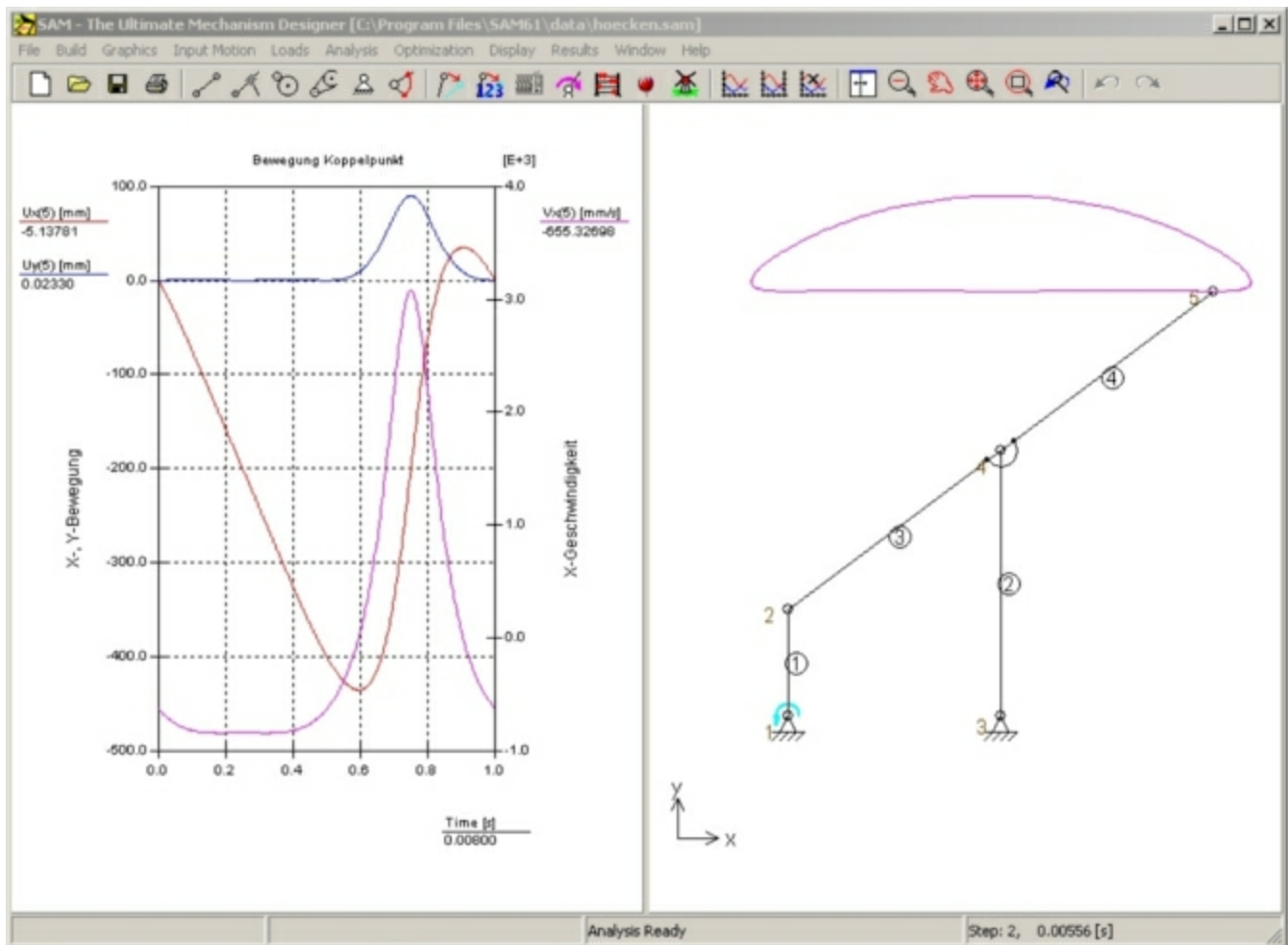


SAM 7.0

The ultimate mechanism designer



Contents

1 Preface	8
1.1 What is SAM ?.....	8
1.2 What's new in SAM 7.0.....	8
1.3 About this User's Guide.....	9
1.4 Purchase Agreement.....	10
2 Overview	12
2.1 The Design Process.....	12
2.2 Capabilities.....	12
Design	12
Modelling	13
Input Motion	13
CAD Interface	13
Analysis Results	13
Post-Processing	14
Optimization	14
2.3 History, Present and Future.....	15
2.4 European Academic Software Award.....	16
3 System Requirements & Installation	18
3.1 System Requirements.....	18
3.2 Installing SAM.....	18
3.3 Use of SAM 3.0, 4.x, 5.x and 6.x project files.....	19
4 A Guided Tour	22
4.1 Running a sample project.....	22
A quick example	22
4.2 Using the Design Wizard to create a mechanism.....	33
4.3 Building a new mechanism (without Design Wizard).....	38
Example of a 4-bar mechanism (motion analysis)	39
5 Design Wizards	52
5.1 4-Bar Mechanism.....	52
General (nodes)	52
General (nodes, angle, dimension)	53
Angle Function Synthesis	54
3-Position Synthesis (I)	57
3-Position Synthesis (II)	59
5.2 Exact Straight Line Mechanism.....	59
Symmetric Slider-Crank Mechanism	60

Geared Double-Crank Mechanism	60
Hypo-Cyclic Planetary Gear Mechanism	61
Planetary Gear-Coupled Dyad	61
Belt-Coupled Dyad	62
5.3 Approximated Straight Line Mechanism.....	62
Watt	63
Chebyshev	63
Roberts	64
Evans	64
Hoecken	65
6 Modelling Considerations	68
6.1 Units	68
6.2 Exact Nodal Coordinates.....	68
6.3 Motion File.....	69
Format	69
Importing a motion file	70
Exporting a motion file	70
6.4 Multiple Input Motions.....	71
6.5 Linear Actuator.....	71
6.6 Inclined Linear Guiding.....	71
6.7 Building Multi-Node Linkages.....	72
6.8 Modelling a rack/pinion.....	73
6.9 Bearing Forces.....	75
6.10 Using Graphics.....	78
6.11 User Defined Results.....	79
7 Optimization	84
7.1 Introduction.....	84
7.2 Optimization Objective	85
Reference or Target Definition	86
7.3 Parameter Space.....	86
7.4 Constraints.....	88
7.5 Optimization Options.....	89
Settings Evolutionary Algorithm	90
Settings Simplex Algorithm	91
8 Examples	94
8.1 Motion Analysis.....	94
Crank Rocker Mechanism	94
Quick-Return Mechanism	94
Slider Driven Gear Transmission	97
Double Crank Mechanism	98
Planetary Mechanism (Hypo-Cyclic)	98
8.2 Force Analysis.....	100
Transfer Mechanism (incl. gravity effects)	100

Belt drive with external force	102
8.3 Optimization.....	104
Driving Torque Minimization	104
Optimization of trajectory	106
Constrained Optimization	108
9 Elements	118
9.1 Beam.....	118
9.2 Slider.....	121
9.3 Gear.....	123
9.4 Belt.....	127
9.5 Sensor.....	131
9.6 Spring, Damper, Friction.....	132
9.7 Rotational Spring, Damper, Friction.....	137
9.8 Non-linear spring.....	142
10 Integrated Analysis Environment	146
10.1 Keyboard and Mouse	146
10.2 Toolbar.....	147
10.3 Menu Commands.....	149
File Menu	149
Build Menu	150
Graphics Menu	151
Input Menu	152
Input Dialog Box.....	153
Loads Menu	154
Force/Torque Dialog Box.....	155
Analysis Menu	156
Optimization Menu	156
Display Menu	157
Results Menu	157
Window Menu	158
Help Menu	158
11 Literature	160
12 Appendix	162
12.1 Analysis Problems.....	162
12.2 DXF Template File.....	162
12.3 What was new in SAM 6.2 ?.....	163
12.4 What was new in SAM 6.1 ?.....	163
12.5 What was new in SAM 6.0 ?.....	163
12.6 What was new in SAM 5.1 ?.....	165
12.7 What was new in SAM 5.0 ?.....	165
12.8 What was new in SAM 4.2 ?.....	166

12.9	What was new in SAM 4.1 ?.....	167
12.10	What was new in SAM 4.0 ?.....	167
13	Glossary of Terms	170
	Index	171

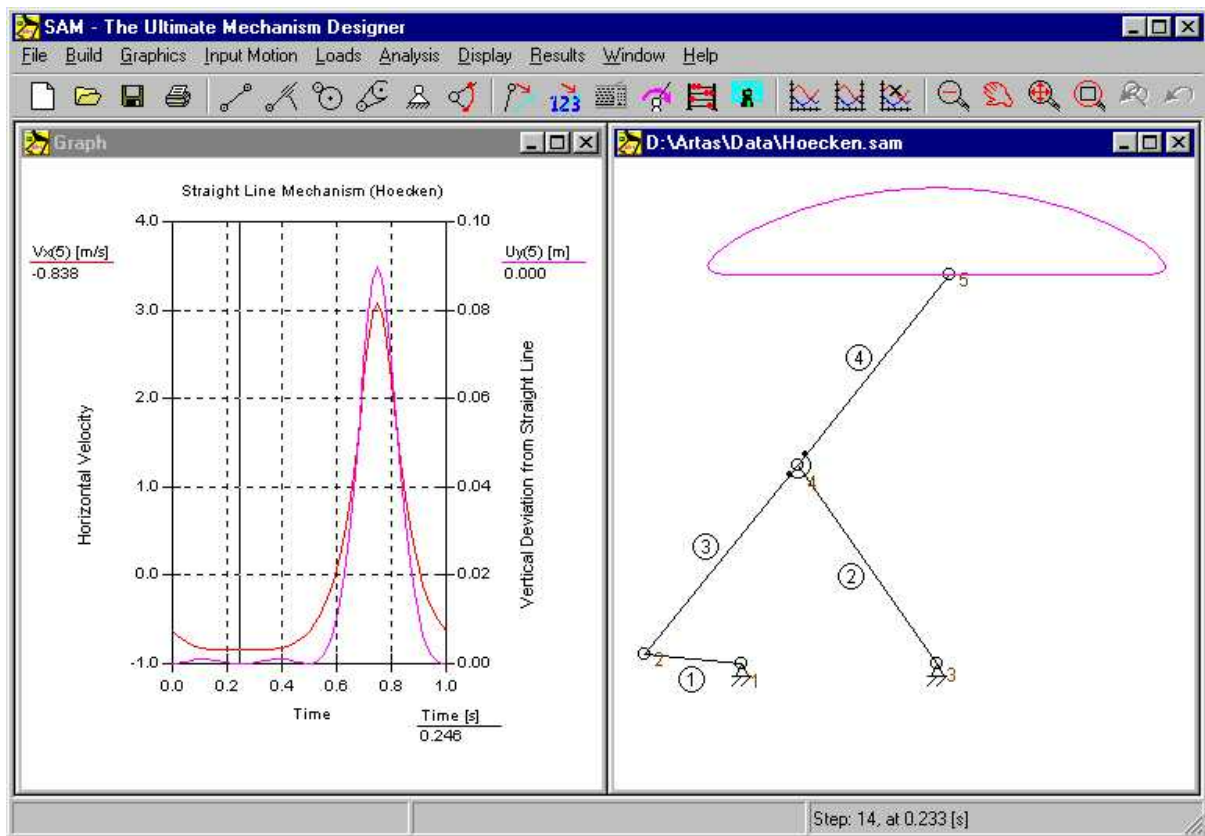
Chapter

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, springs, dampers and friction elements. SAM integrates pre-processing, numerical analysis and post processing, such as animation and xy-plots, in an easy to-use environment offering pull-down menus, mouse support and help facilities.



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's 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, ...

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: Getting Started 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: Optimization explains the various optimization possibilities.

Chapter 8: Examples presents picture stories of various mechanism projects.

Chapter 9: Elements tells you all about the various elements that can be used.

Chapter 10: Integrated Environment discusses in more detail the use of mouse and keyboard, the toolbar and the various menu items.

Chapter 11: Appendix

Chapter 12: Glossary of Terms

Chapter 13: Index

1.4 Purchase Agreement

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

ARTAS authorizes you to make archival copies of this software for the sole purpose of back-up and protecting your investment from loss. Under no circumstances may you copy this software or documentation for the purpose of distribution to others. Under no condition may you remove the copyright notices made part of the software or documentation.

This software may be used by one person on as many computer systems as the person wants. We expect that group projects making use of this software will purchase a copy of the software and documentation for each member of the group. Contact us for volume discounts and site licensing agreement.

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

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

Chapter



2

2 Overview

2.1 The Design Process

The process of designing mechanisms, basically can be divided into two distinct phases, namely

- Synthesis
- Analysis

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 - there are also special programs for certain types of mechanisms (slider-crank, 4-bar mechanism ...) - 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.

2.2 Capabilities

2.2.1 Design

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

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

In case these design wizards do not provide the solution to the specific design problem, the user has to rely on his 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
- belt, gear
- sensor (both translational and rotational)
- spring, damper and friction element (both translational and rotational)
- non-linear spring

which 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 multiple 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, such as

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

are available and can be combined to form any desired input diagram. Inputs can also be read from an external ASCII file or defined via a table to enable the definition of arbitrary motions. This latter feature is especially handy for the modeling of non-standard cam profiles.

2.2.4 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.

2.2.5 Analysis Results

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

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

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

- driving torque (force)

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

2.2.6 Post-Processing

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

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

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

2.2.7 Optimization

Included in PROFESSIONAL version of SAM only !

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

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

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

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

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

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

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

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

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

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

2.3 History, Present and Future

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

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

1989 : SAM 0.1 First DOS version

1991 : SAM 1.0 First official release, kinematics only

1991 : SAM 1.1 Printer/plotter drivers

1992 : SAM 2.0 Kinetostatics of linkage mechanisms

1993 : SAM 2.1 CAD import added

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

1995 : SAM 3.0 First Windows version

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

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

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

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

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

2007 : SAM 6.0 Optimization, Context sensitive right mouse button

2010 : SAM 6.1 Clamped cubic spline, extended automated optimization, new display features,

automated updates, videos in GIF format, ...

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

Due to its capabilities, ease of use and modest system requirements more than 100 polytechnics and universities have already adopted SAM for their mechanism courses and practical exercises. Furthermore SAM is being used by a significant number of companies, among which renowned international companies active in the field of automobiles and packaging machines.

Future plans for SAM include :

- ·Extension toward modelling of to model rack/pinion and cam mechanisms
- ·Synthesis of cam mechanisms
- ·Dynamics module next to the existing kinematics and kinetostatics
- ·Link to Matlab/Simulink

2.4 European Academic Software Award

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

Chapter

3

3 System Requirements & Installation

3.1 System Requirements

SAM requires the following hardware and system software :

- MS Windows Vista, Windows 7, Windows 8 or higher
- At least 50Mb harddisk space (incl. manual).

3.2 Installing SAM

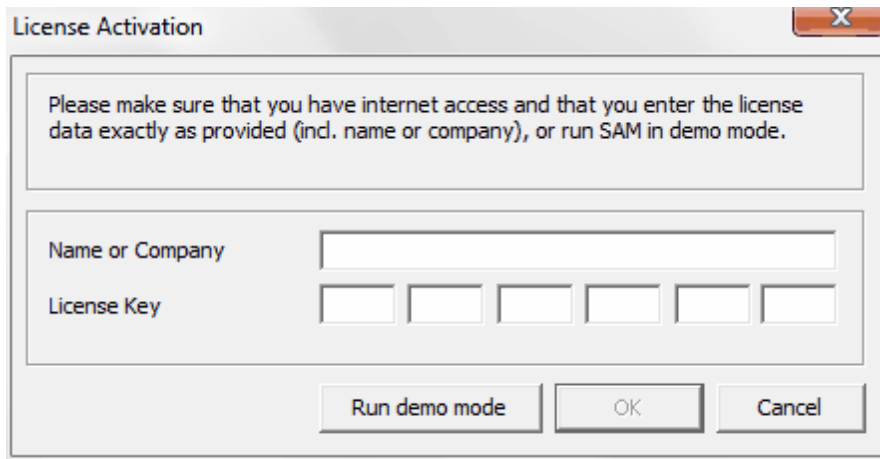
SAM comes in a compressed format. The installation program will expand the files and install each at the proper location on the hard disk.

- Run "SAM70_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.

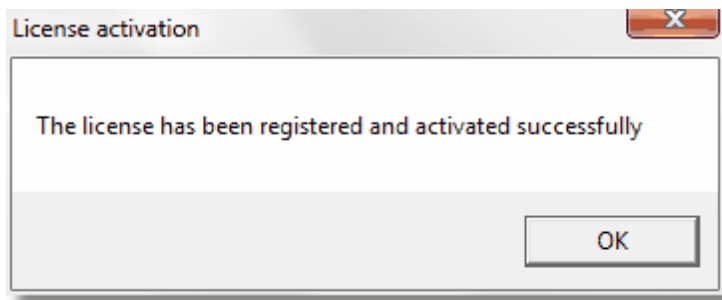
Node-Locked License

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.

**Floating License**

Please contact your IT-manager.

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

The project file format of SAM 7.0 differs from that of earlier versions. An old project file can generally be loaded in SAM 7.0. The first time this project is saved in SAM 7.0 the user is asked to confirm that the data will be stored in the new format. This process is irreversible, so in case of very important project-files we advise to make back-up copies.

Note : SAM 3.0, 4.x or 5.x, 6.0, 6.1 can not open 7.0 project files.

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 Animation from the Display menu or click the Windmill icon in the toolbar to start the animated display.

3. View analysis results 

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

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

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

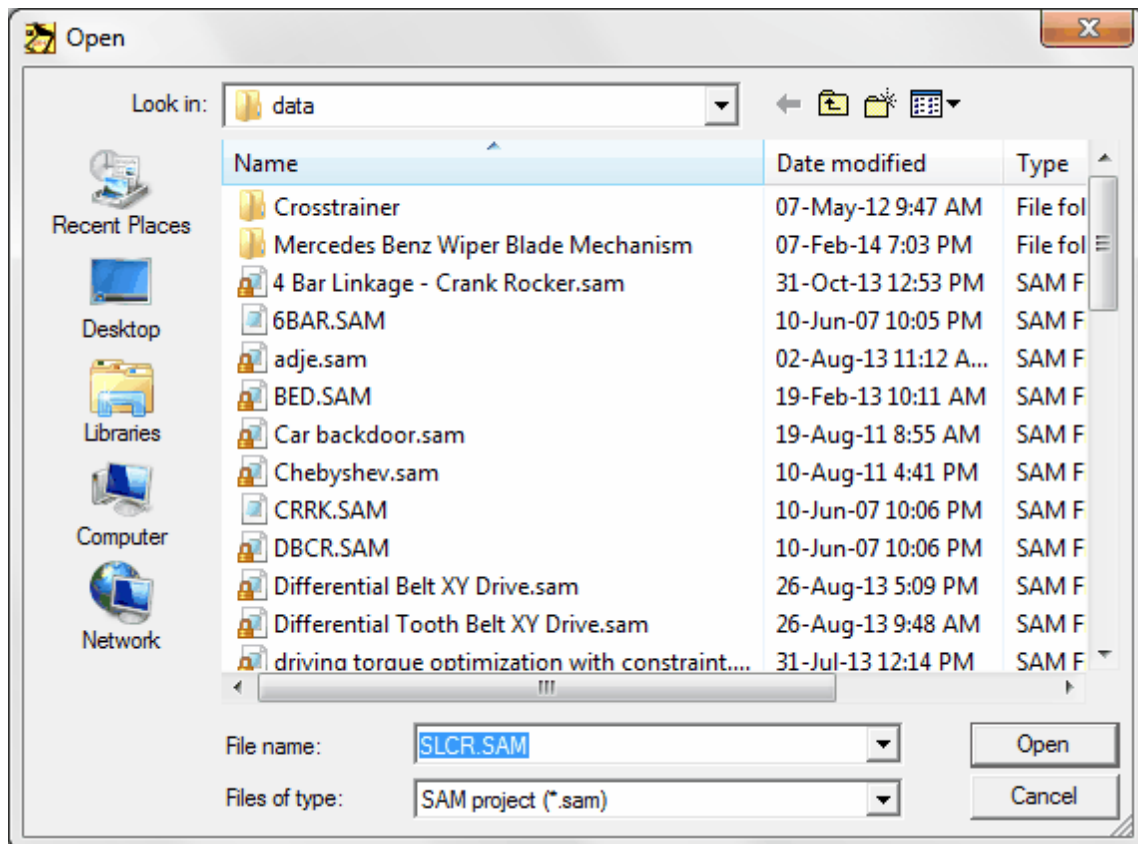
4.1.1 A quick example

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

Motion Analysis

Choose Open from the file menu or click 

The Open dialog will appear.

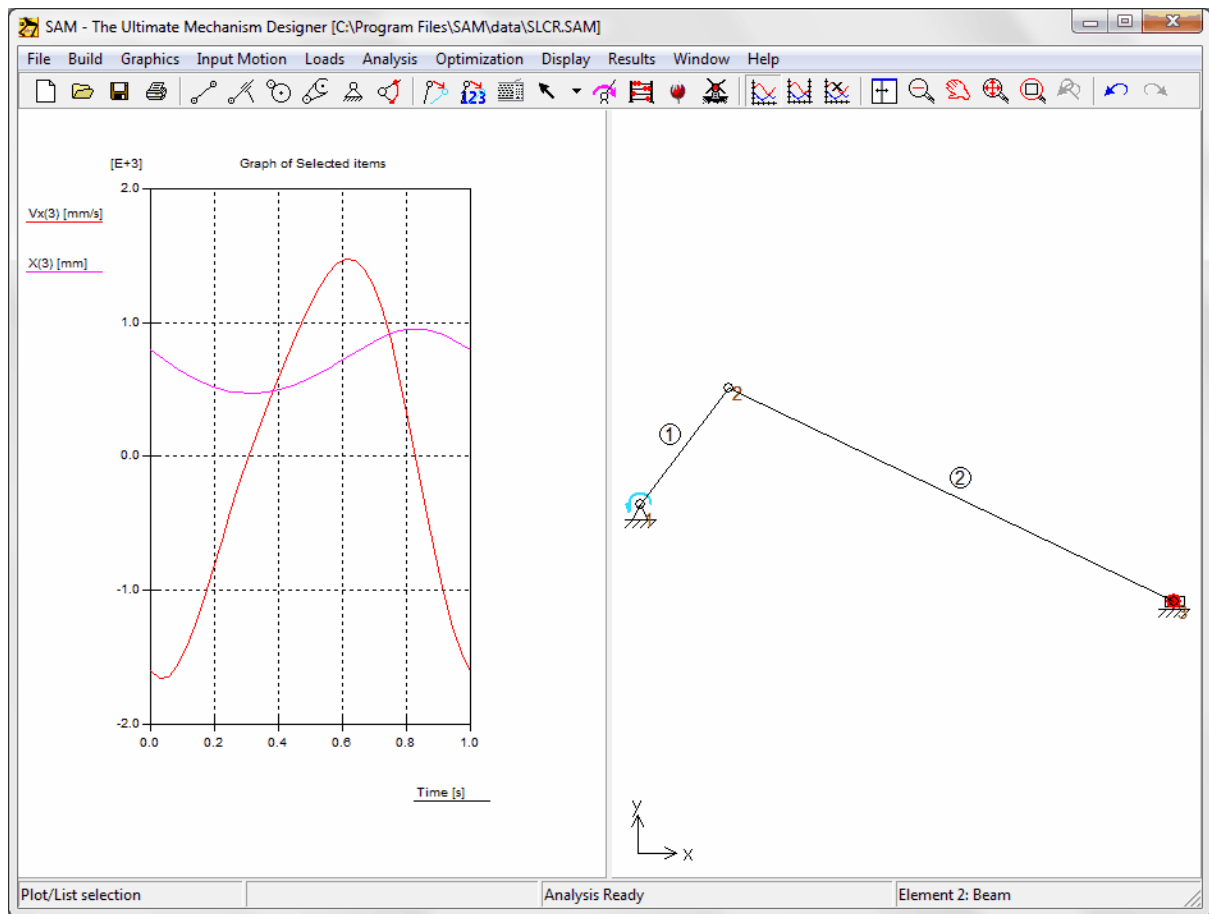


Open dialog box

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

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

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



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

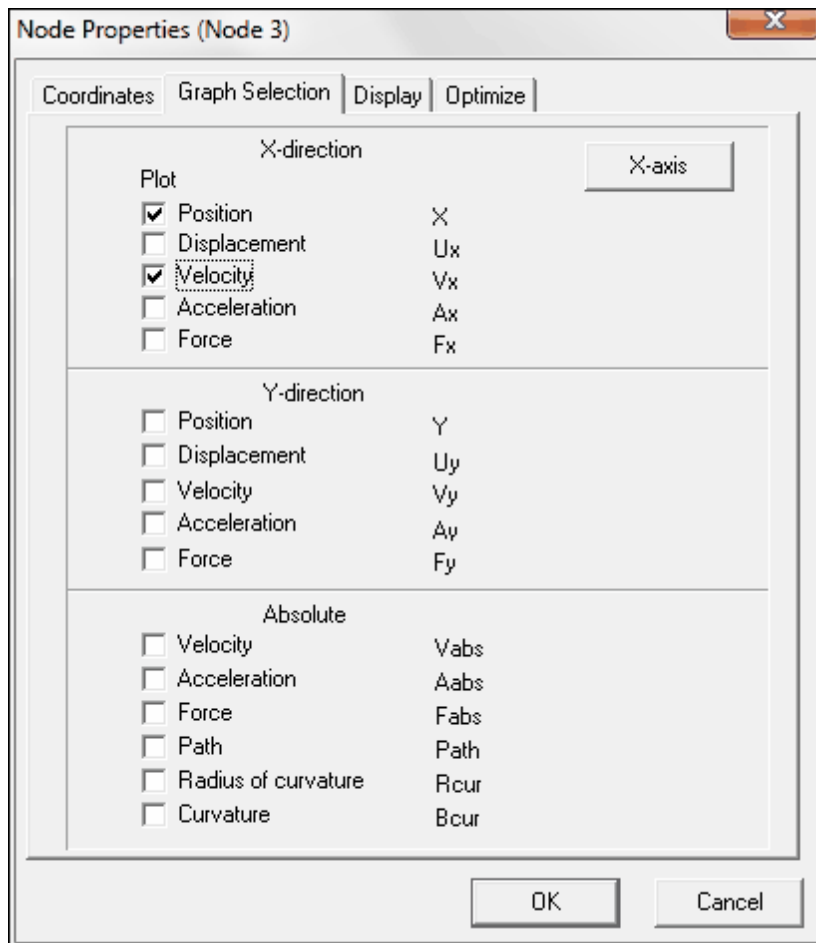
In the right window the mechanism is shown (including a mass of 25kg in node 3), whereas the left window shows the position and velocity of node 3 in x-direction.

