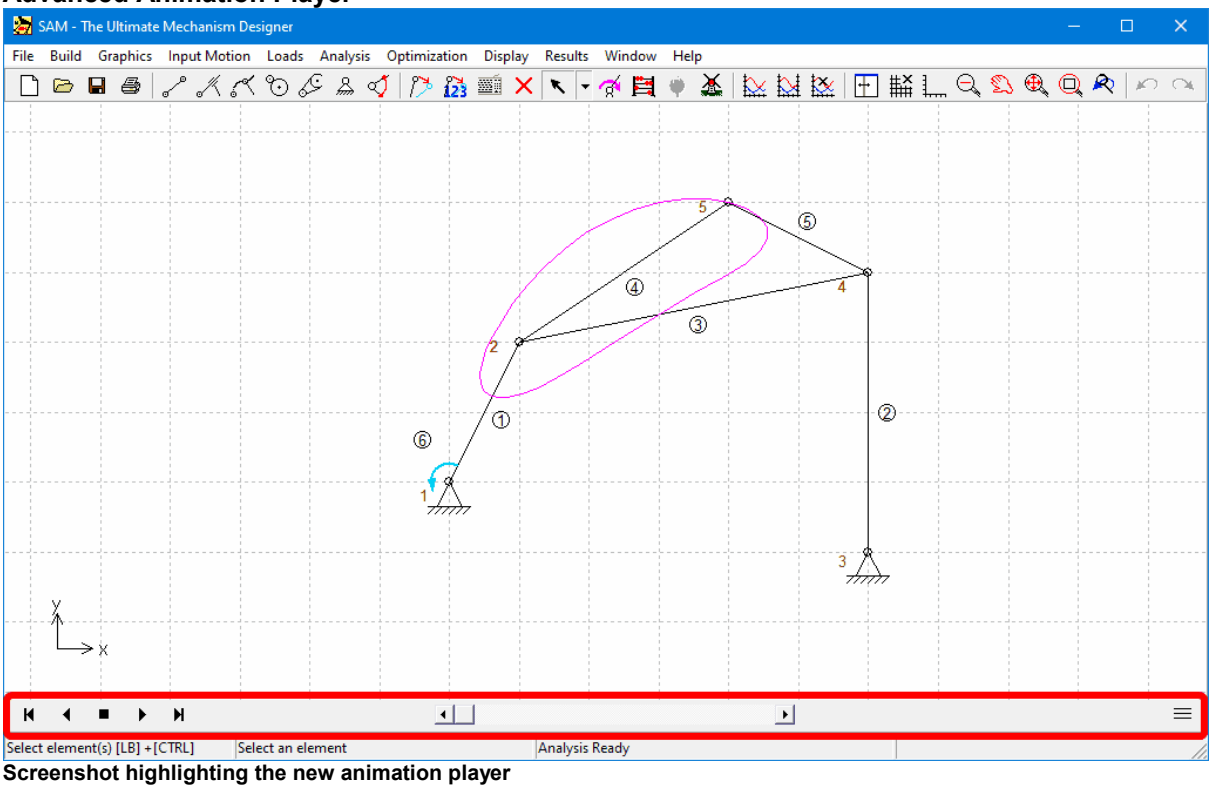
Mobility analysis based on Singular Value Decomposition (SVD)
A SVD analysis is introduced to analyse whether the mechanism is kinematical indeterminate and/or statically overdetermined. In contrast to the previous analysis based on the theory of Grübler, the SVD is able to give the right diagnosis in 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)

© 2024 ARTAS - Engineering Software

Advanced Animation Player



Screenshot highlighting the new animation player

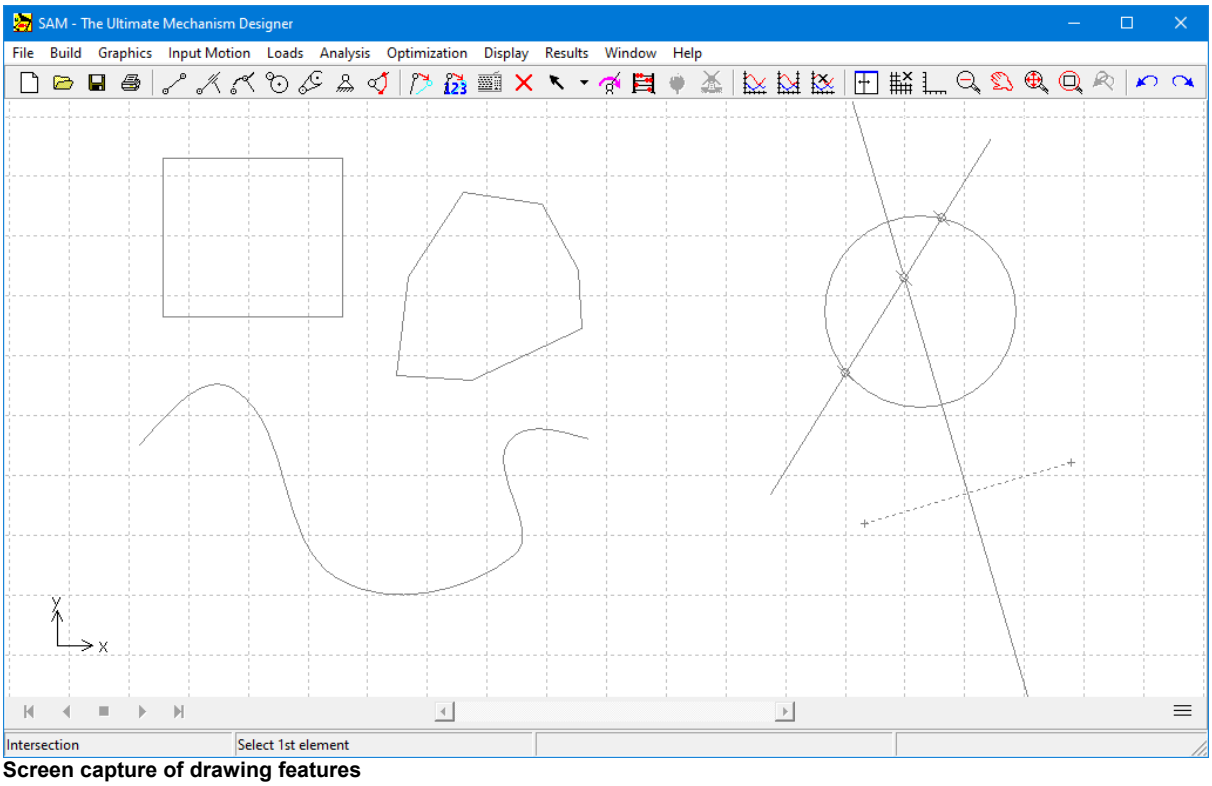
(Semi-) dynamic construction aid

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

© 2024 ARTAS - Engineering Software



Screen capture of drawing features

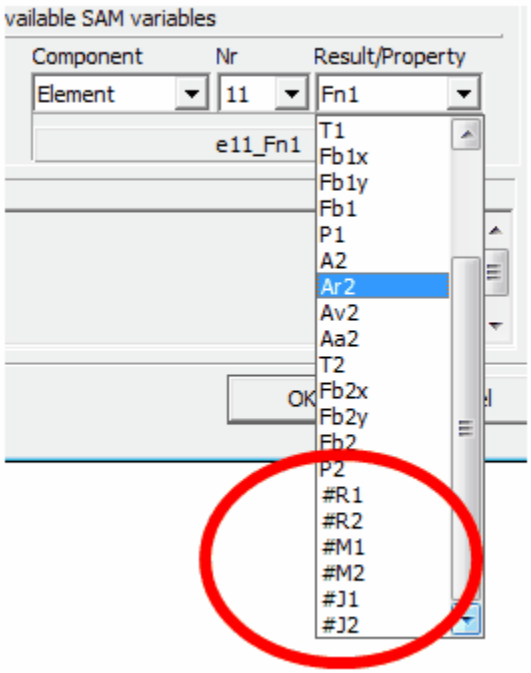
New functions available in formula editor (User Defined Results)

- 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.

© 2024 ARTAS - Engineering Software



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

14.8 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, ...

© 2024 ARTAS - Engineering Software

14.9 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.

14.10 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.

14.11 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

© 2024 ARTAS - Engineering Software

reference function. When designing for example fitness equipment one is gnerally 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 of 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

© 2024 ARTAS - Engineering Software

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

14.12 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).

14.13 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:

© 2024 ARTAS - Engineering Software

- 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 ars available

Generation of video file (AVI-format)

14.14 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 geartransmission 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

14.15 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.

© 2024 ARTAS - Engineering Software

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 indentical 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 eachother (Windows/Tiled) and the graph cursor is ON, you can move this cursor from left to right and the mechanism will move accordingly.

14.16 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).

© 2024 ARTAS - Engineering Software

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" opertion 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(x2+y2)) can be plotted.

© 2024 ARTAS - Engineering Software

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 is 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.