Select Animation in the Display menu or click 

You will now see an animated display of the mechanism.

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

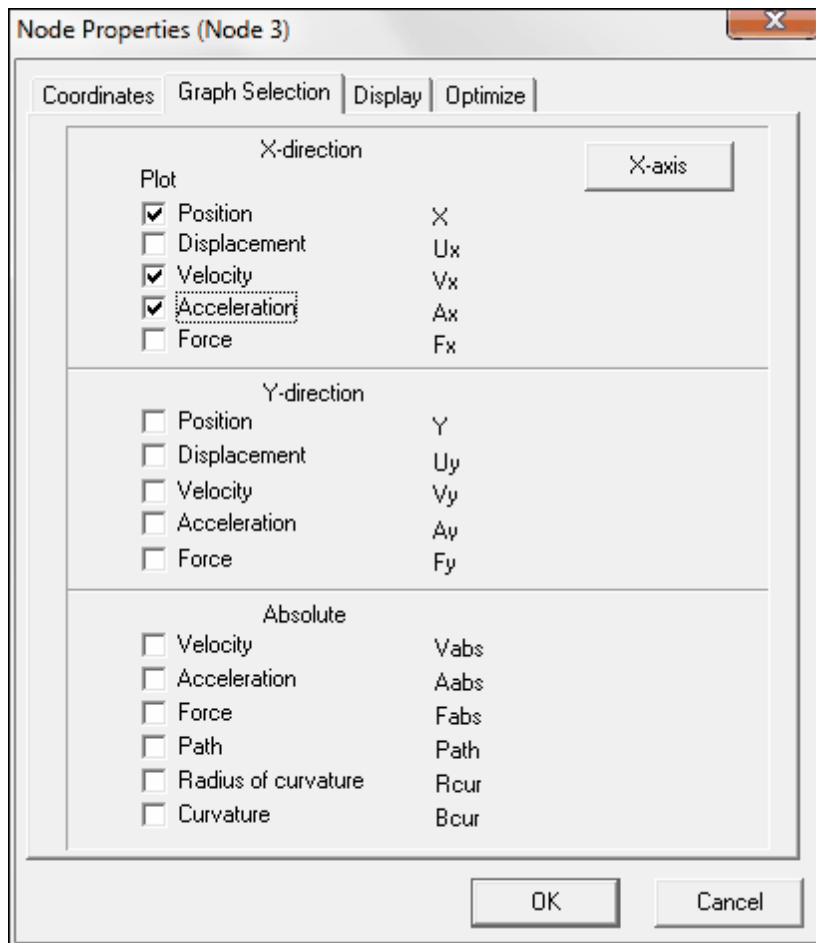
A dialog box will appear that shows all properties of that node. The checked items are currently plotted in the graph.



Node properties dialog box

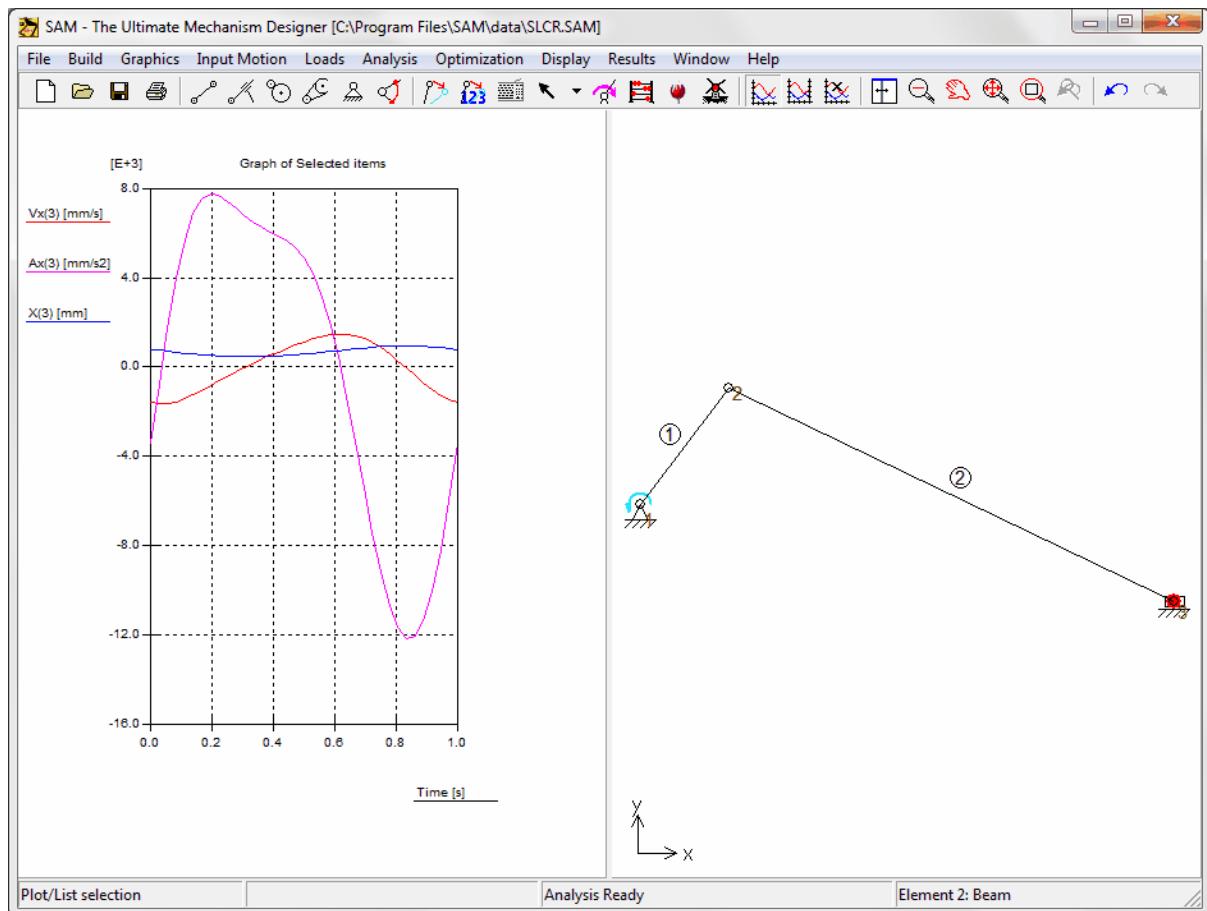
As you can see most items are blanked, meaning that these are not plotted (and also not calculated). If you want more results, you have to select that item as well and press OK.

Check the Acceleration Ax and press OK.



Node properties dialog box

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

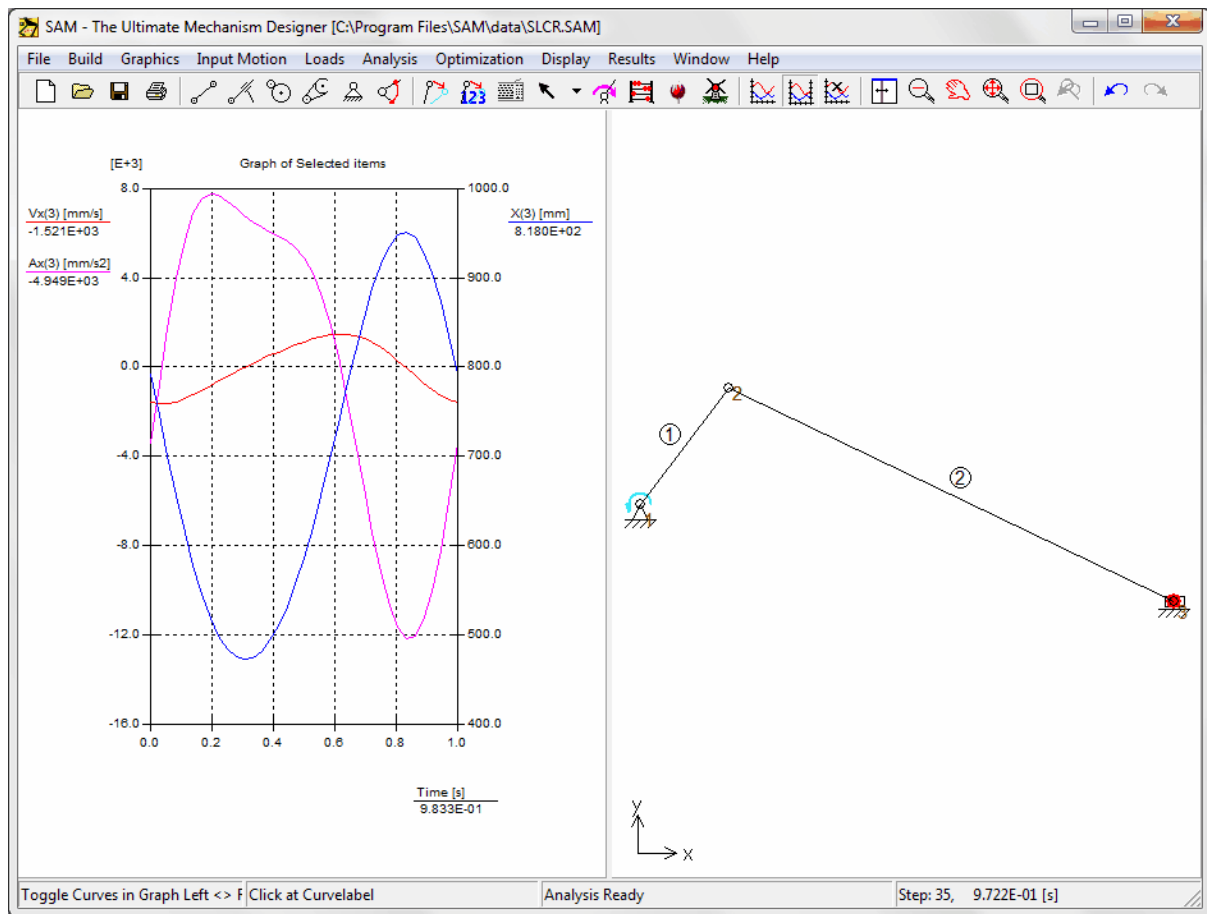


a. X-position, X-velocity and X-acceleration of slider as function of time. Mechanism

Choose "Toggle Curve" from Result menu or click 

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

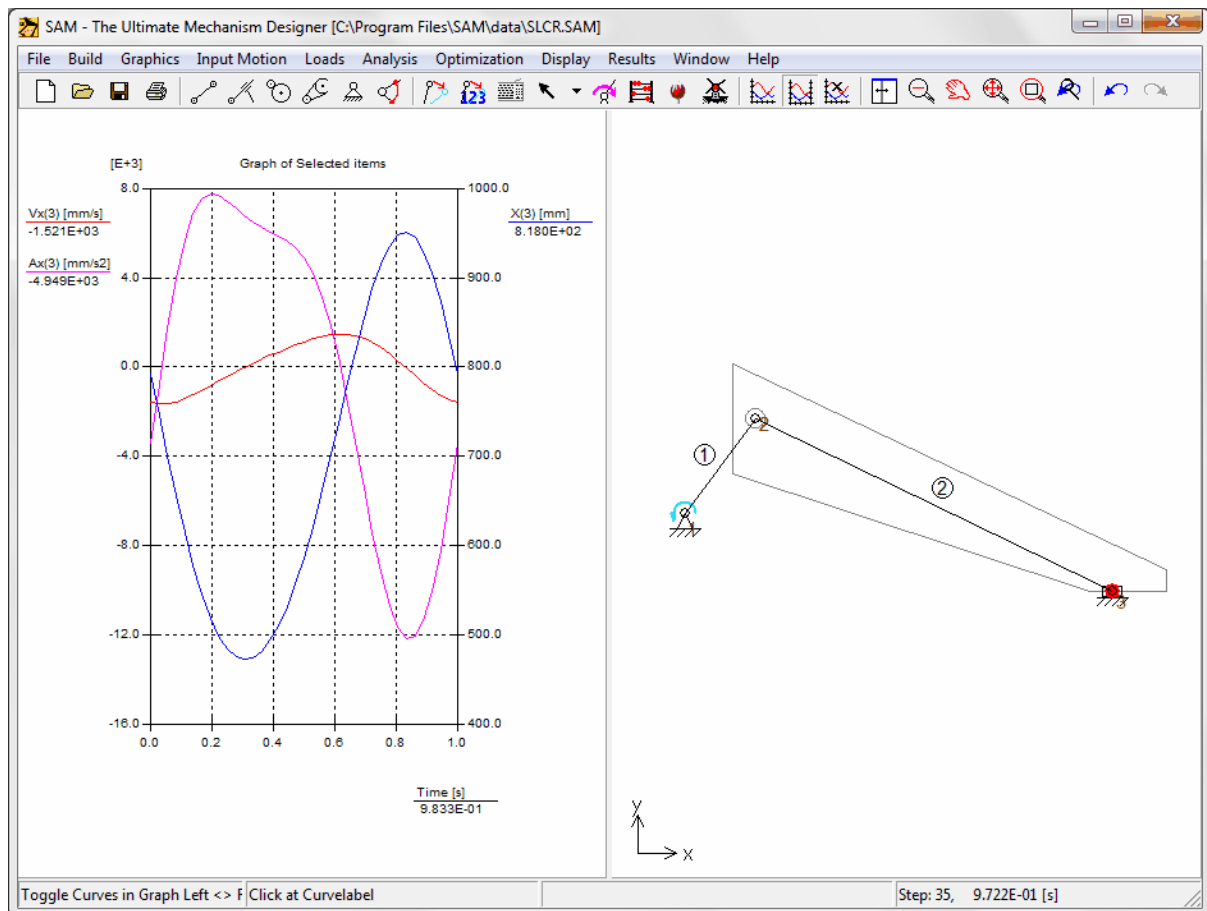
The X(3)-label has moved to the right side and the X-displacement of node 3 X(3) is scaled according to the right y-axis scaling (this double y-scaling mechanism is especially useful in case the range of the displayed variables differs significantly).



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

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

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



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

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

First you have to select the group of graphical items that you want to attach (click the DXF-data that you just imported), then you have to click on the element to which you want to attach that group (click element No.2).

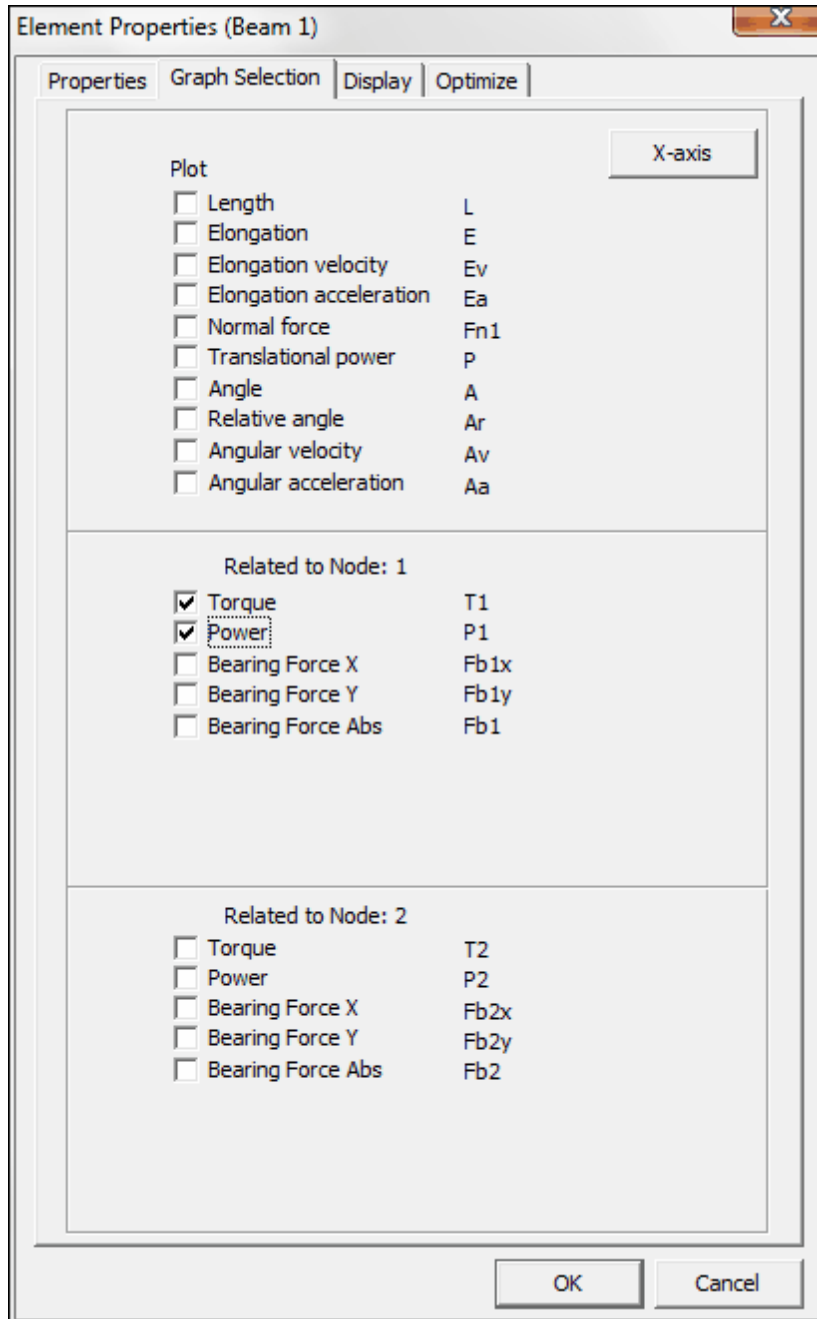
Select Animation in the Display menu or click 

You will now see an animation in which the graphical items move together with element No.2.

Force Analysis

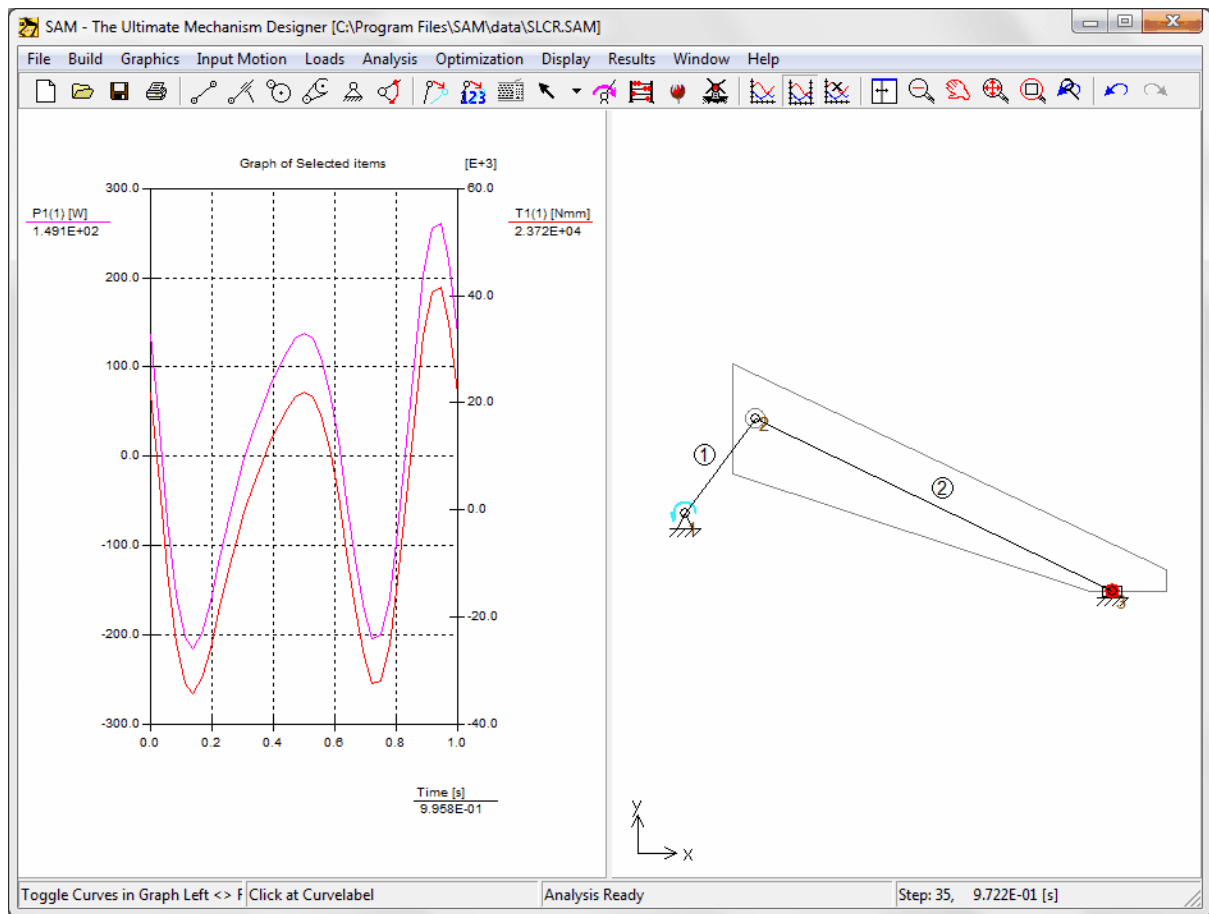
If you want you can remove the DXF graphics since it is not essential for the force analysis. First we will clear all results and then look at the required driving torque and the required driving power. Then, we will look at the bearing forces in point 1 and point 2.

Results>Clear Selection, then select element 1 and activate the items shown below



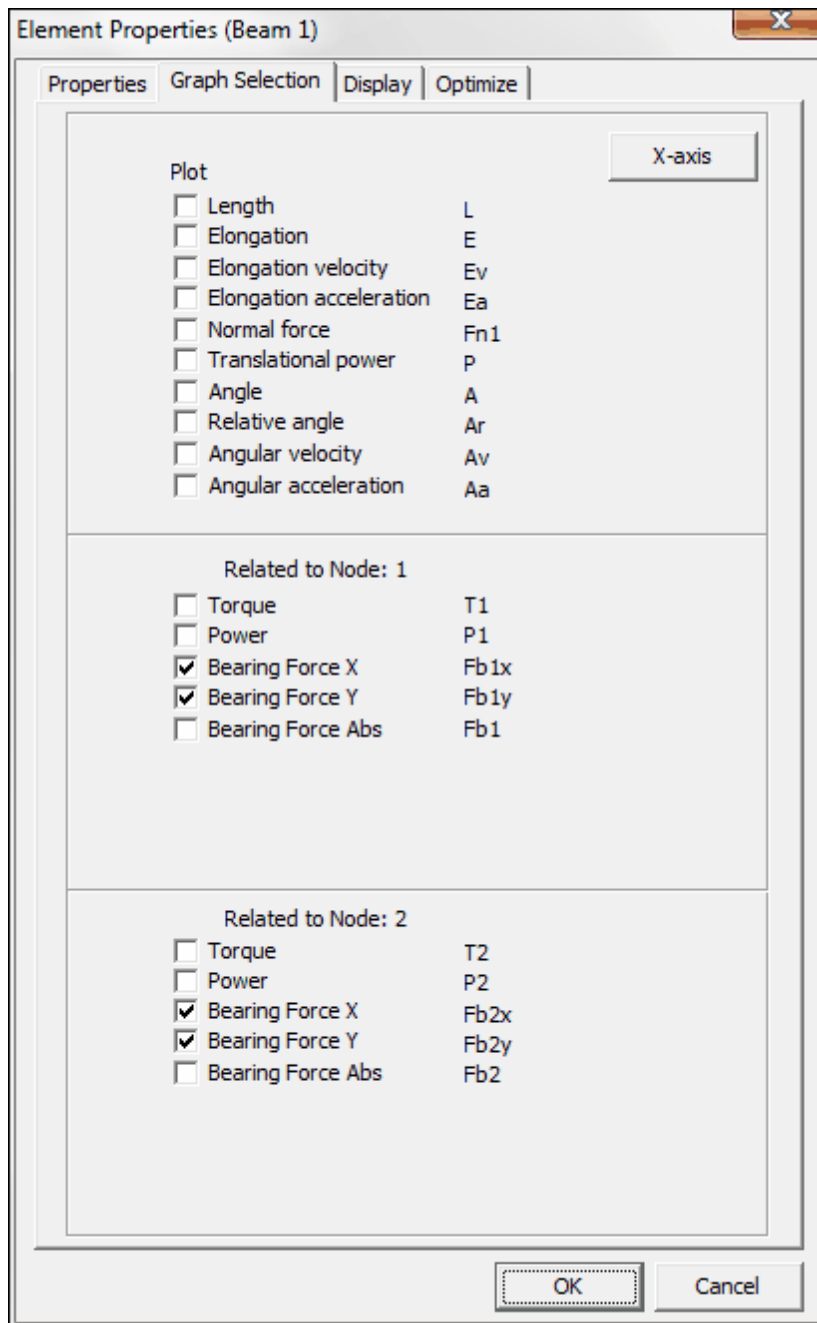
Element properties dialog box

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

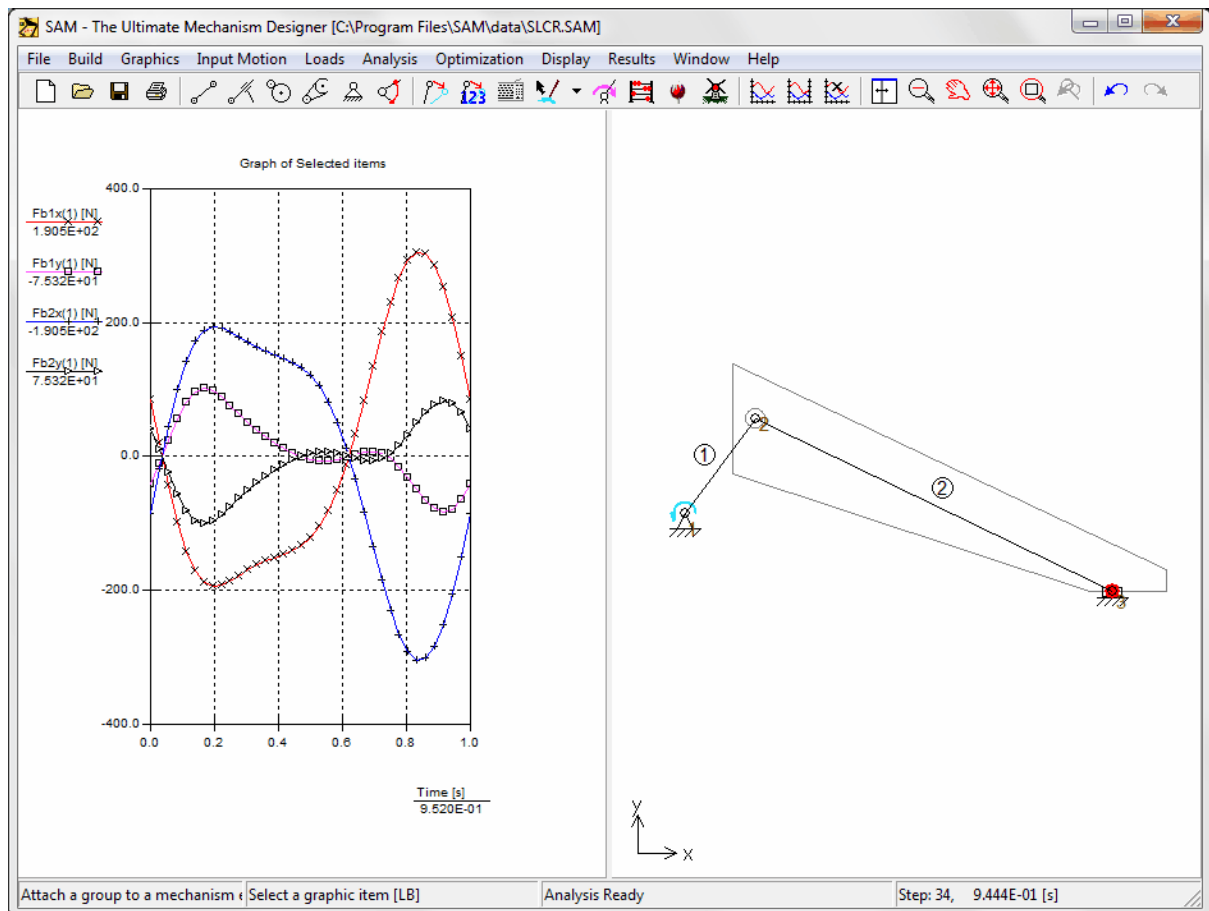


Screen shot showing the mechanism plus the driving torque and driving power

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



Element properties dialog box



Screen shot showing the mechanism plus the x/y bearing forces of element 1 in nodes 1 and 2

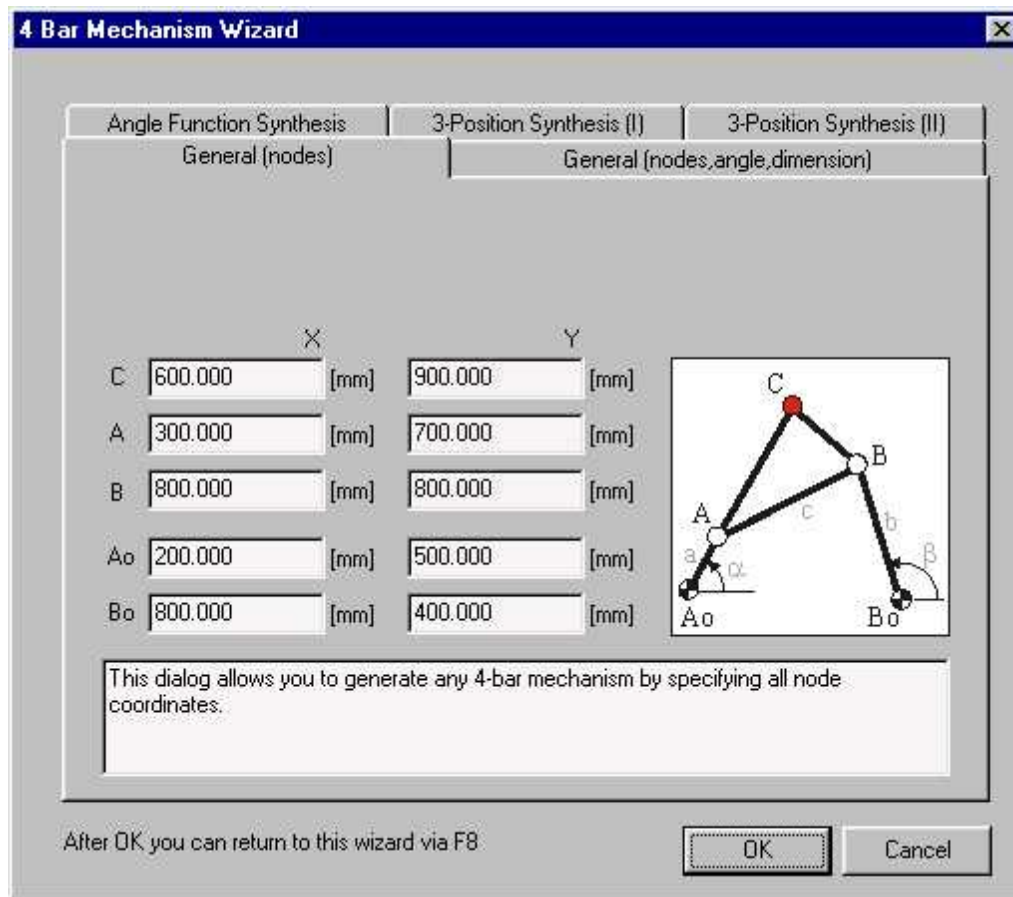
This is the end of our first quick tour based on an existing mechanism project file. In the next chapters you will learn how to design a mechanism based on the Design Wizard and how to "manually" build your own mechanisms.

4.2 Using the Design Wizard to create a mechanism

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

Select File/Wizard/4-Bar Mechanism

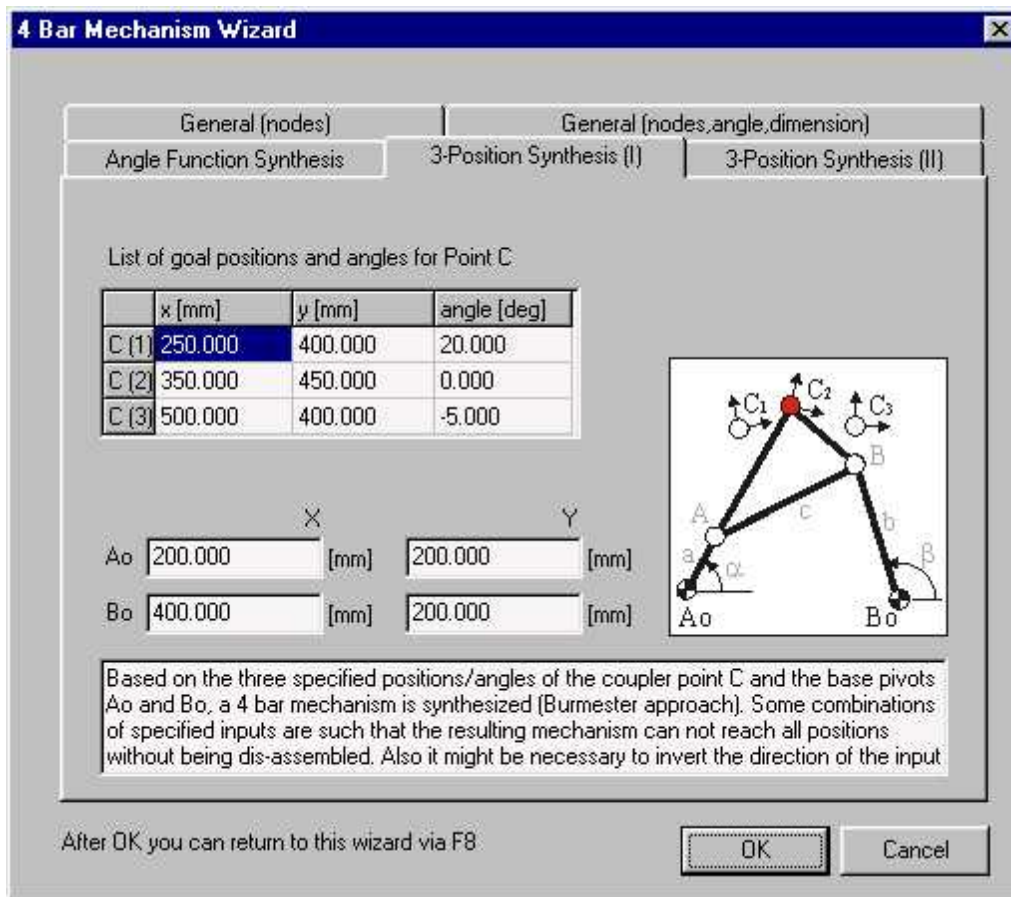
A dialog box will appear with 5 tabs related to specific design tasks that can be fulfilled by a 4-bar mechanism.



Design Wizard "4-Bar Mechanism"

Choose "3-Position Synthesis (I)"

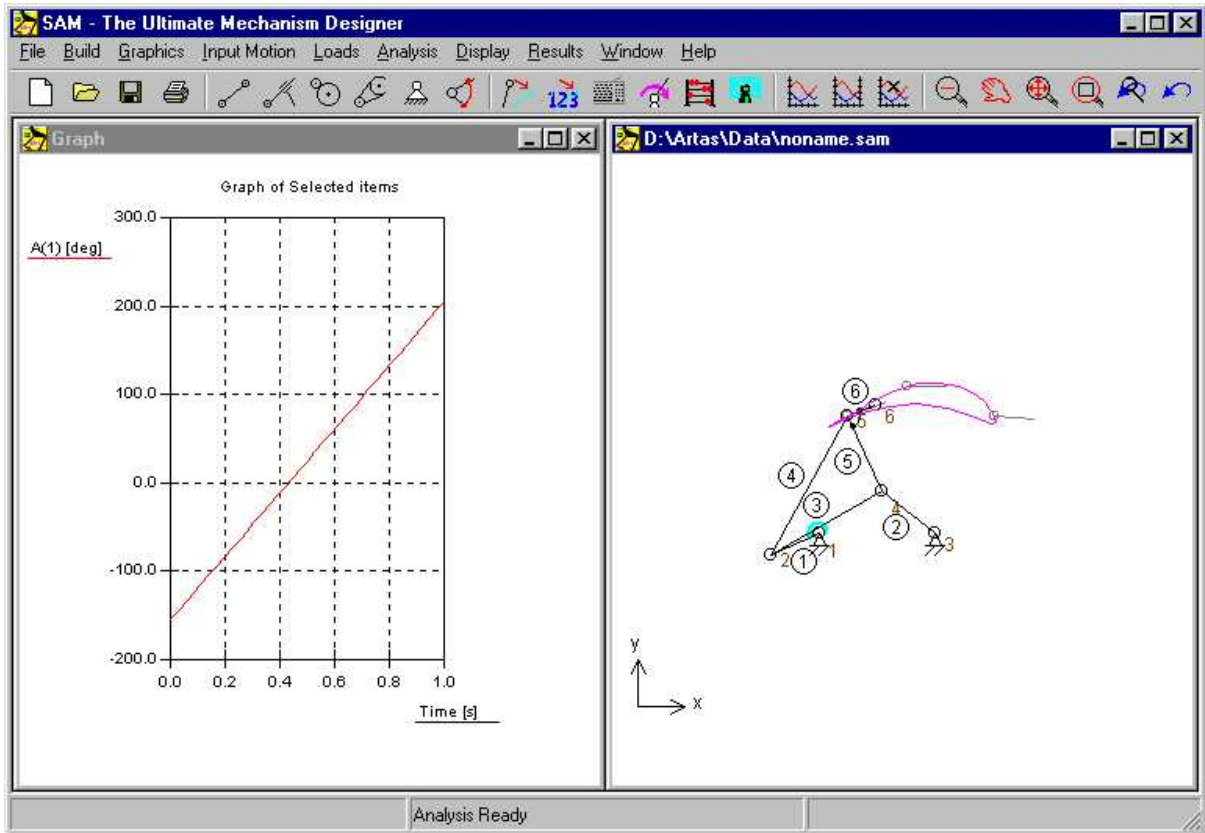
This brings the "3-Position Synthesis (I)" tab to the front position. In this dialog you can specify 3 positions/angles of a the coupler plane plus the location of the fixed nodes Ao and Bo. As a first introduction we suggest to accept the default values.



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

Press OK to accept the current values

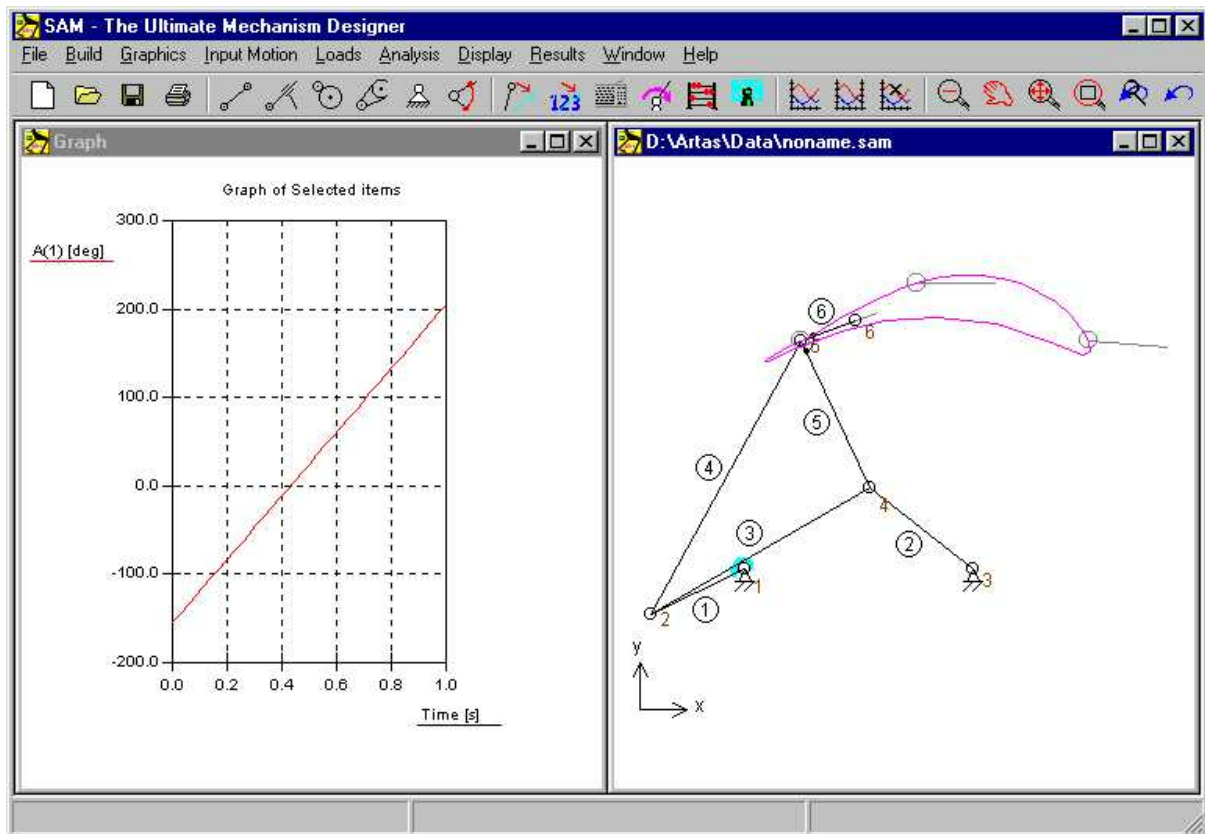
Based on the three specified positions/angles of the coupler point C and the base pivots A0 and B0, a 4-bar mechanism is synthesized, analyzed (based on a certain input motion) and the path of the coupler point (+ the 3 specified goal positions/angles) is displayed in the mechanism window. Initially, 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  Animation

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

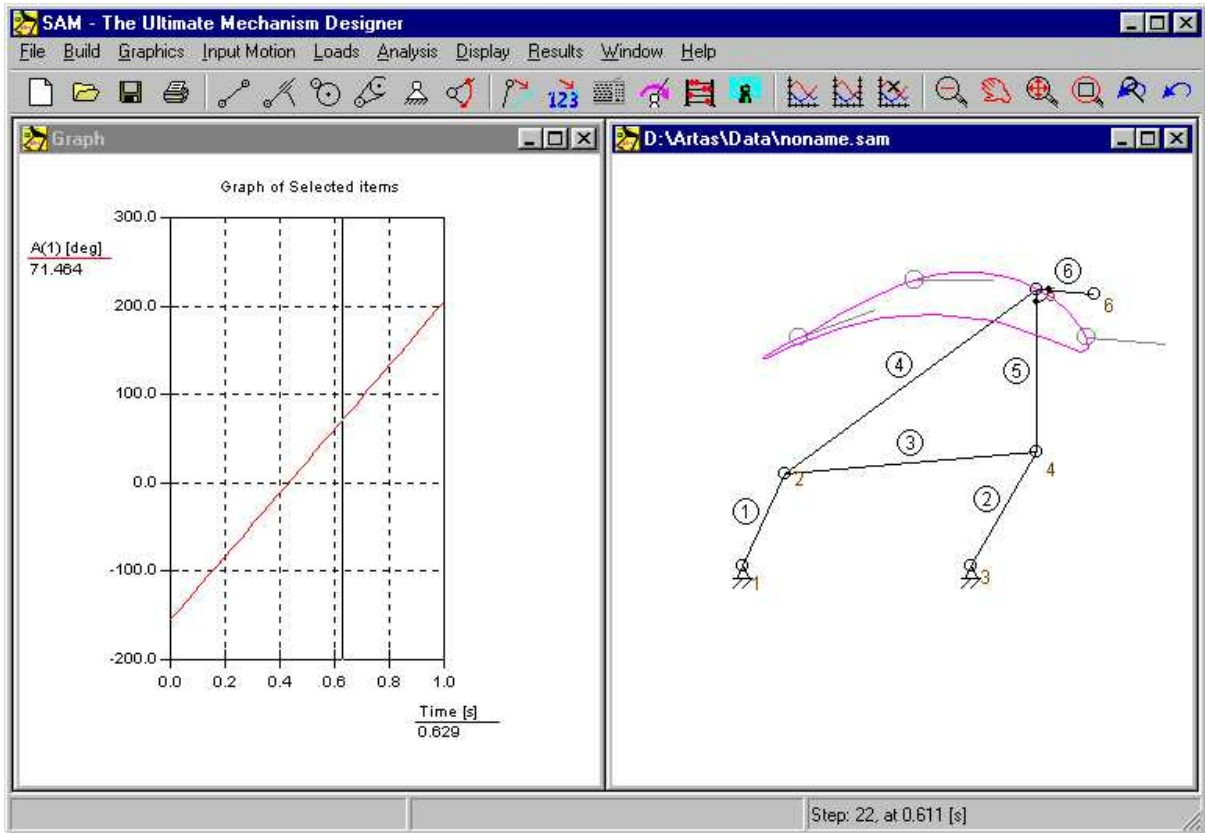


Result of Design Wizard "4-Bar Mechanism - 3-Position Synthesis (I)"

Press  again to stop the animation and move mouse over graph windows

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

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



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


Press <F8> to return to the Design Wizard.

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



4.3 Building a new mechanism (without Design Wizard)

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

1. Create new project

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

2. Assemble mechanism from basic elements ...

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

3. Define support conditions

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


4. Define inertia's, external forces, gravity

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

5. Specify the input motion(s)

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

6. Analysis

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

7. Animate Mechanism

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

8. View Analysis Results

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

9. Save Project

Choose Save from the File menu to save the project.

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

Select New from the File menu

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



Work Space dialog box

Click OK to accept the default values of the workspace.