© 2024 ARTAS - Engineering Software

Index

- 3 -

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

- 4 -

4-Bar Mechanism 58, 60

- A -

A quick example 30, 85, 160, 182, 185, 187, 228
About this User's Guide 13, 23, 24, 26, 30, 80, 126, 182, 183, 187, 226, 228
Analysis Menu 193
Analysis Problems 200
Analysis Results 21
Approximated Straight Line Mechanism 69, 70, 71, 72, 73, 74, 226

- B -

Beam 140
Bearing Forces 94
Belt 160
Belt drive with external force 123
Belt-Coupled Dyad 69
Building a new mechanism (without Design Wizard) 45, 46, 186, 188, 189
Building Multi-Node Linkages 84

- C -

CAD Interface 21, 30, 80, 85, 185, 187, 201, 228
closed Bezier curve 147
collision 109
collision detection 76
Constrained Optimization 22, 130
Crank Rocker Mechanism 114

- D -

Design 16, 60, 69, 226
Display Menu 194, 227
Distance 88
Double Crank Mechanism 117
Driving Torque Minimization 121, 126
DXF Template File 201

- E -

European Academic Software Award 24
Exact Nodal Coordinates 80, 182, 183, 186
Exact Straight Line Mechanism 64
Example of a 4-bar mechanism (motion analysis) 46, 58, 80, 104, 105, 108, 110, 140, 182, 183, 186, 228
Exporting a motion file 82

- F -

force analysis 94
force in a local coordinate system 97
Force/Torque Dialog Box 191
Forces in Moving Nodes 94
Format 82

- G -

Gear 151
Geared Double-Crank Mechanism 66
General (nodes) 58
Graphics Menu 187

- H -

Help Menu 195
History
Present and Future 23, 80, 82, 107, 114, 227
Hypo-Cyclic Planetary Gear Mechanism 67

- I -

Import points 147
Import points and control points 147

© 2024 ARTAS - Engineering Software

Importing a motion file 82
Inclined Linear Guiding 83
Input Dialog Box 189, 226
Input Motion 17, 41, 45, 46, 62, 64, 81, 82, 126, 182, 188, 189, 191, 194, 200, 201, 226, 228
Installing SAM 26
Introduction 104

- K -

Keyboard and Mouse 182

- L -

LB 182
Linear Actuator 83, 140, 188
Loads Menu 191

- M -

Maximize(cost function+penalty) 105
Modelling 17, 23, 46, 60, 69, 83, 84, 123, 143, 151, 160, 164, 170, 183, 186, 187, 188, 200, 226, 227

- N -

Node-Locked License 26
Non-linear spring 175, 177

- O -

Objective Performance Number 105, 107
Optimization Menu 193
Optimization Objective 105, 128
Optimization of trajectory 128
Optimization Options 110, 193
options 185
Overview - Capabilities - Optimization 22, 23

- P -

Penalty Function 105
Planetary Gear-Coupled Dyad 68
Planetary Mechanism (Hypo-Cyclic) 117
Post-Processing 22, 30, 41, 45, 46, 105, 107, 114, 121, 126, 128, 194, 226, 227
Preference 185
project options 185
Purchase Agreement 13, 30, 85, 170, 182, 187

- Q -

Quick-Return Mechanism 114

- R -

RB 182
Reaction forces in fixed node 94
Reference or Target Definition 107
Running a sample project 30, 45, 46, 85, 185, 193, 194

- S -

Settings Evolutionary Algorithm 112
Settings Simplex Algorithm 112
Slider 143
Slider Driven Gear Transmission 116
Symmetric Slider-Crank Mechanism 65

- T -

The Design Process 16, 17, 30, 41, 46, 58, 59, 60, 62, 64, 65, 83, 84, 116, 126, 128, 183, 186, 187
Transfer Mechanism (incl. gravity effects) 121
trial license 26

- U -

Units 80, 81, 82, 188, 189, 194, 228
User Defined Results 86
Using Graphics 85
Using the Design Wizard to create a mechanism 41, 58, 62

- W -

What is SAM ? 8, 13, 17, 21, 22, 23, 41, 45, 46, 67, 68, 80, 84, 85, 94, 104, 105, 107, 108, 110, 112, 116, 117, 126, 128, 151, 164, 170, 182, 183, 185, 186, 187, 188, 193, 194, 200, 224, 226, 227
What was new in SAM 4.0 ? 228
What was new in SAM 4.1 ? 227

© 2024 ARTAS - Engineering Software

What was new in SAM 4.2 ? 227
What was new in SAM 5.0 ? 226
What was new in SAM 5.1 ? 226
What's new in SAM 6.0 ? 13, 17, 22, 23, 30, 45, 104, 108, 110, 112, 121, 123, 126, 151, 160, 164, 170, 175, 177, 183, 186, 195, 224, 226, 227
Window Menu 195

© 2024 ARTAS - Engineering Software