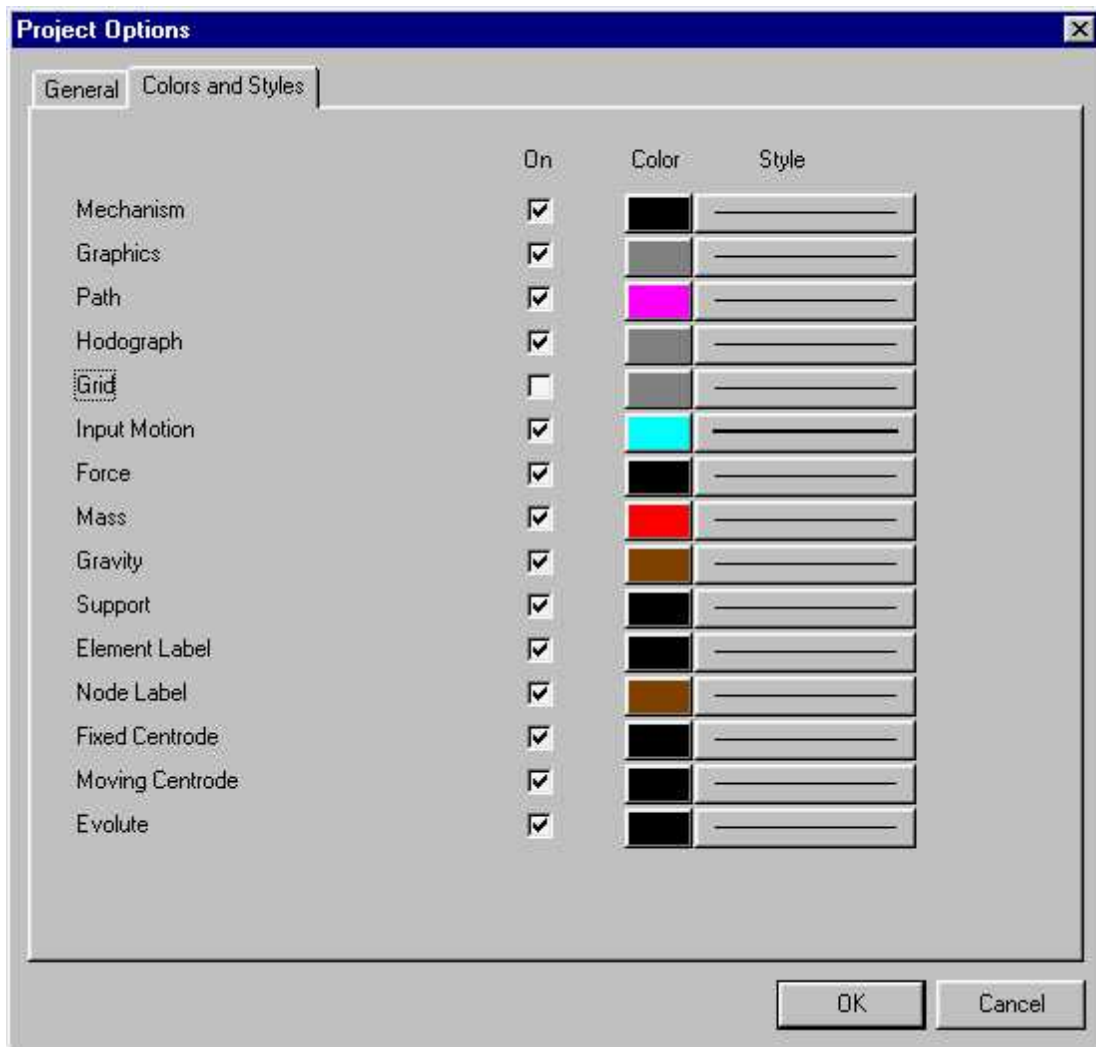
You will now see an empty workspace with a graph-window and a mechanism window.

Maximize the mechanism-window by pressing the corresponding button.

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

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

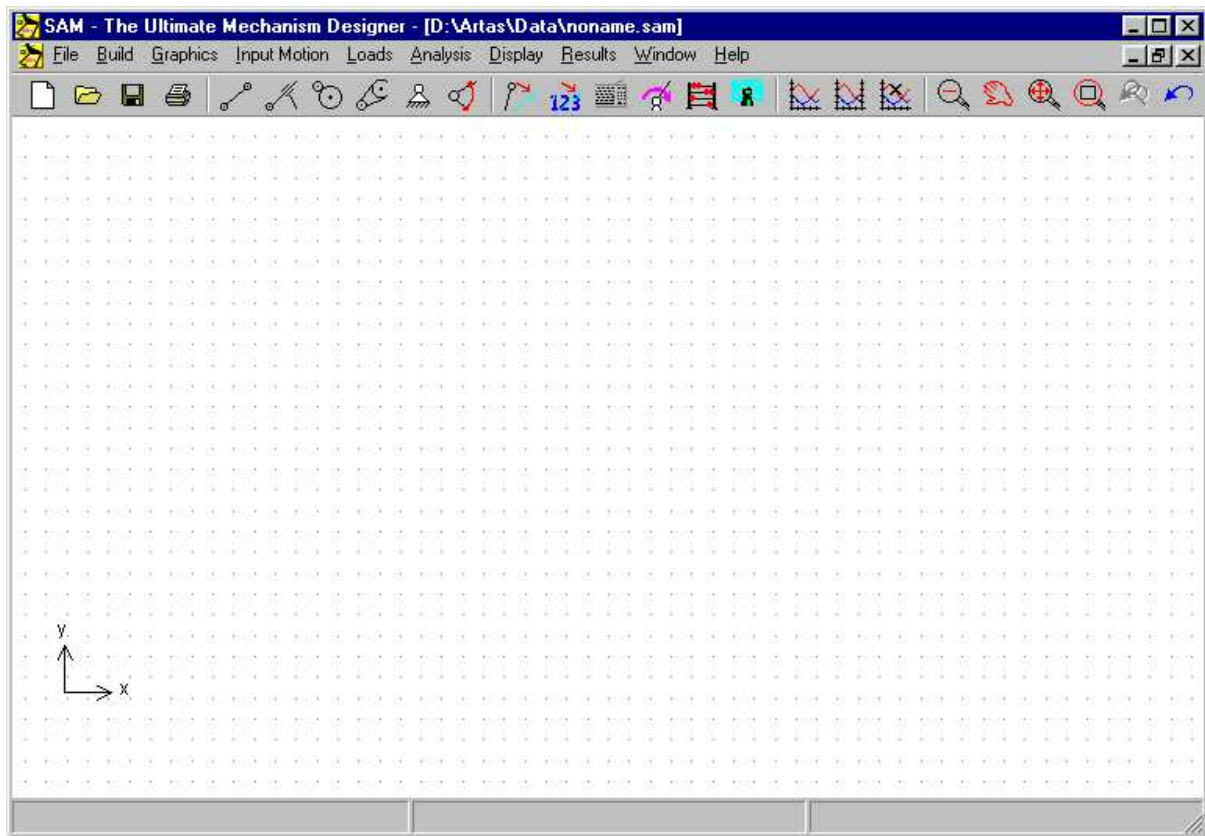
In the dialog box that will appear you can customize the display of various items. As you can see the Grid is switched off.



Display Options dialog box

Switch the display of the Grid ON and click the OK button

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



Workspace with default grid

Select Beam from the Build menu or click 

You can now start creating a beam element.

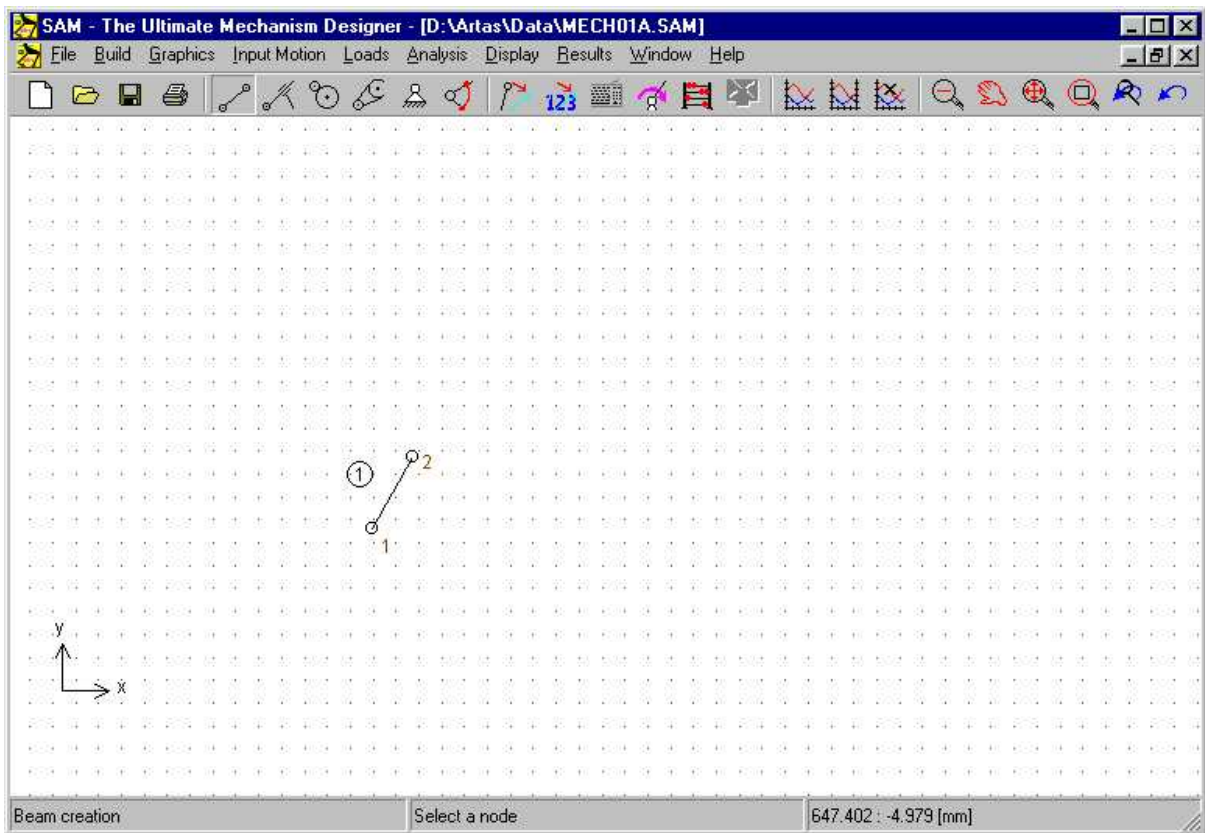
Move the mouse to the starting point of the element and press the left button to create the first node.

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

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

While moving the mouse the current shape of the element will be displayed. If you want to cancel the creation process halfway (first node is already chosen but not the second node) press the right mouse button Once an element has been created and you want to delete it again, you can select Delete from the Build menu and click the element that you want to delete.

Your screen should now look similar to this :

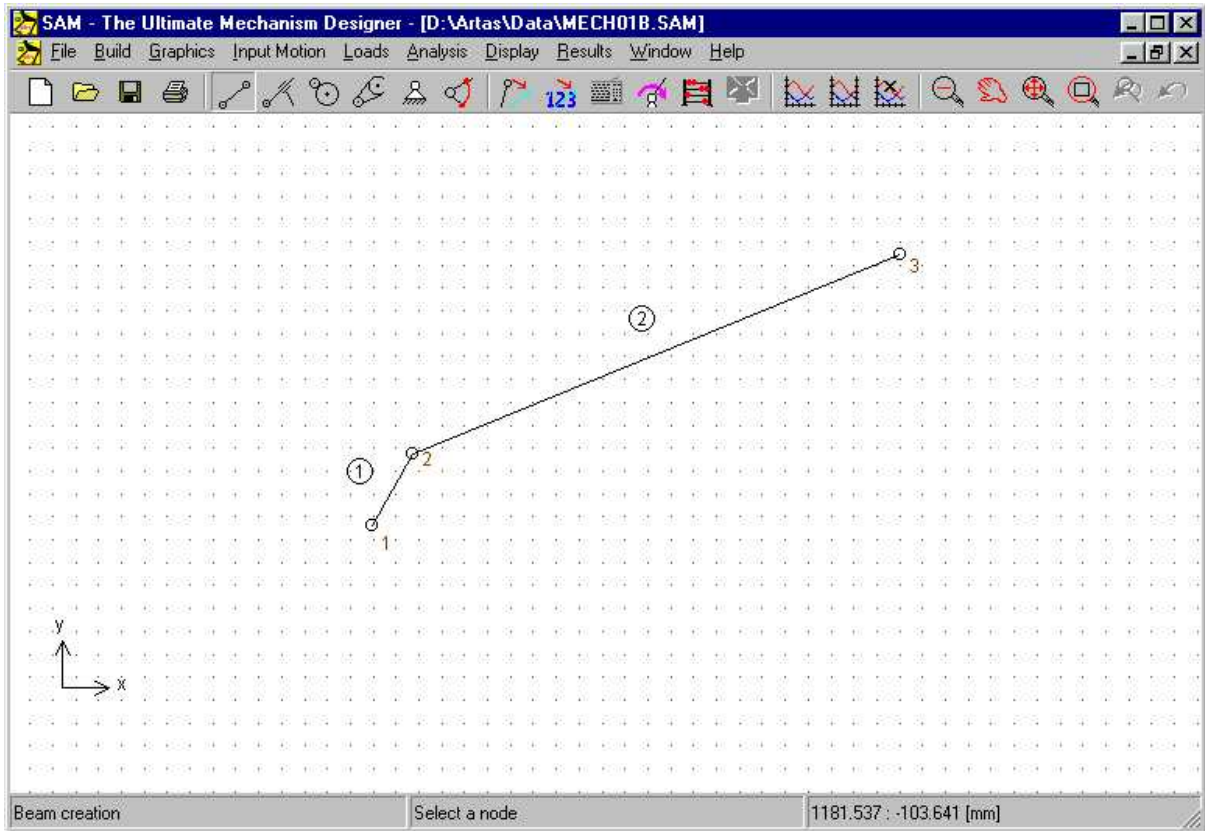


Workspace after creation of first element.

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

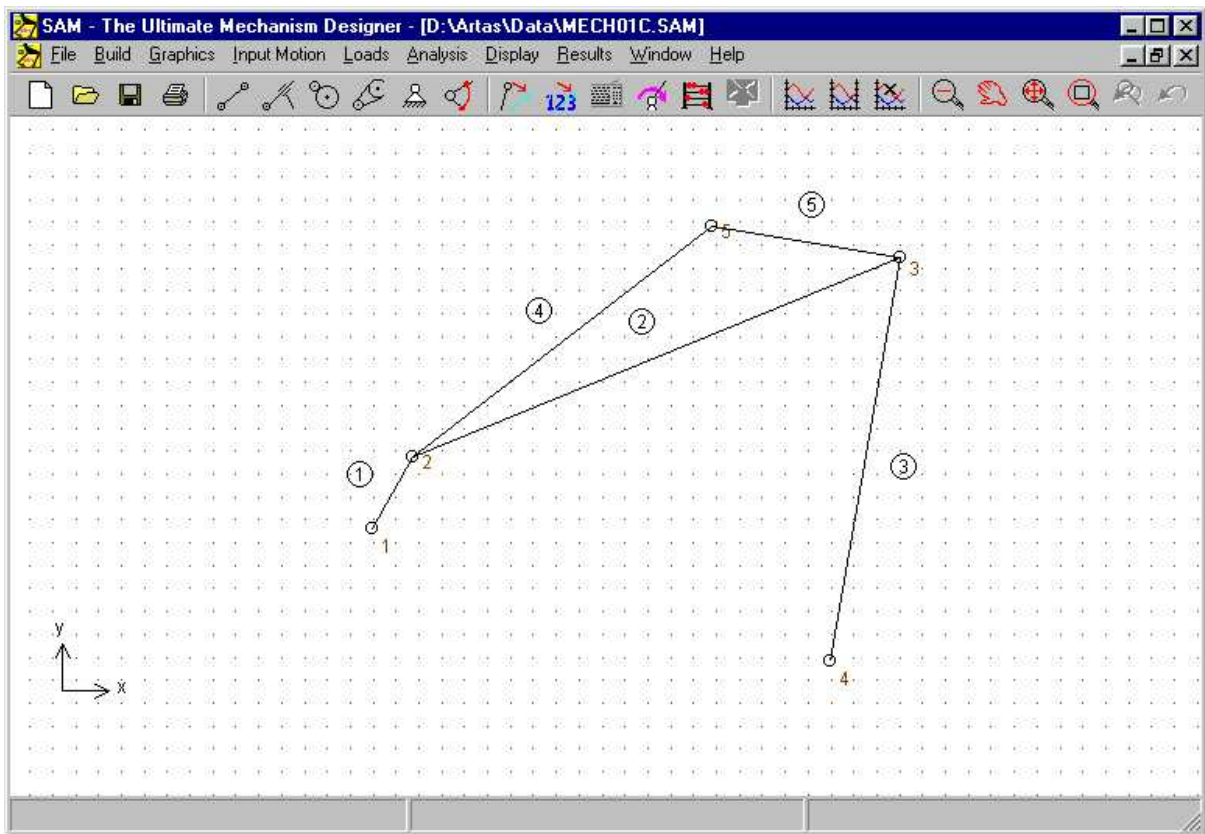
The second linkage is created in the same way, except that one of the nodes of the new element has to coincide with an already existing node of the first element in order to create a connection.

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




Workspace after creation of second linkage

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



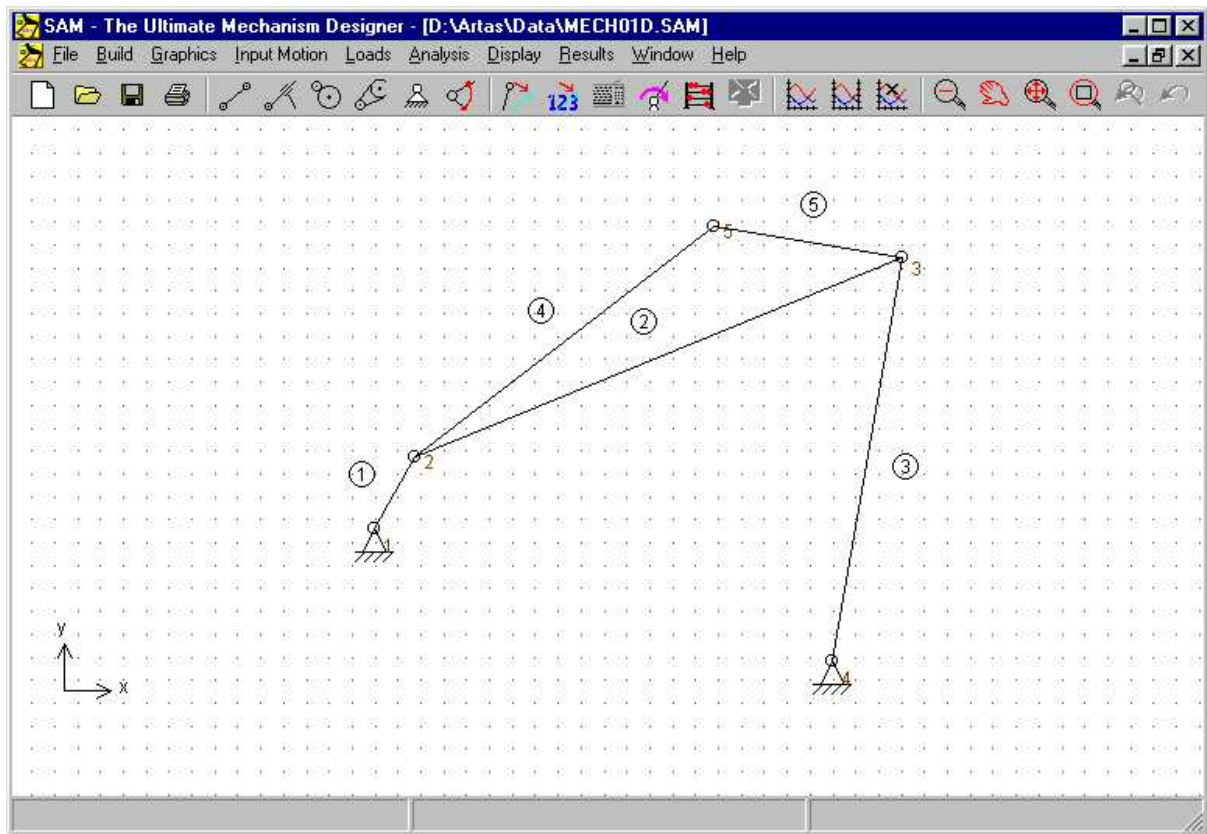
All elements of the 4-bar mechanism have been assembled

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

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

Apply the same support conditions to node 4.

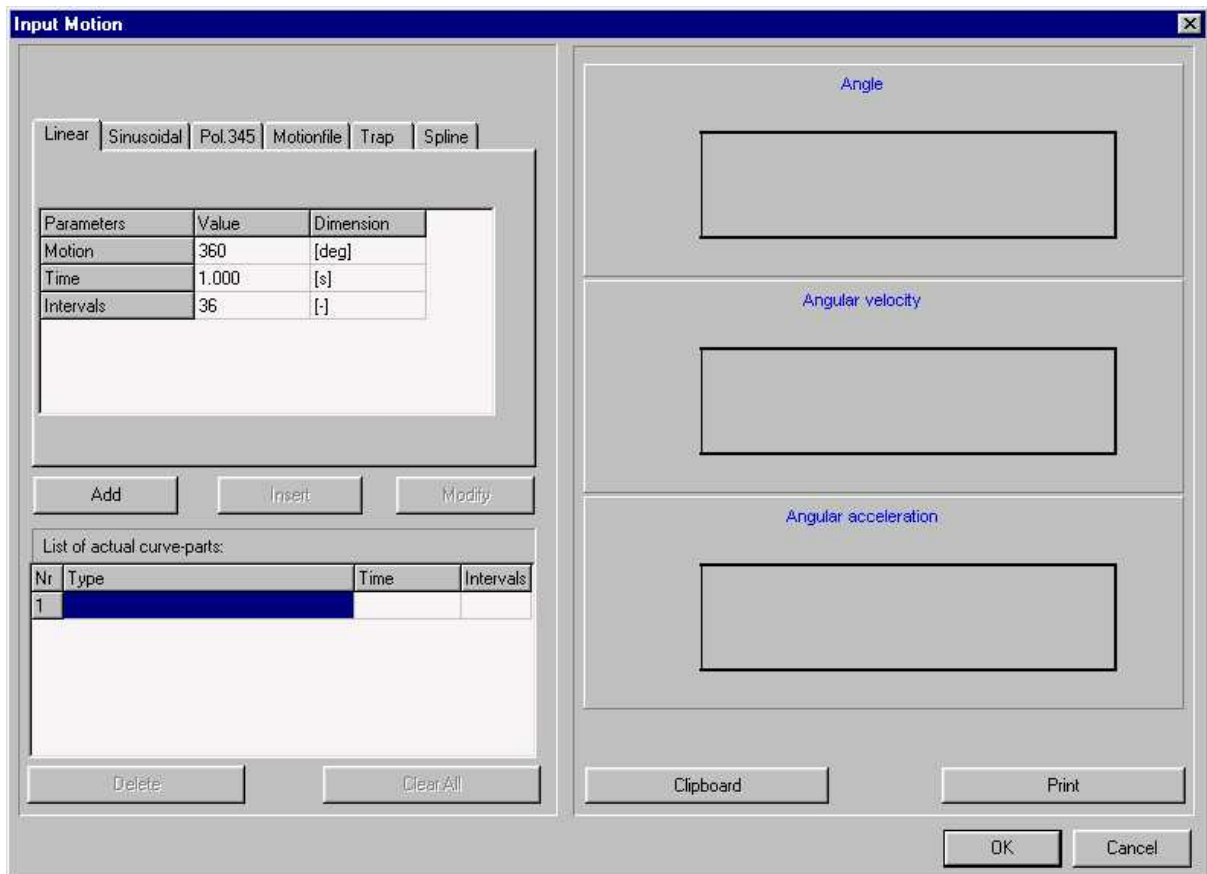
This will result in the following situation on your work space.



4-bar mechanism including support conditions

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

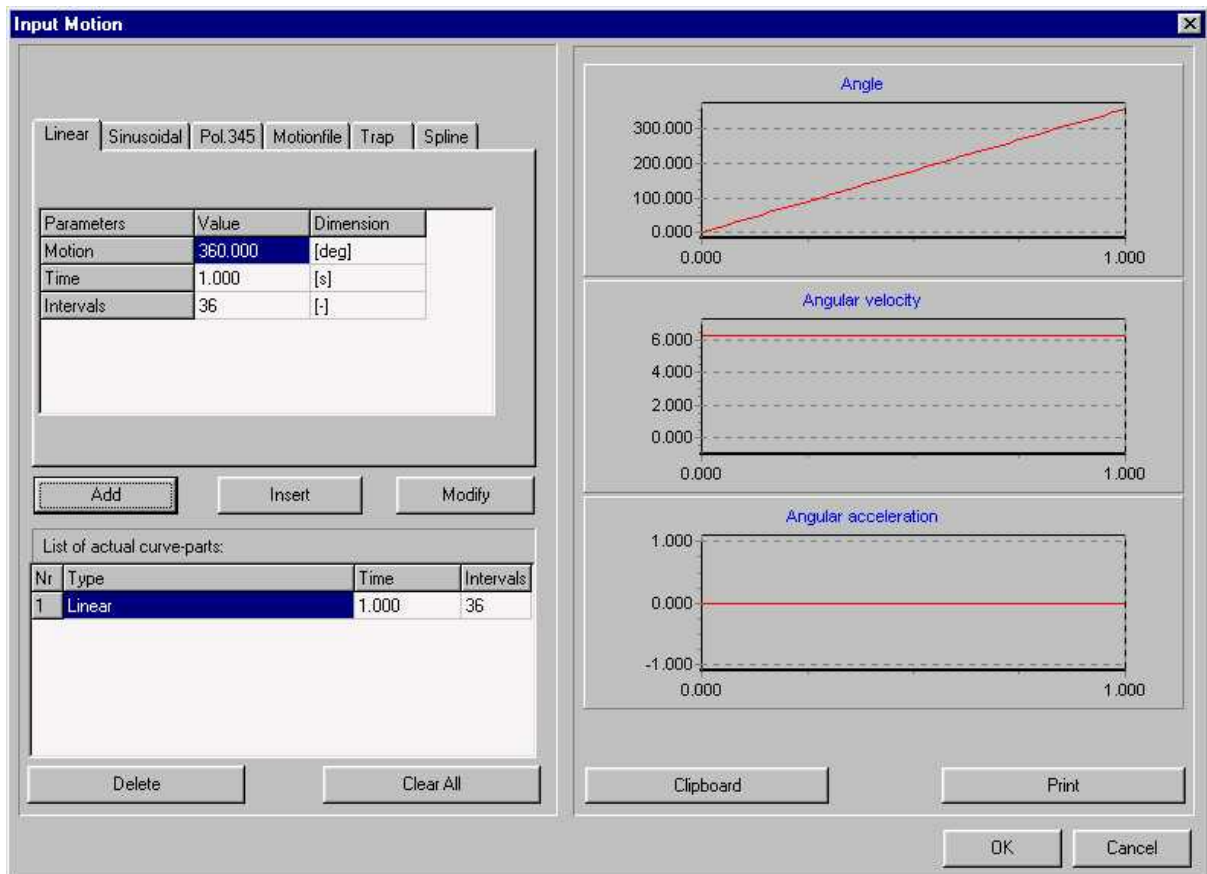
The Input Motion dialog box will appear. The heart of the input dialog box is formed by a list box, which contains a sequential list of selected basic motion blocks (the graph shows the equivalent of these data). Initially, this list is empty.



Input Motion dialog box

Click the "Add" button

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



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

Press OK to accept the current input motion.

Select Animation in the Display menu or click 

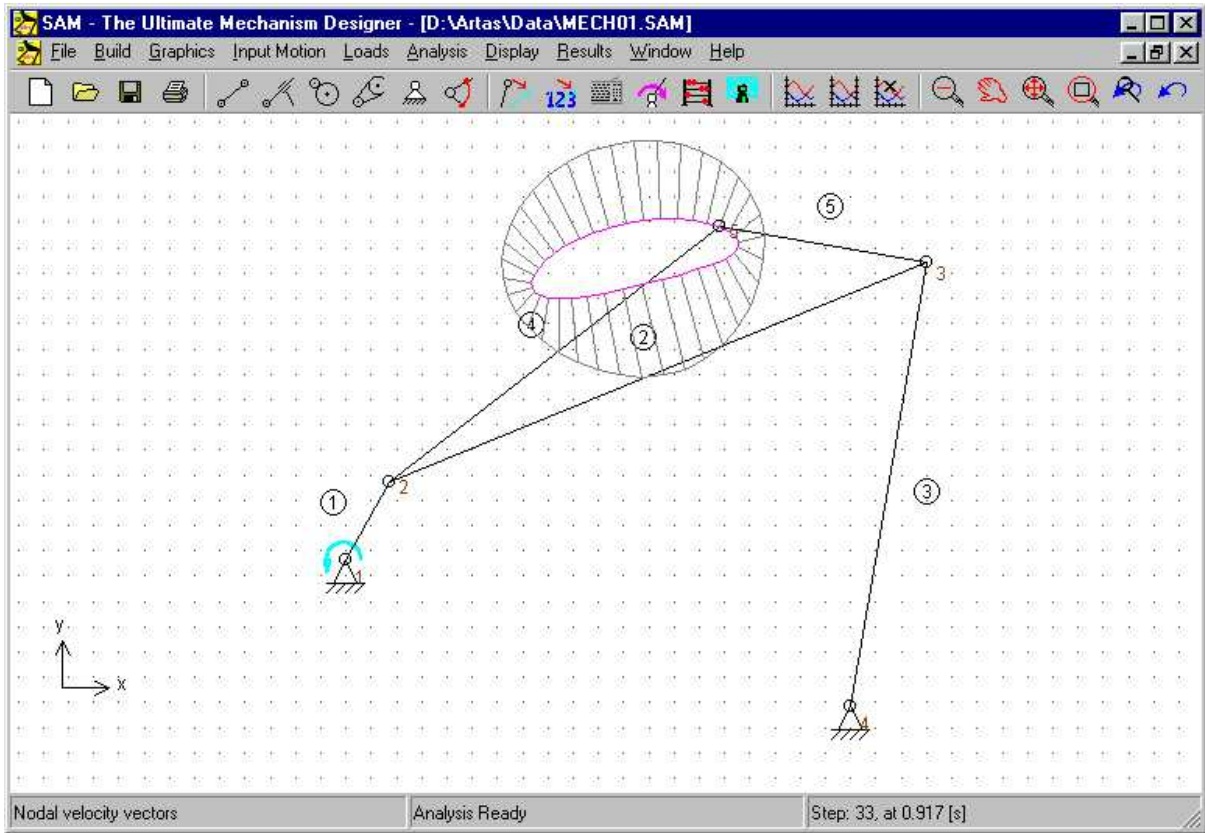
You will now see an animated display of the mechanism.

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

You will now see the path of that point.

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

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



Path and velocity hodograph of coupler point

Chapter

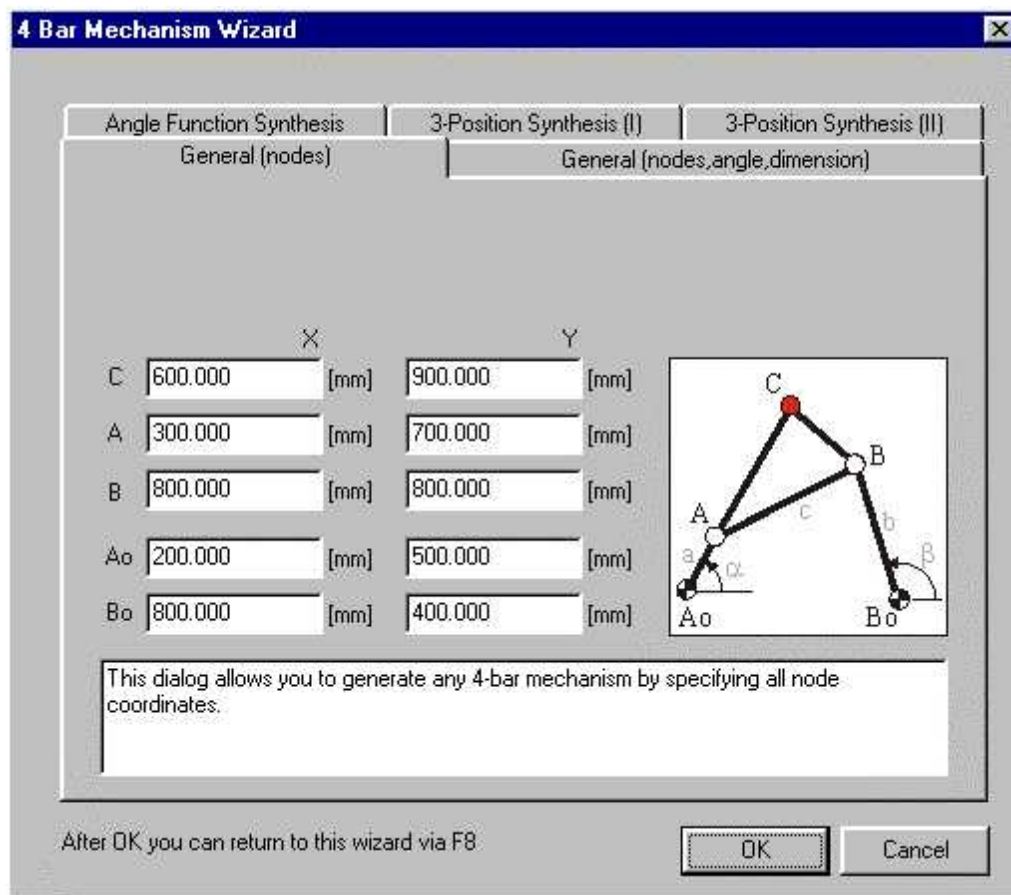
5

5 Design Wizards

5.1 4-Bar Mechanism

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

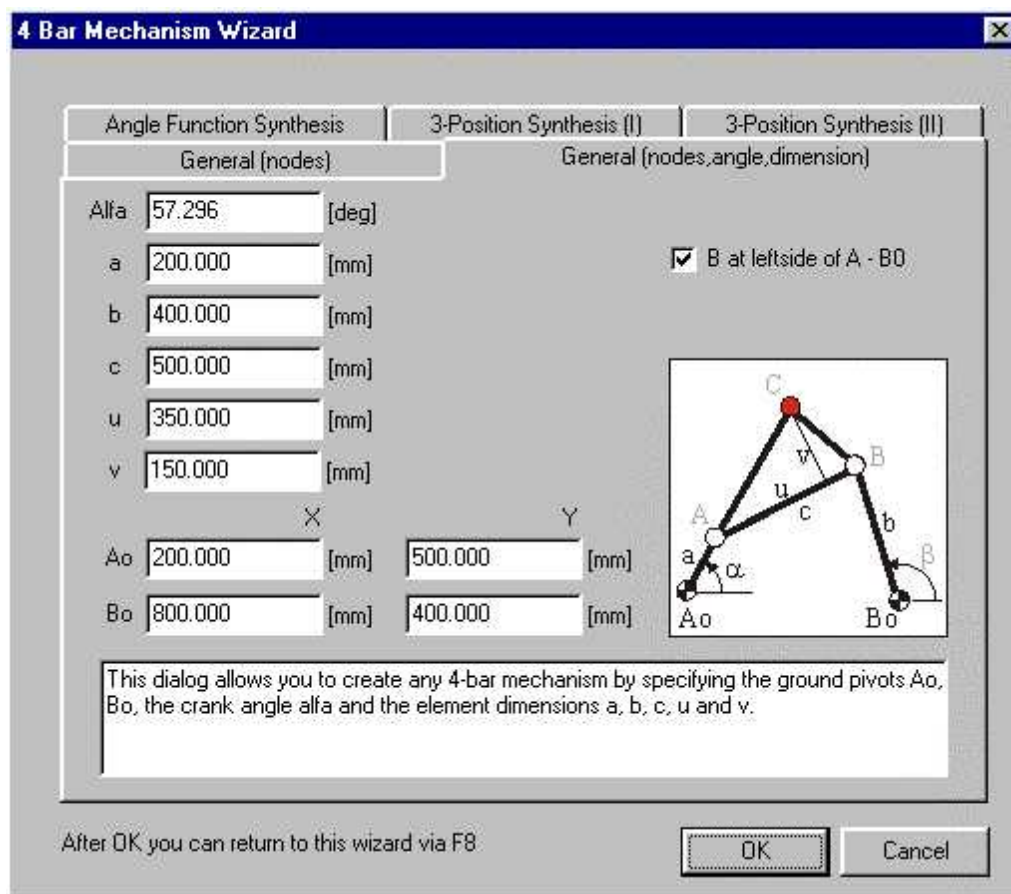
5.1.1 General (nodes)



4-Bar Mechanism Wizard: General (nodes)

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

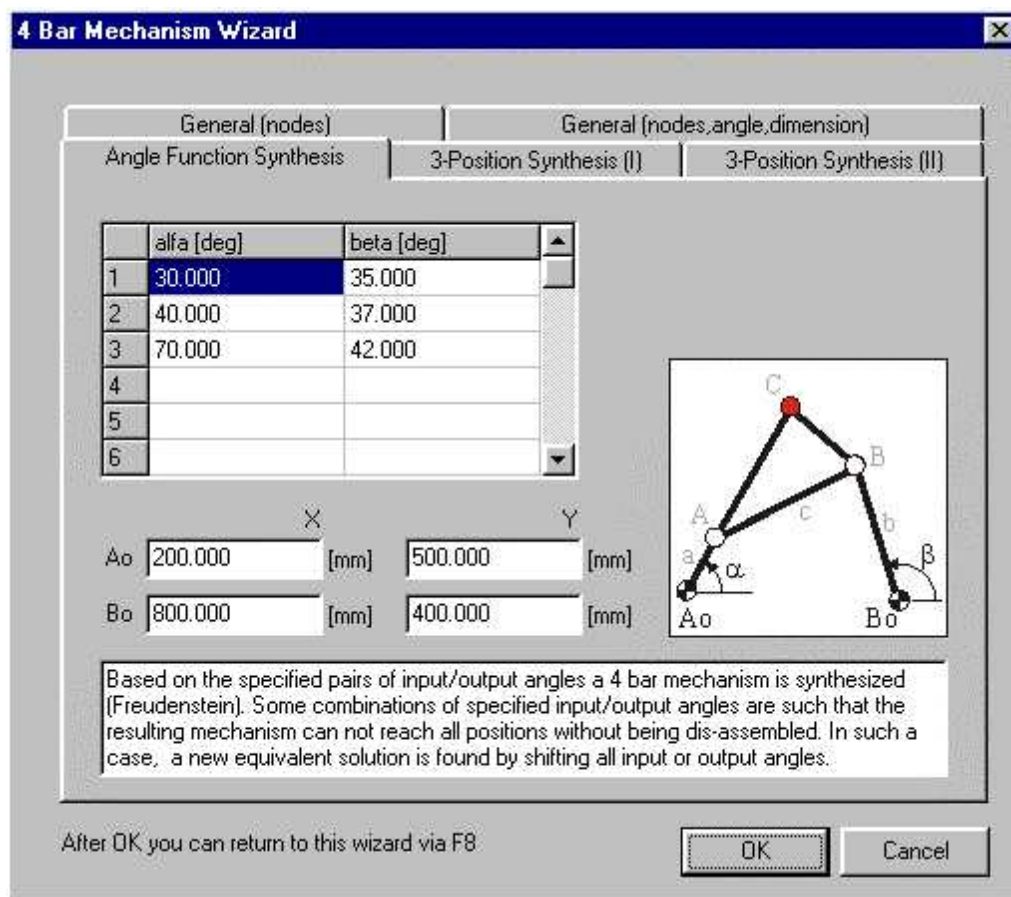
5.1.2 General (nodes, angle, dimension)



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

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

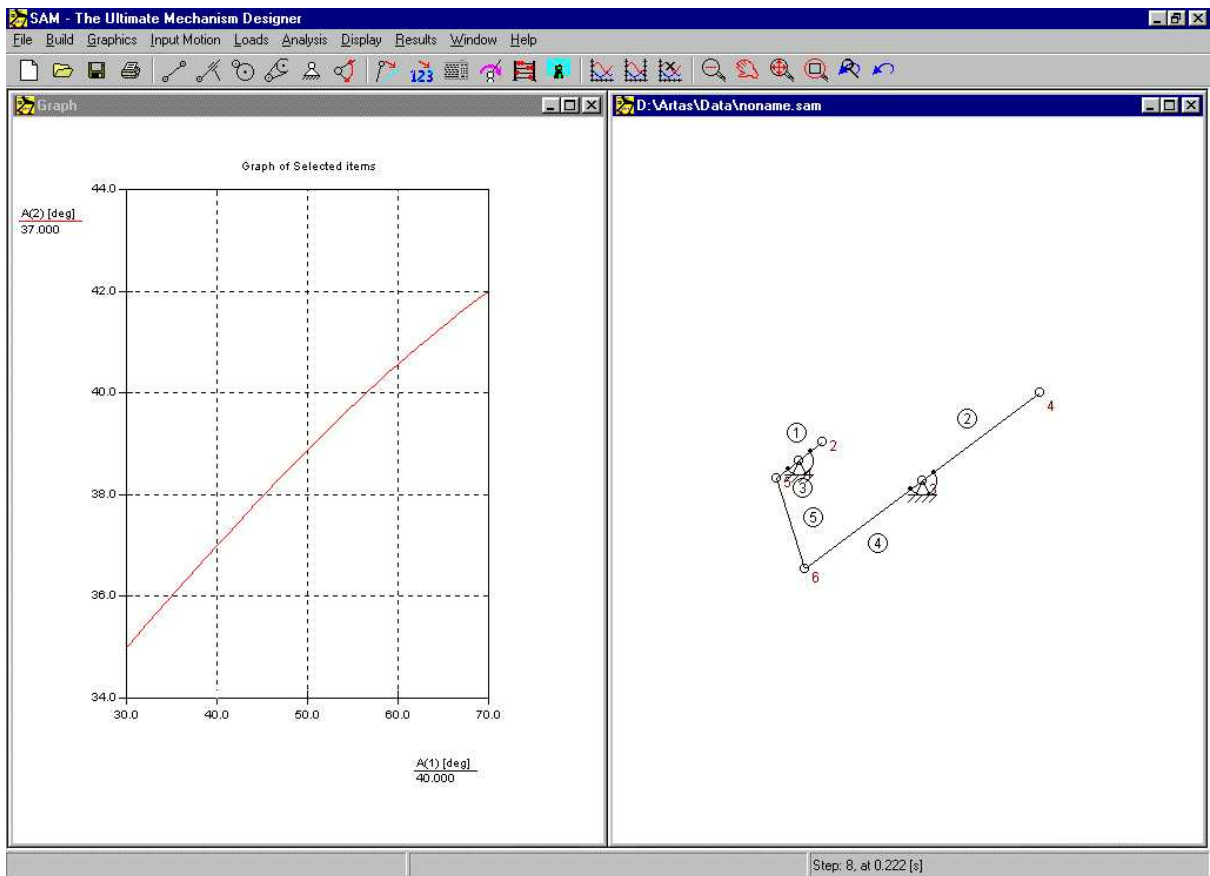
5.1.3 Angle Function Synthesis



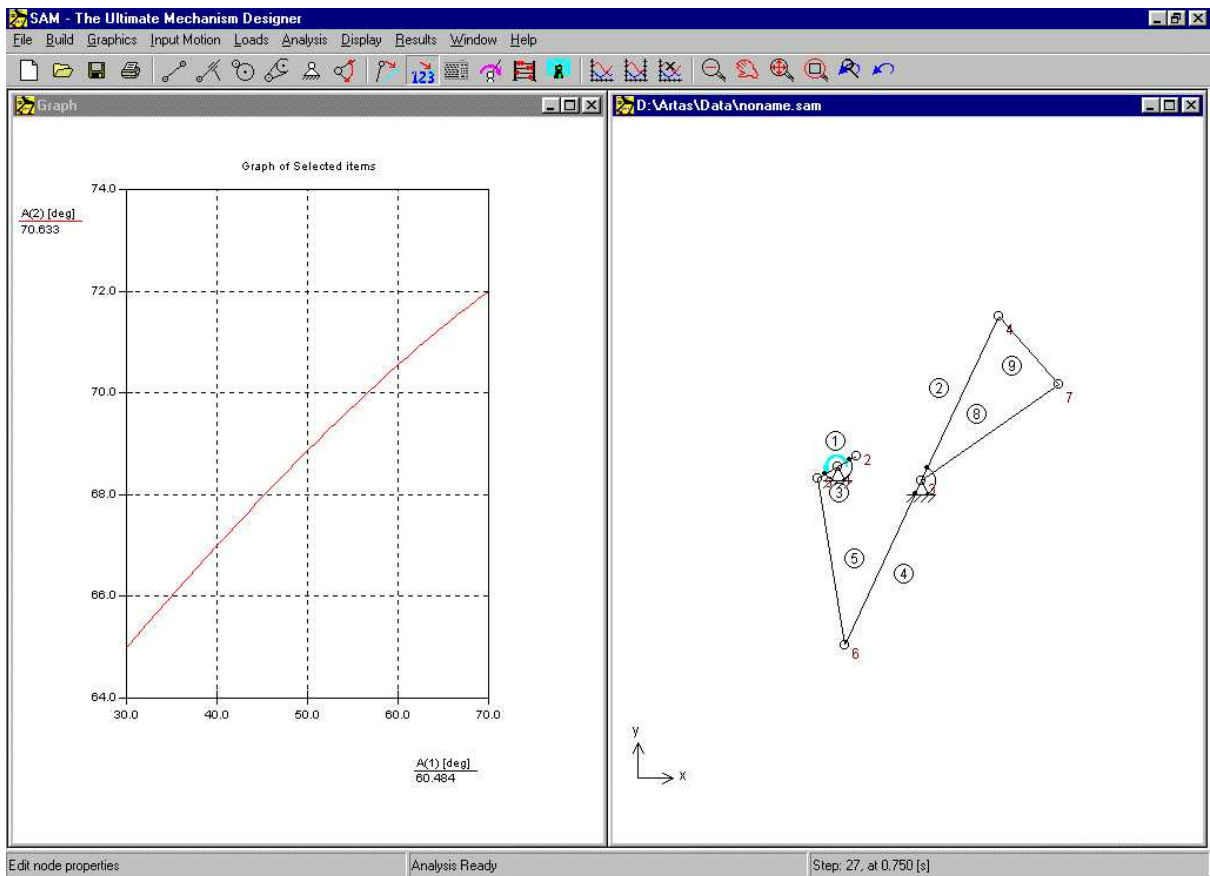
4-Bar Mechanism Wizard: Angle Function Synthesis

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 inbetween positions. In such a situation, the user can attempt to look for an alternative solutions by shifting all input or all output angles and correcting this shift by adding an extra beam to the crank to compensate for the offset-angle.

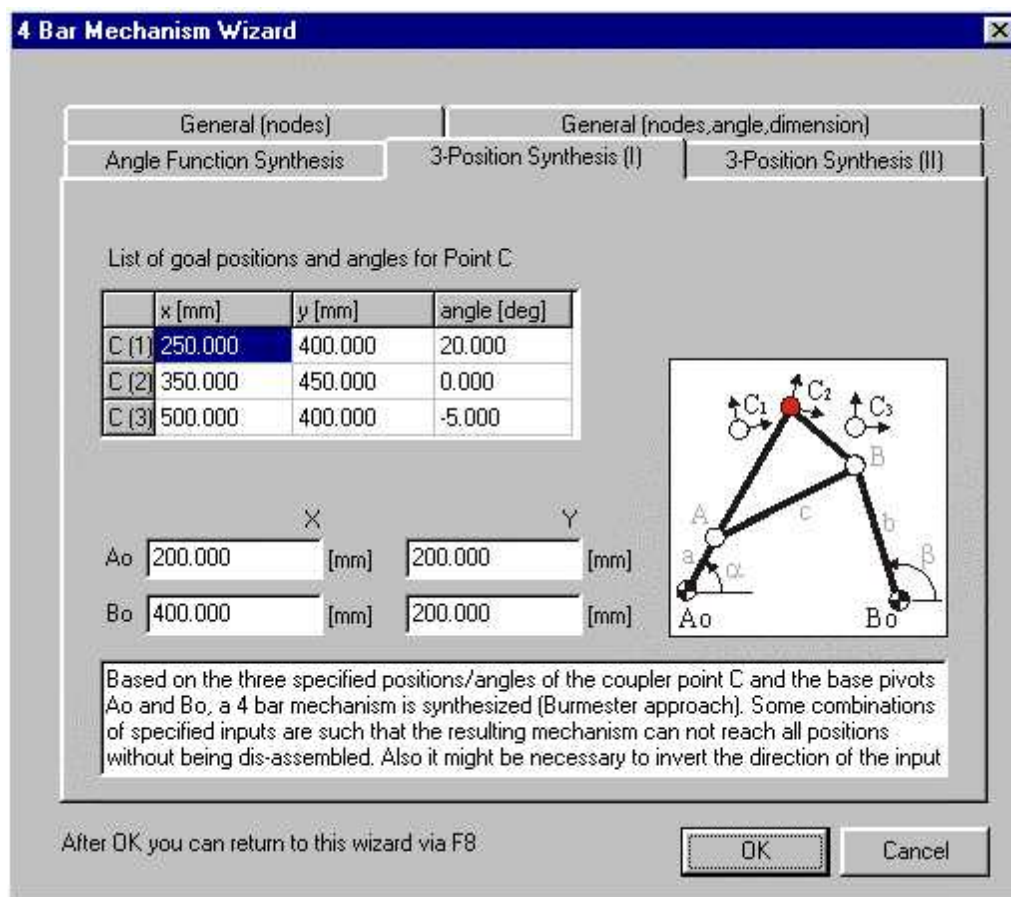


Result of angle function generation (based on default values)



Result of angle function generation (30-degree shift in output angle requirement is compensated by extra beam)

5.1.4 3-Position Synthesis (I)



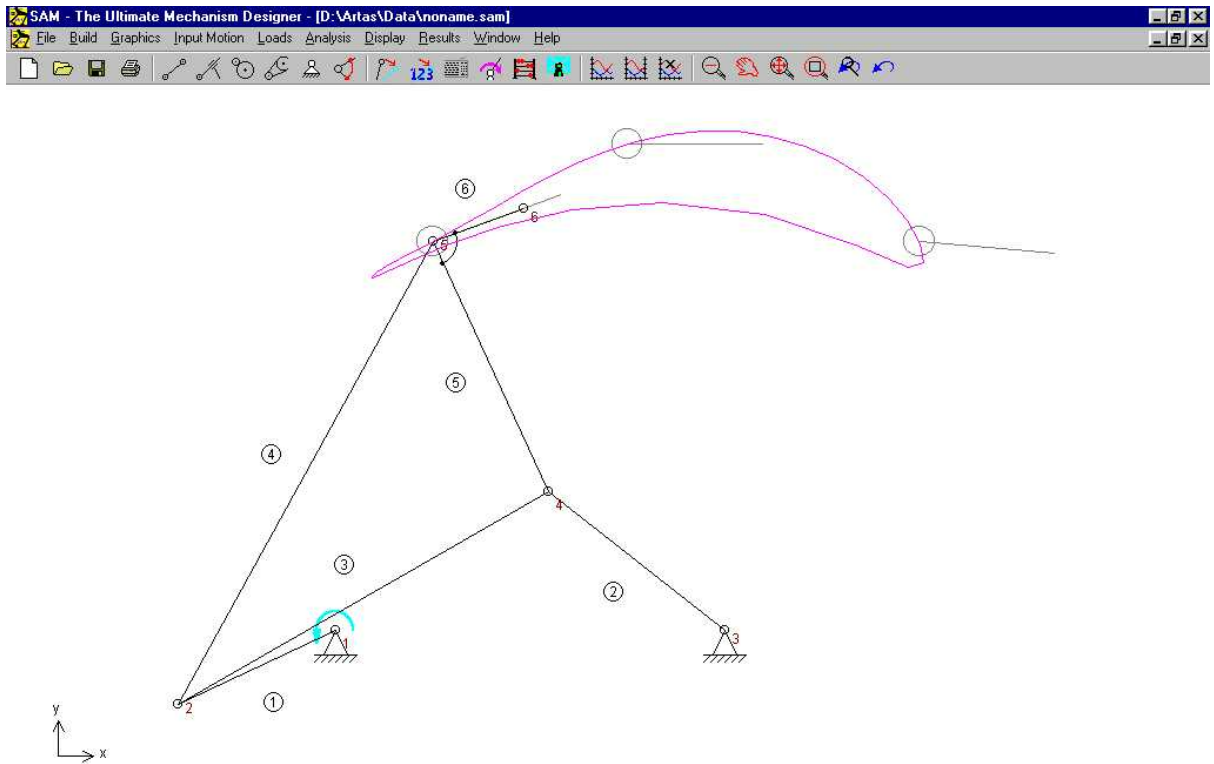
4-Bar Mechanism Wizard: 3-Position Synthesis (I)

Based on three specified positions/angles of the coupler point C and the location of the base pivots A₀ and B₀, a 4-bar mechanism is synthesized, which generates a motion of the coupler point that passes exactly through these three prescribed positions with the prescribed orientations of the coupler plane.

Some combinations of requirements are such that the resulting mechanism can not reach all positions without being dis-assembled inbetween positions. In such a situation, the user can attempt to specify different locations of the base pivots A₀ and B₀. Furthermore, in many design situations only the starting and end position are really fixed and it is tolerable to vary the middle position.

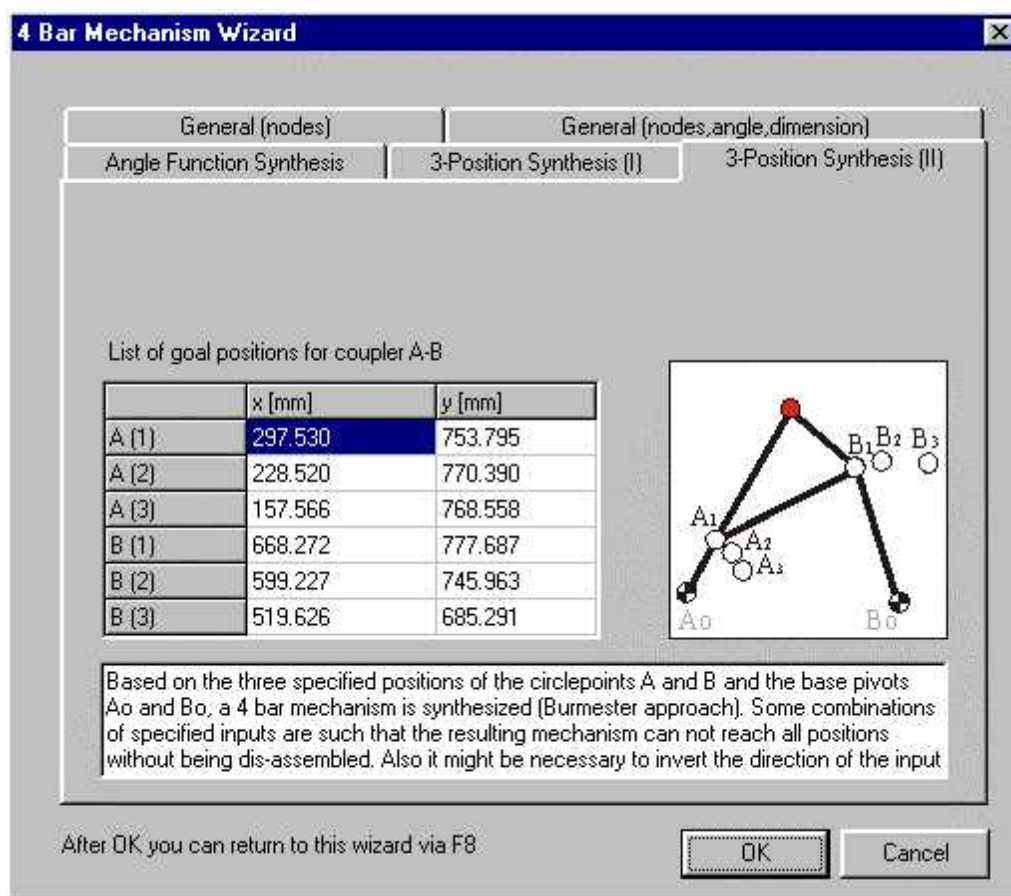
In case the couplerpoint passes through the prescribed positions in the wrong order, one has to change the direction of the input motion.

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



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

5.1.5 3-Position Synthesis (II)



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

Based on three specified positions of the circlepoints A and B and location of the base pivots A₀ and B₀, a 4-bar mechanism is synthesized, which generates a motion that satisfies these requirements.

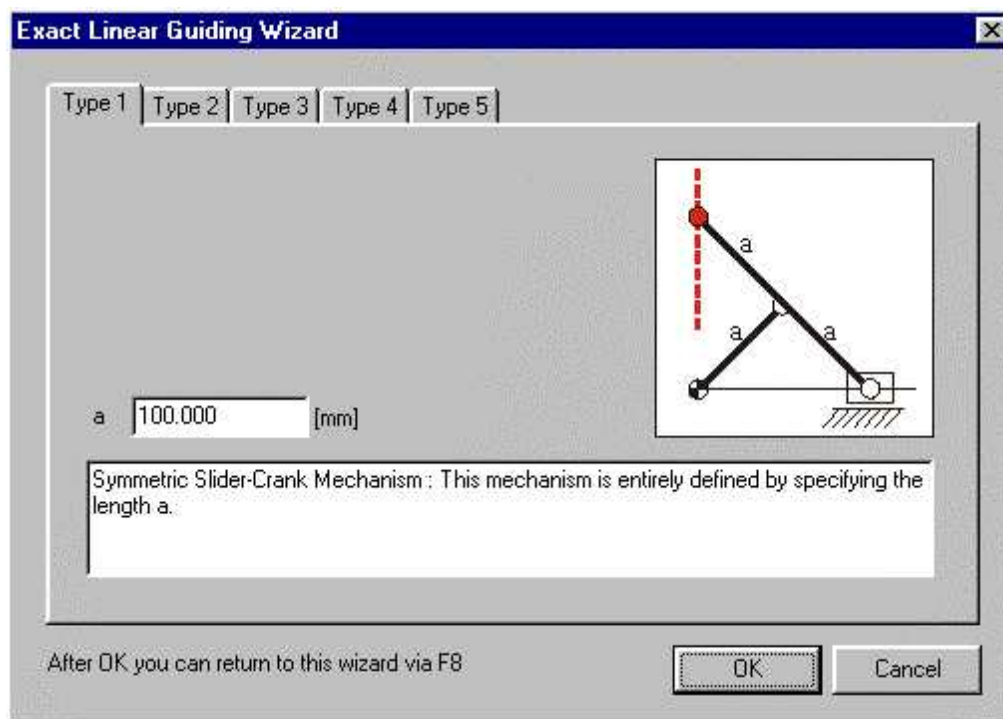
Some combinations of requirements are such that the resulting mechanism can not reach all positions without being dis-assembled inbetween positions. In such a situation, the user can attempt to specify different locations of the base pivots A₀ and B₀. Furthermore, in may design situations only the starting and end position are really fixed and it is tolerable to vary the middle position.

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

5.2 Exact Straight Line Mechanism

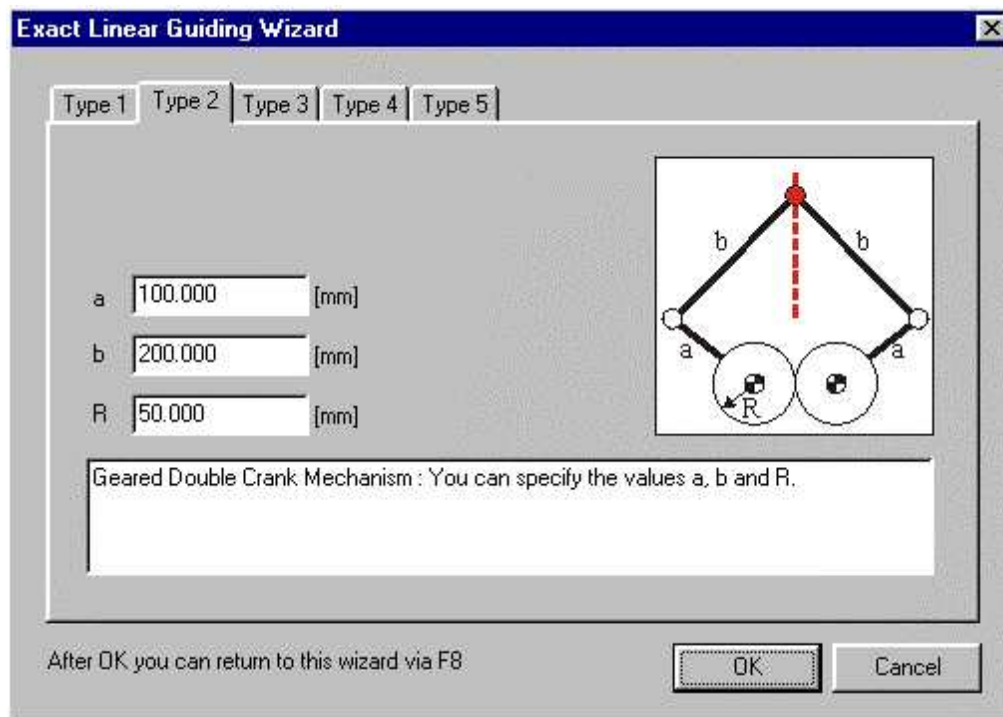
The menu File/Wizard/Exact Linear Guiding invokes the dedicated dialog, which allows you to generate a number of parametrically predefined straight line mechanisms in a very easy way.

5.2.1 Symmetric Slider-Crank Mechanism



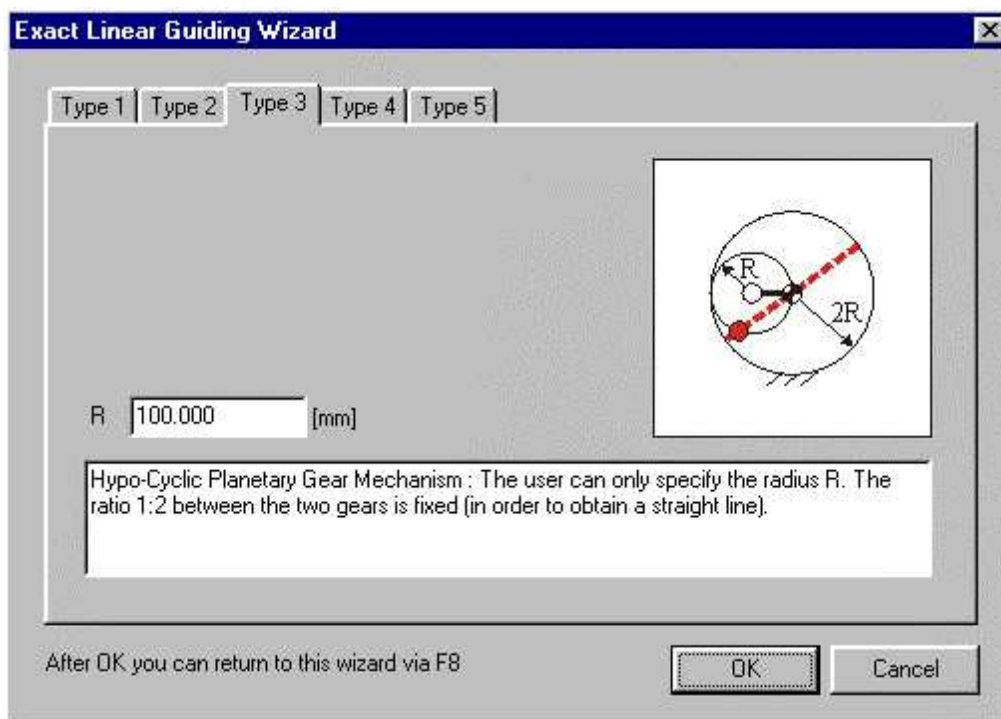
Exact Straight Line Motion Wizard: Symmetric Slider-Crank Mechanism

5.2.2 Geared Double-Crank Mechanism

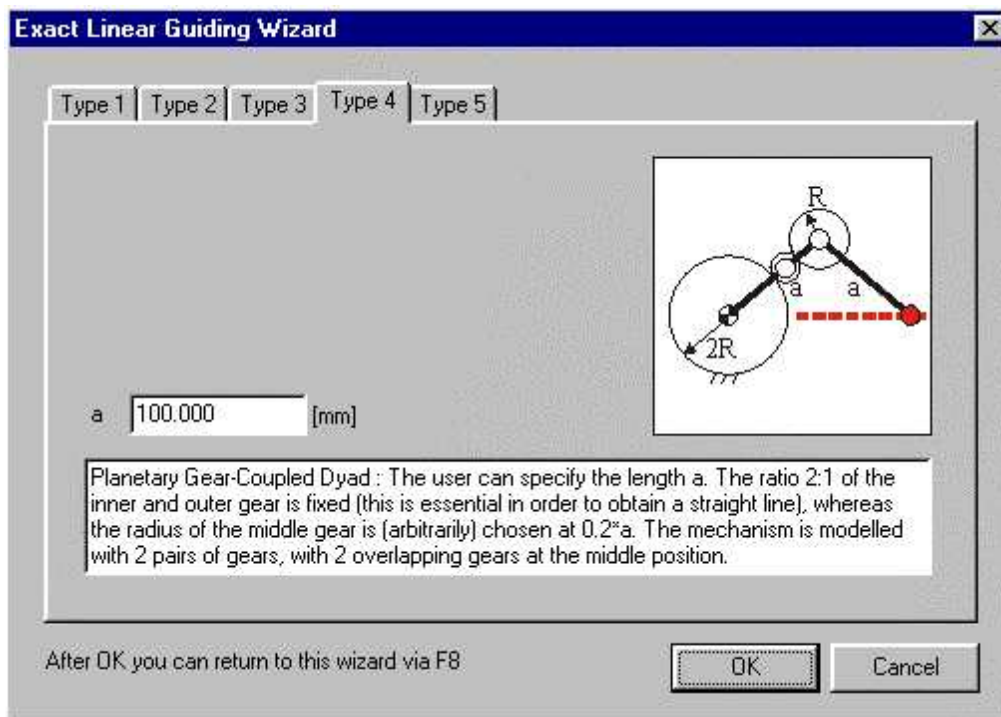


Exact Straight Line Motion Wizard: Geared Double-Crank Mechanism

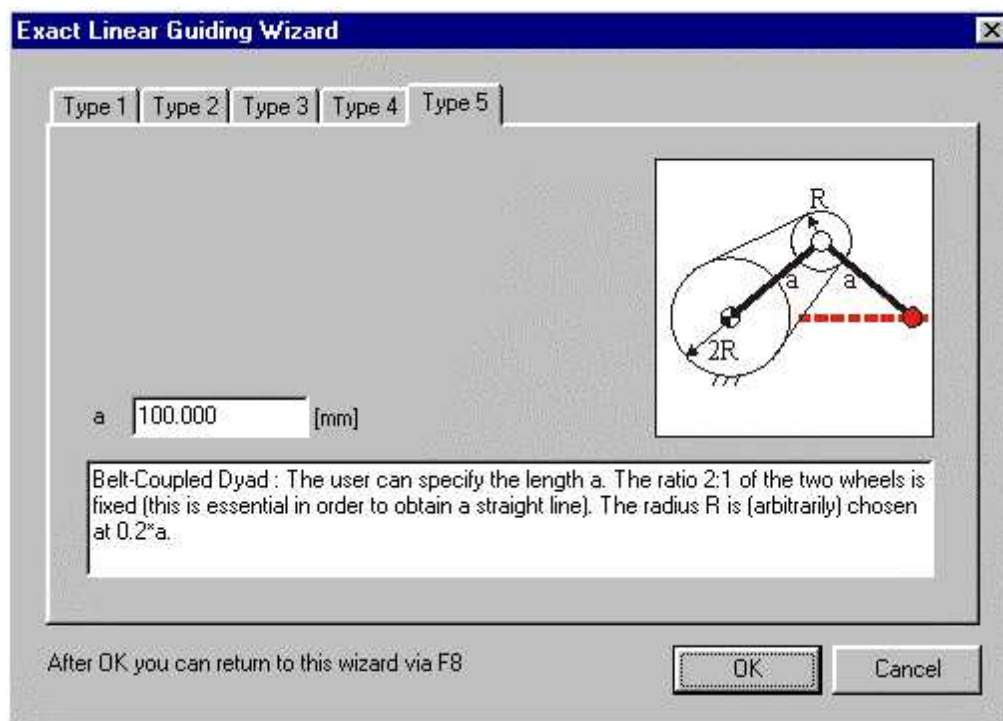
5.2.3 Hypo-Cyclic Planetary Gear Mechanism



5.2.4 Planetary Gear-Coupled Dyad



5.2.5 Belt-Coupled Dyad

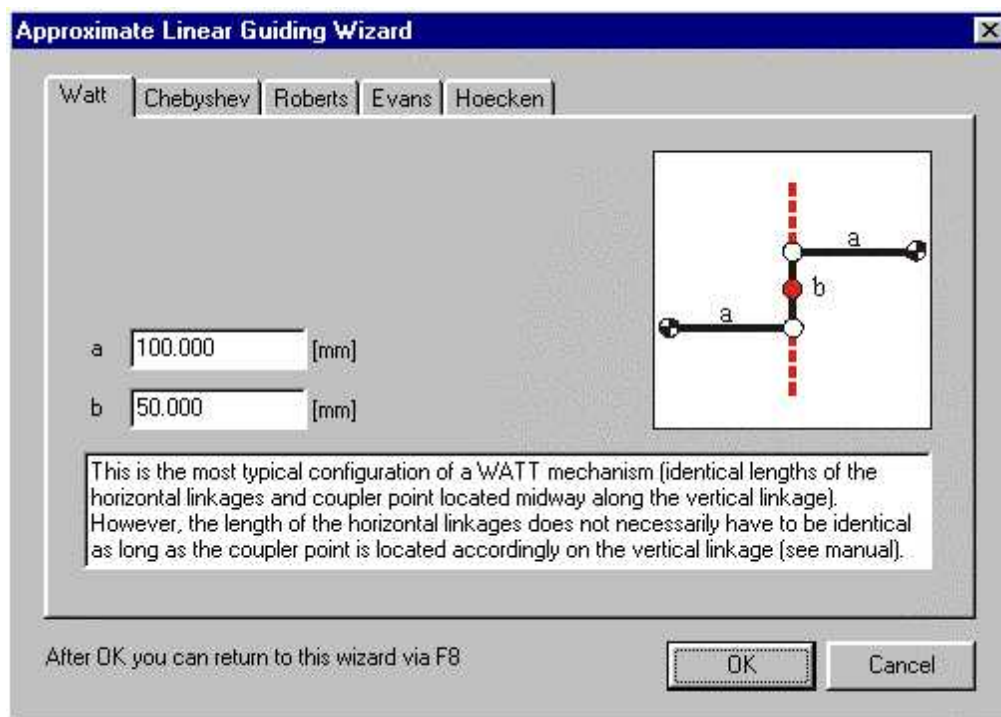


5.3 Approximated Straight Line Mechanism

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

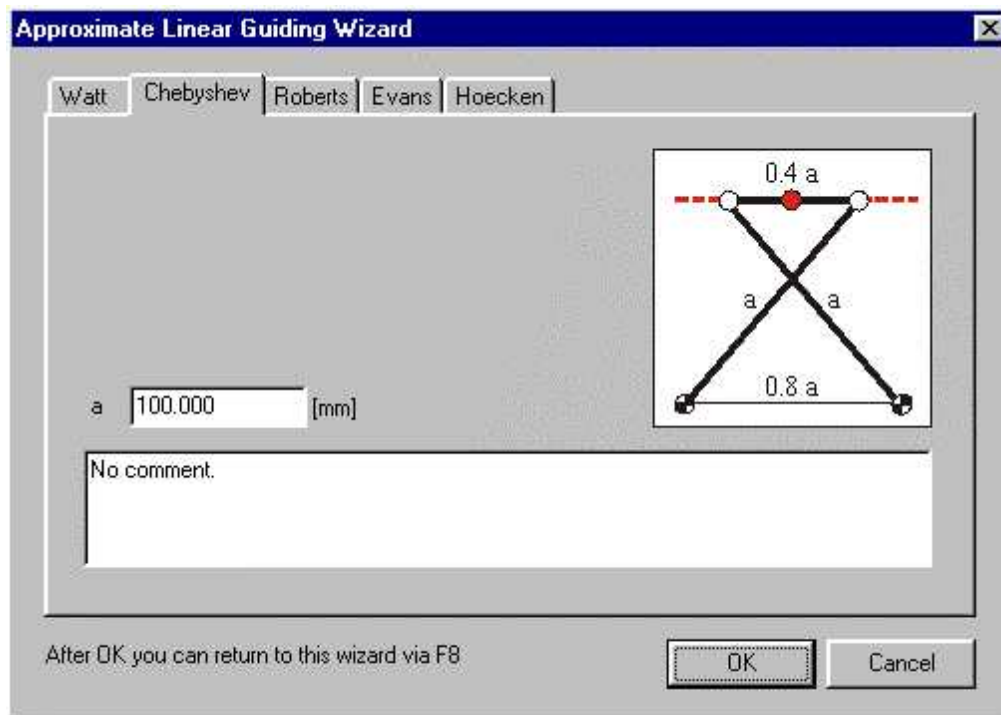
- Watt
- Chebyshev
- Roberts
- Evans
- Hoecken

5.3.1 Watt



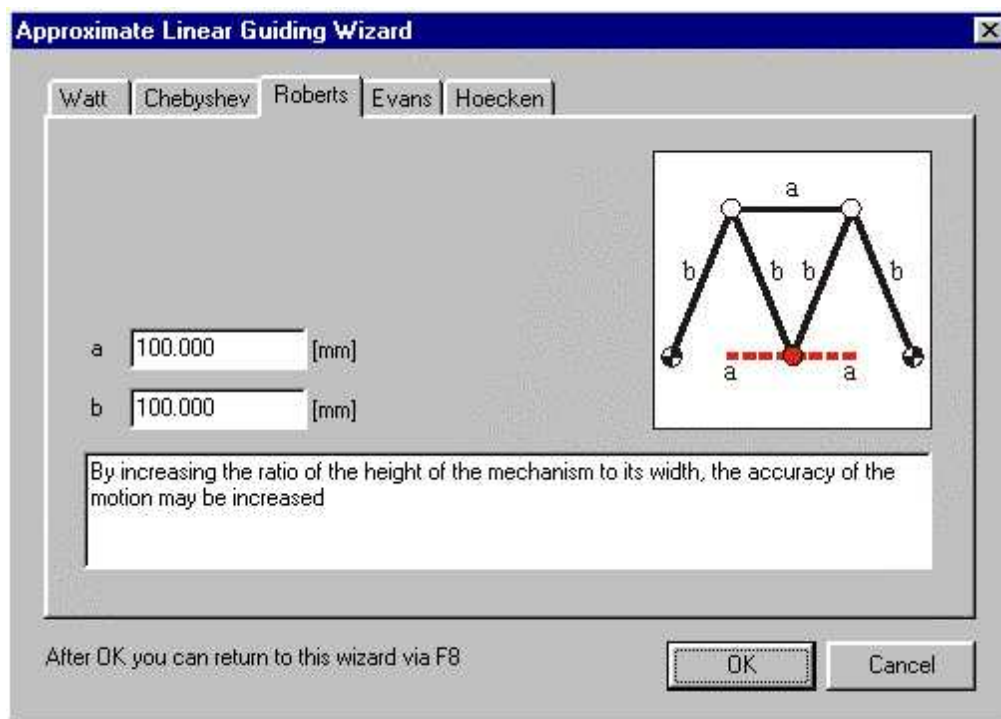
Approximated Straight Line Motion Wizard: Watt

5.3.2 Chebyshev



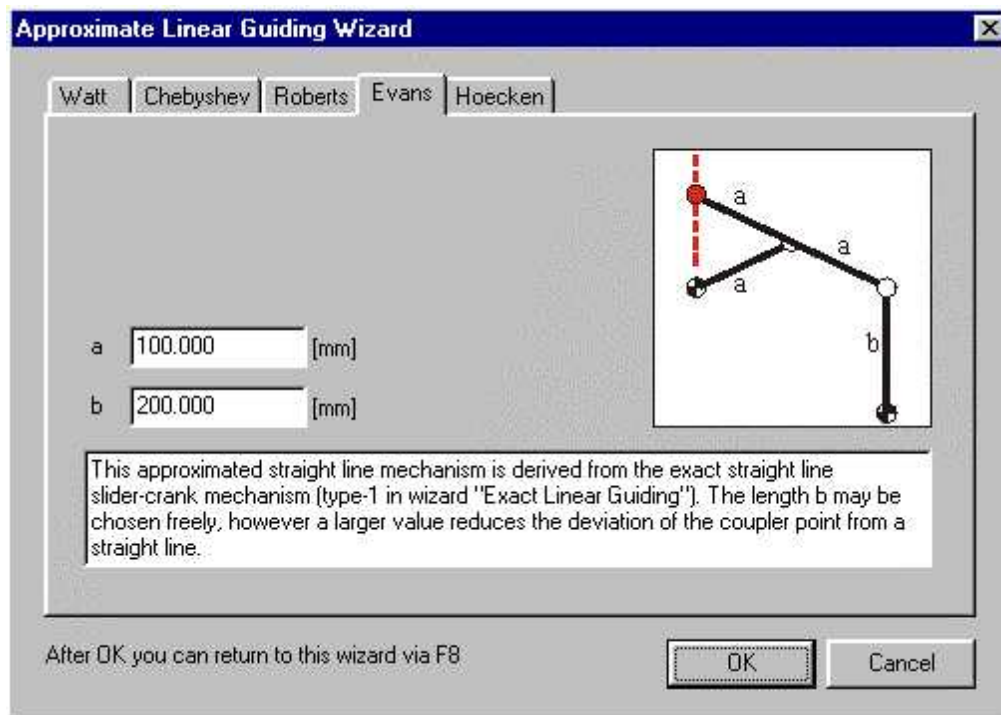
Approximated Straight Line Motion Wizard: Chebyshev

5.3.3 Roberts



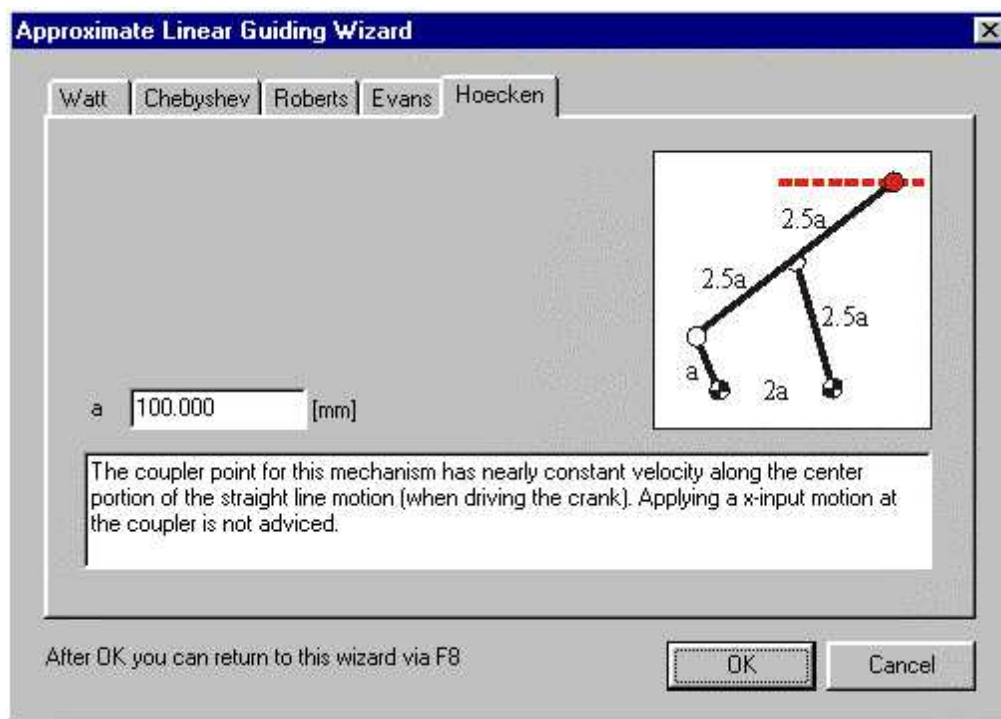
Approximated Straight Line Motion Wizard: Roberts

5.3.4 Evans



Approximated Straight Line Motion Wizard: Evans

5.3.5 Hoecken



Chapter



6

6 Modelling Considerations

6.1 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 : Data is always stored and interpreted in accordance with the SI(rad) unit system. While opening a file (SAM project file, motion file or optimization reference file) the data in that file is interpreted in accordance with the SI(rad) unit system and converted to the currently active unit system.

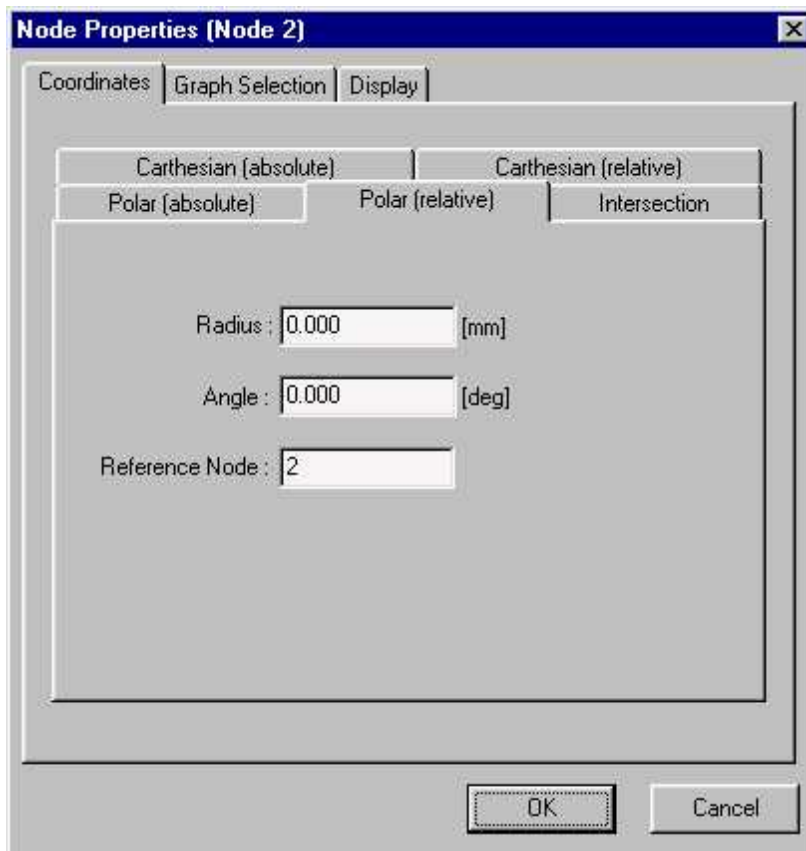
In the menu File/Preferences/Numbers you can select the way in which real numbers are represented. In the Results/Export dialog you can define different settings that only affect the exported data.

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).

6.2 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 the keyboard button  allows manual entry or selection at any stage.



Dialog for the "construction" of exact nodal coordinates

The (relative) Cartesian and polar coordinate option needs no further explanation. The last option "intersection" can locate the intersection of two circles with center nodes N1 and N2 and radii L1 and L2, respectively. You can select between the two intersections via the checkbox button. CAD-data can be imported in DXF-format and the mouse can snap to all DXF-points. This feature also facilitates the creation of exact nodal coordinates.

6.3 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 unit system of a motion file is always SI (m, m/s, m/s² and rad, rad/s, rad/s²).

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.

6.3.1 Format

A motion file is a simple ASCII text file that defines the motion profile in SI(rad) engineering units at specified moments in time in the following format:

0	0	vo	ao	If this is absent, the default values vo=ao=0 are used for the velocity and acceleration 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.

The engineering unit system used for motion file is ALWAYS the SI(rad) system. While the motion file is imported, its data is automatically converted to the actual units used in the current project. In case motion data is exported to a motion file, a conversion takes place automatically 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, 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
```

6.3.2 Importing a motion file

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

Please note that the data in the motion file is interpreted as being consistent with the SI unit system (m,m/s,m/s² and rad, rad/s, rad/s²). During import of the data a conversion is made to the currently active unit system.

6.3.3 Exporting a motion file

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

When exporting analysis results in the motion file format, you must make sure that you currently have a valid selection of export data. A valid set consists of 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 set that you need for the motion file.

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

6.4 Multiple Input Motions

In SAM one is able to define up to 10 simultaneous input motions. However, the following rule must be considered :

Each motion input must have the same timing !! Calculations are carried out at discrete moments in time, which must coincide for the various inputs.

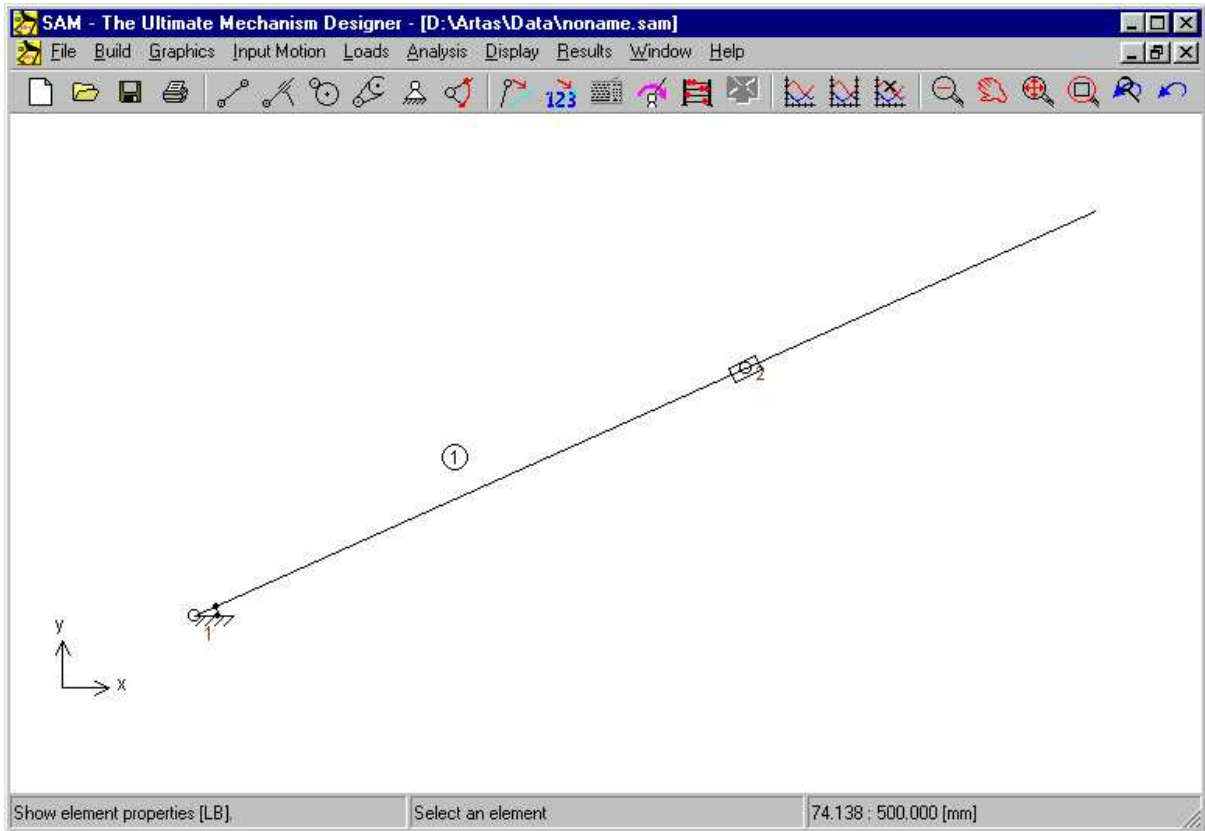
The different motion profiles can be as complex as the user wants, as long as the above mentioned rule is not violated. In case the timing/synchronization of the inputs is different, the last input will completely overrule the timing of the other inputs, which results in a different input motion than expected.

6.5 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.

6.6 Inclined Linear Guiding

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

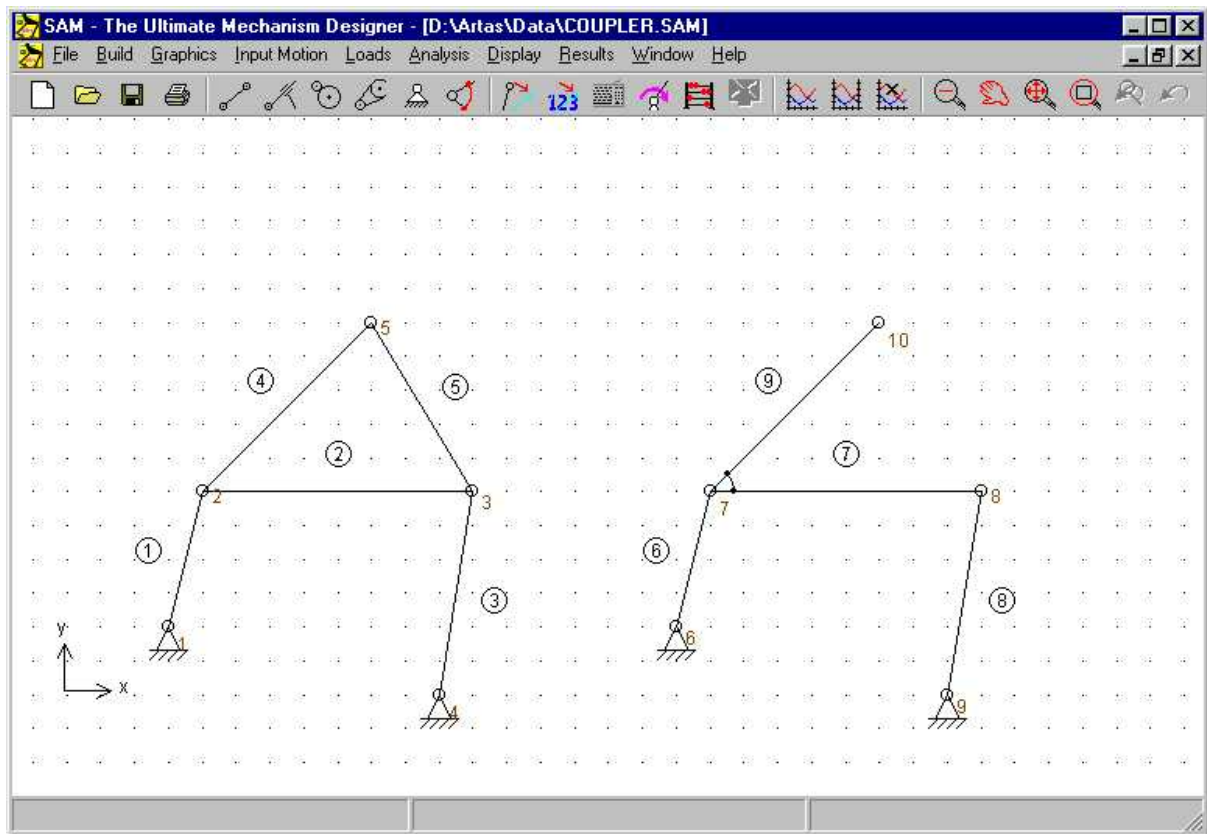


Modeling of an inclined linear guiding using a SLIDER element

6.7 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 followed :

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-joint. (area of the triangle may not become zero, therefore this approach can not be used when the three nodes are located in-line).



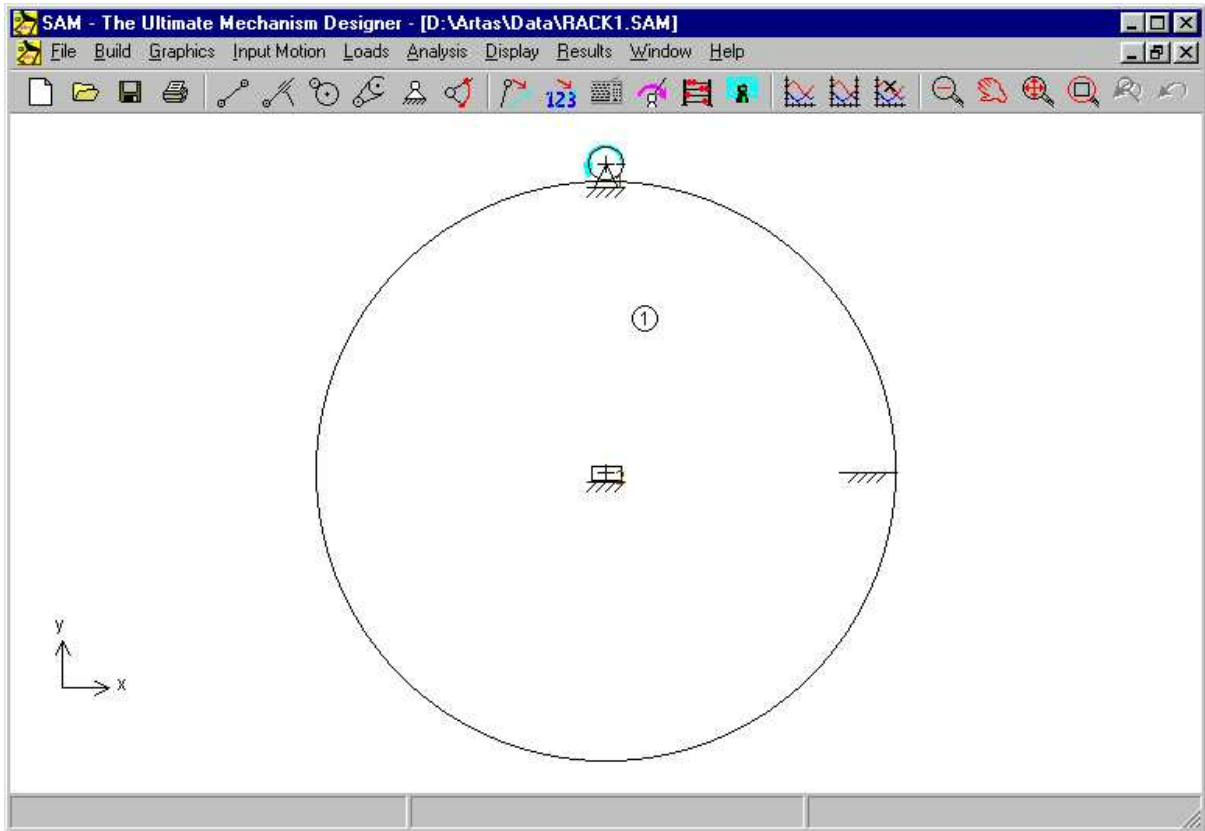
Two ways of modeling a coupler point

6.8 Modelling a rack/pinion

SAM does not (yet) possess a special element to model a rack/pinion. However, it is possible to model such a transmission by using either a gear or a belt element, which will be illustrated via the modelling of a horizontal rack :

a. GEAR ELEMENT (see example rack1.sam)

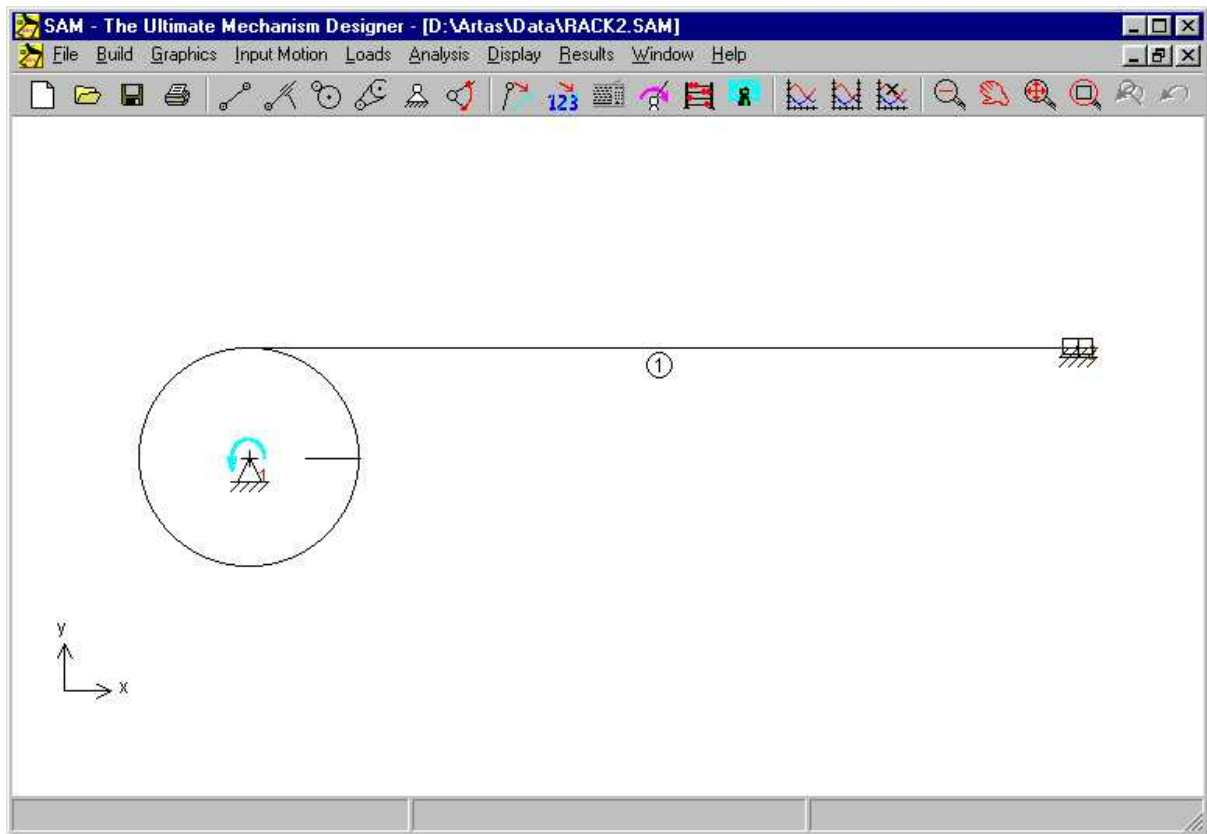
- The gear is modelled in the normal way, whereas the rack is modelled as a huge gear. Both gears have the same x-coordinate.
- The gear is fixed in x- and y-direction
- The rack (huge gear) is fixed in y-direction and in rotation
- The gear is driven



Modelling a rack/pinion using a gear element (RACK1.SAM)

b. BELT ELEMENT (see example rack2.sam)

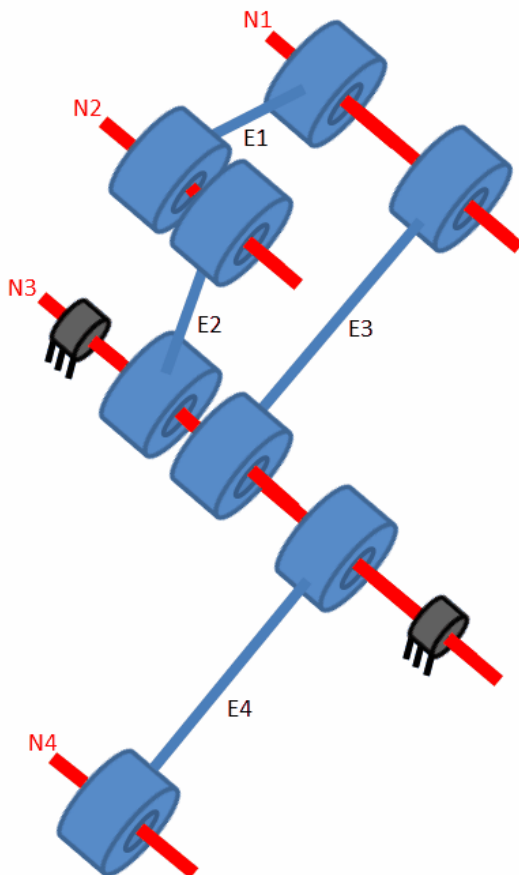
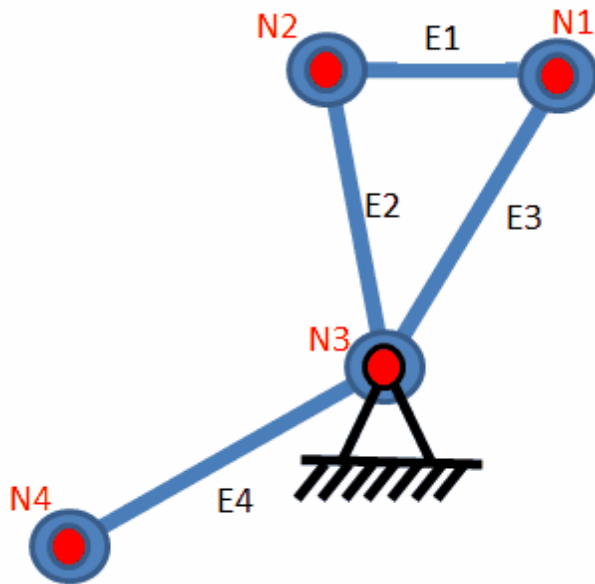
- One of the two wheels (A) of the the belt represents the pinion, whereas the second wheel (B) has a radius that is "almost zero" (due to numerical problems one should not use "zero").
- The y-coordinates of the wheels are chosen as in the example.
- Wheel A is fixed in x- and y-direction
- Wheel B is fixed in y-direction and in rotation
- Wheel A is driven



Modelling a rack/pinion using a belt element (RACK2.SAM)

6.9 Bearing Forces

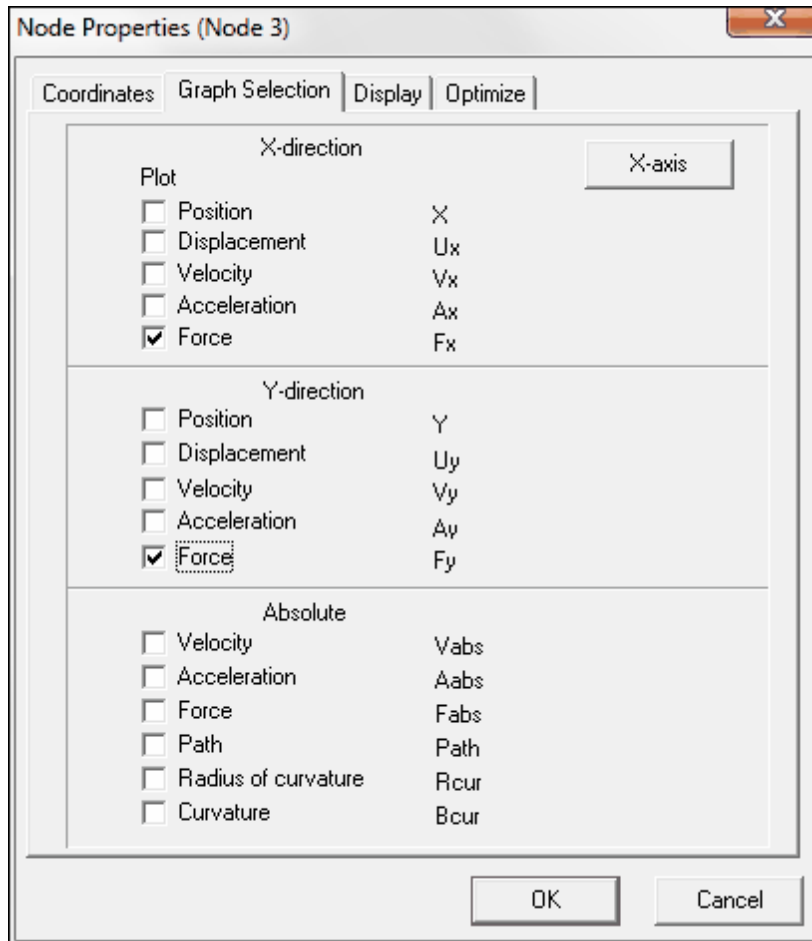
Next to the driving force or driving torque it is also essential to know reaction and bearing forces. To better understand the way reaction and bearing forces are treated in SAM it helps to realize that a mechanism is a collection of elements (blue) that share nodes (red) and to visualize these nodes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



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

Reaction forces in fixed node

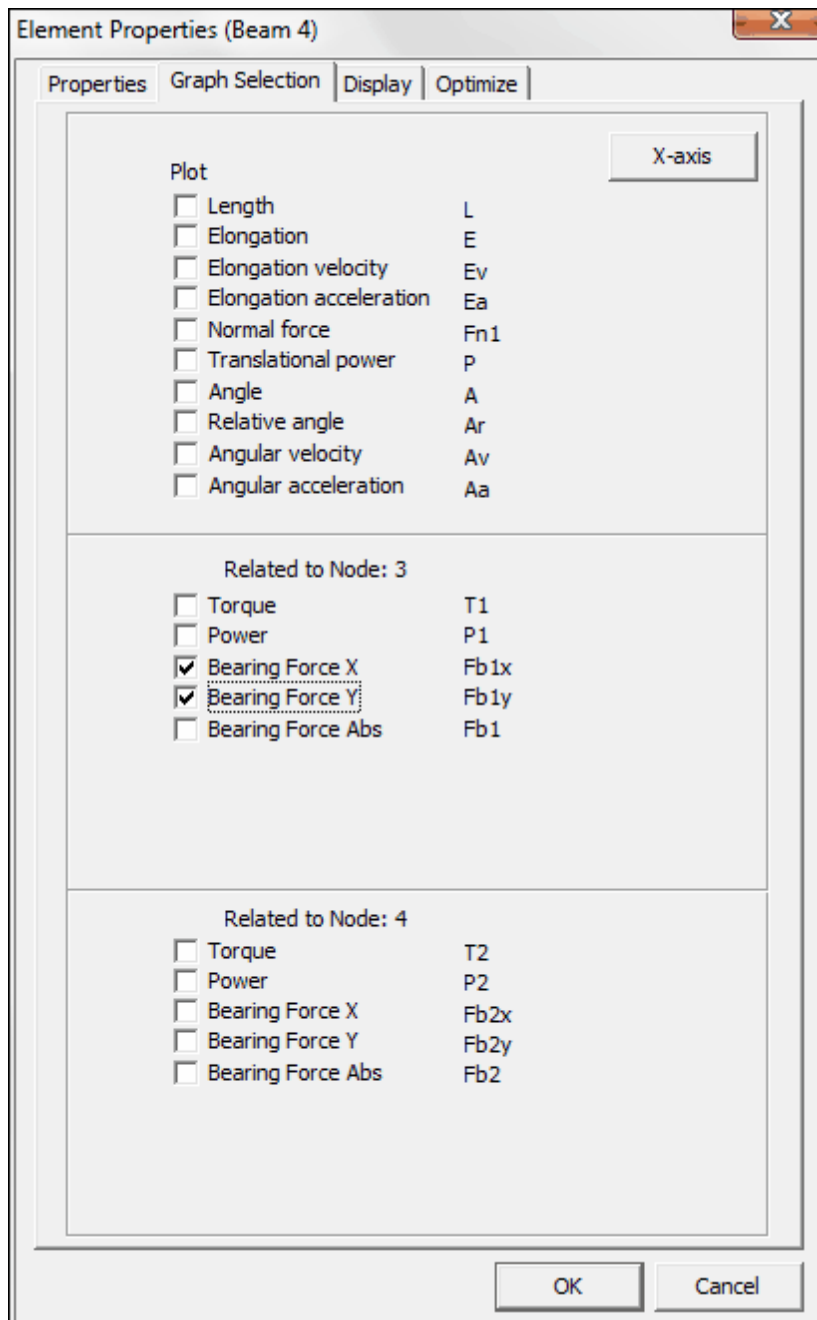
If one needs to know the combined reaction forces that are transferred via node/shaft N3 to the stationary part of the machine one would have to select Fx, Fy and/or Fabs in the node selection dialog.



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

Bearing forces of an element

If one needs to know the forces that are transferred via a bearing of an element to the connecting shaft (node) - and eventually from there to the rest of the mechanism or a support - one would have select Fb1x, Fb1y and/or Fb1 in the element selection dialog.



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

6.10 Using Graphics

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

1. A conceptual design is made in SAM and the relevant data is exported in DXF-format to the CAD program for further processing. This will then lead to a mechanism design in the CAD environment. In

the next step the mechanism design is transferred to SAM in DXF-format. In SAM the various items can be transformed (translated, rotated) and they can be attached to already existing elements.

2. A mechanism design is made directly in the CAD environment, which is then transferred to SAM in DXF-format. In SAM the conceptual mechanism can be constructed on the basis of the imported drawing. Next, groups of graphical items can be attached to elements.

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

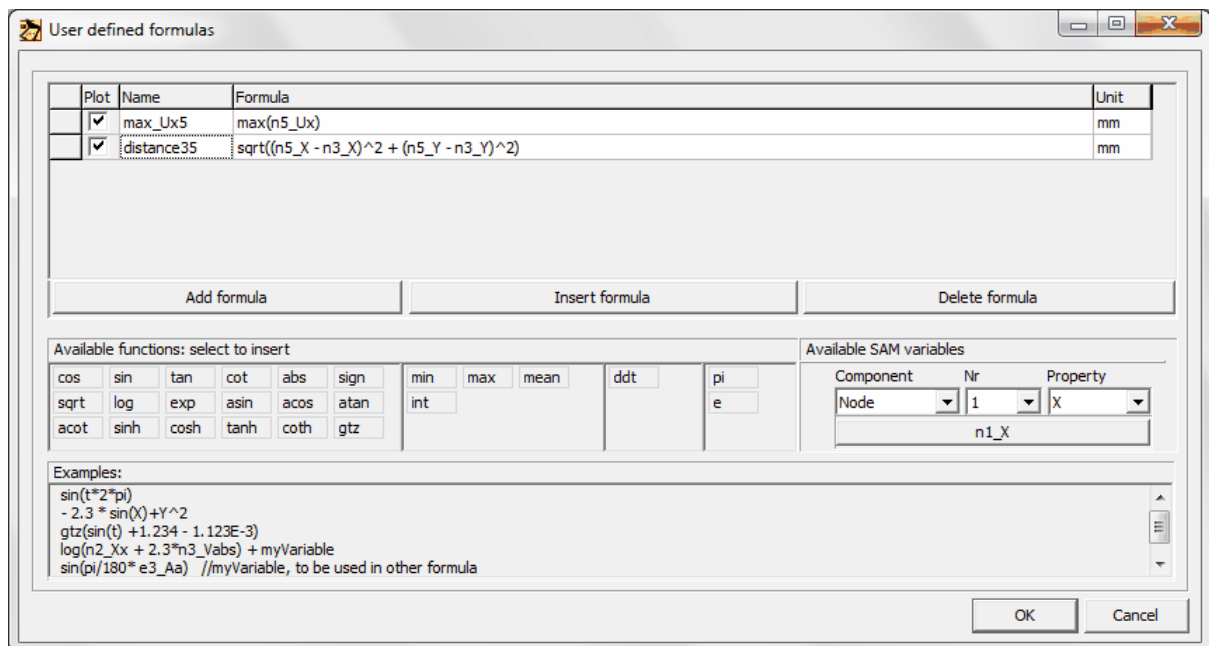
Initially the graphical entities of each imported DXF-file are treated as one group.

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

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

6.11 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:

- $1+(2-5)*3+8/(5+3)^2 \text{ sqrt}(2)$
- $(a+b)*(a-b) x^2+3*x+1$
- $256.33*\exp(-t/12) (1+(2-5)*3+8/(5+3)^2)/\text{sqrt}(5^2+3^2)$
- $2+3x+2x^2 \ 0.25x + 3.5y + 1$
- $(12.3)/(856.6) (-1)^{(2n+1)}*x^n/n!$
- $\text{sqrt}((n5\_X-n3\_X)^2 + (n5\_Y-n3\_Y)^2)$  // distance between node 5 and node 3
- $e2\_A - e3\_A$  // relative angle between element 2 and element 3

SAM variables are reserved names and can not be used to identify a user defined result.

The list consists of:

- t (time)
- ni\_xxx (property xxx of node i)
- ej\_yyy (property yy of element j)
- examples: n1\_Vabs (absolute velocity of node 1), e3\_Ar (angle of element 3 relative to initial position of mechanism)

Constant numbers can be integers, decimal, or exponential

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

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

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

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

- exponents and roots



- multiplication and division
- addition and subtraction

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

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

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

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

#### Minus Sign

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

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

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

#### Internal precision

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



# Chapter



7

## 7 Optimization

### 7.1 Introduction

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

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

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

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

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

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

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

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

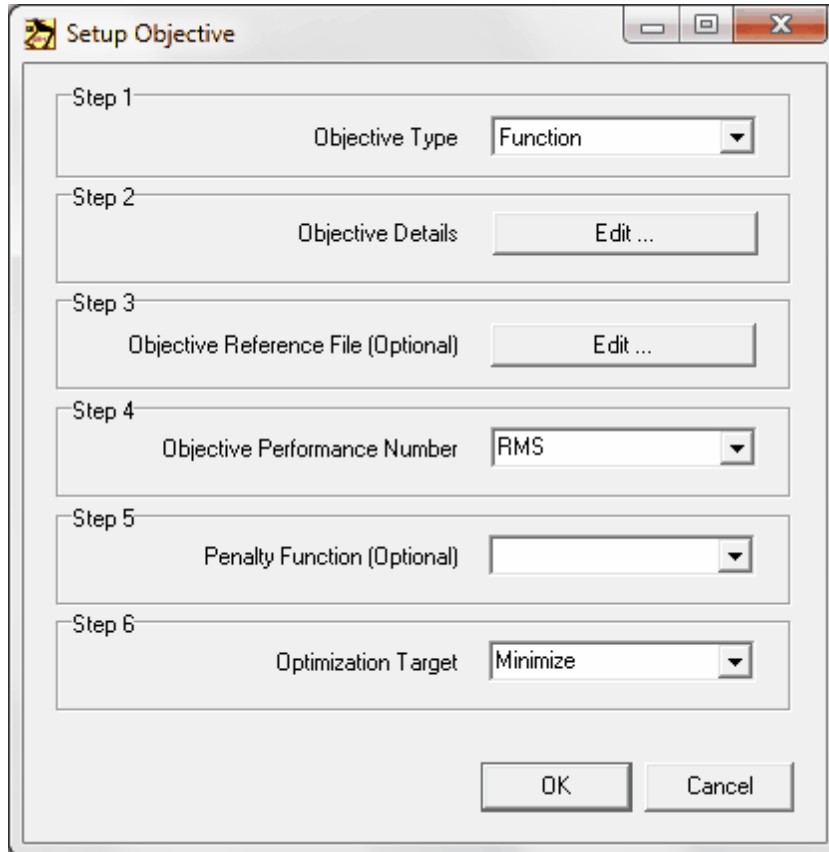
First, the entire parameter space is explored globally using a combination of a pure Monte-Carlo technique and a so-called Evolutionary Algorithm, which is an optimization technique derived from Genetic Optimization. The top list of such a global exploration are shown in the Explore list box, which displays 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 an 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.

## 7.2 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

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

### Step 2: Objective Details

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

### Step 3: Objective Reference File (Optional)

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

**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 (maximum, absolute maximum, 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.

**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.

**7.2.1 Reference or Target Definition**

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

**Function**

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

**Path without timing**

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

**Path including timing**

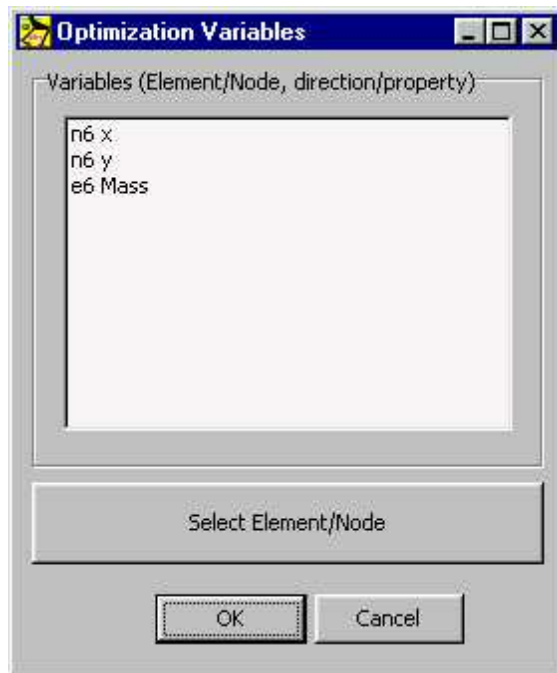
```
time_1 x_1 y_1
time_2 x_2 y_2
: : :
time_n x_n y_n
```

- 
1. In general, all target or reference data is interpreted in SI(rad) units and converted to the current unit system. However, when optimizing user defined variables the numbers in the reference file are interpreted as is and compared to the numerical result of the user defined variables (without considering engineering units).
  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.
- 

**7.3 Parameter Space**

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

plus a button to select additional parameters.



**Dialog showing currently selected parameters plus button to select additional parameters**

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



Dialog optimization settings of a node

## 7.4 Constraints

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

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

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

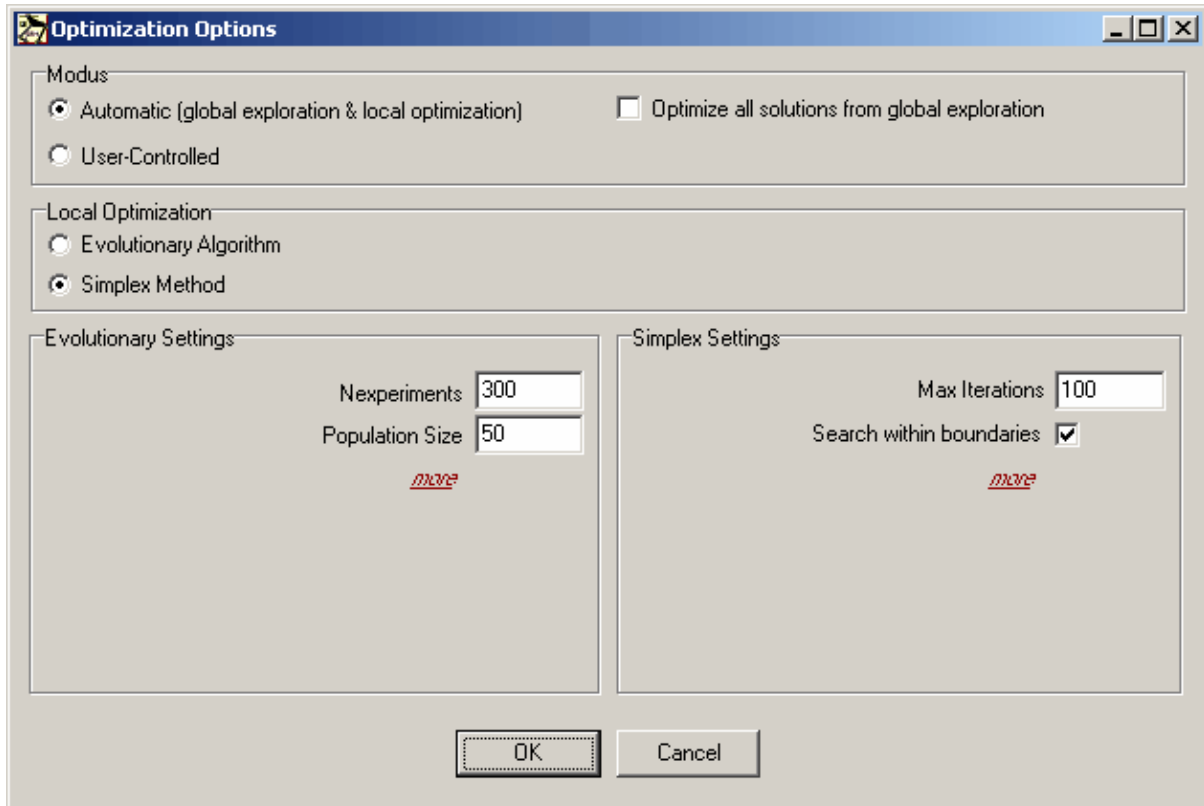
The idea is to increase the error function when the constraint is violated. A distinction can be made with respect to feasibility of the violation. When a small violation can be accepted, the exterior type can be



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.

## 7.5 Optimization Options

The menu Optimization/Options invokes the following dialog :



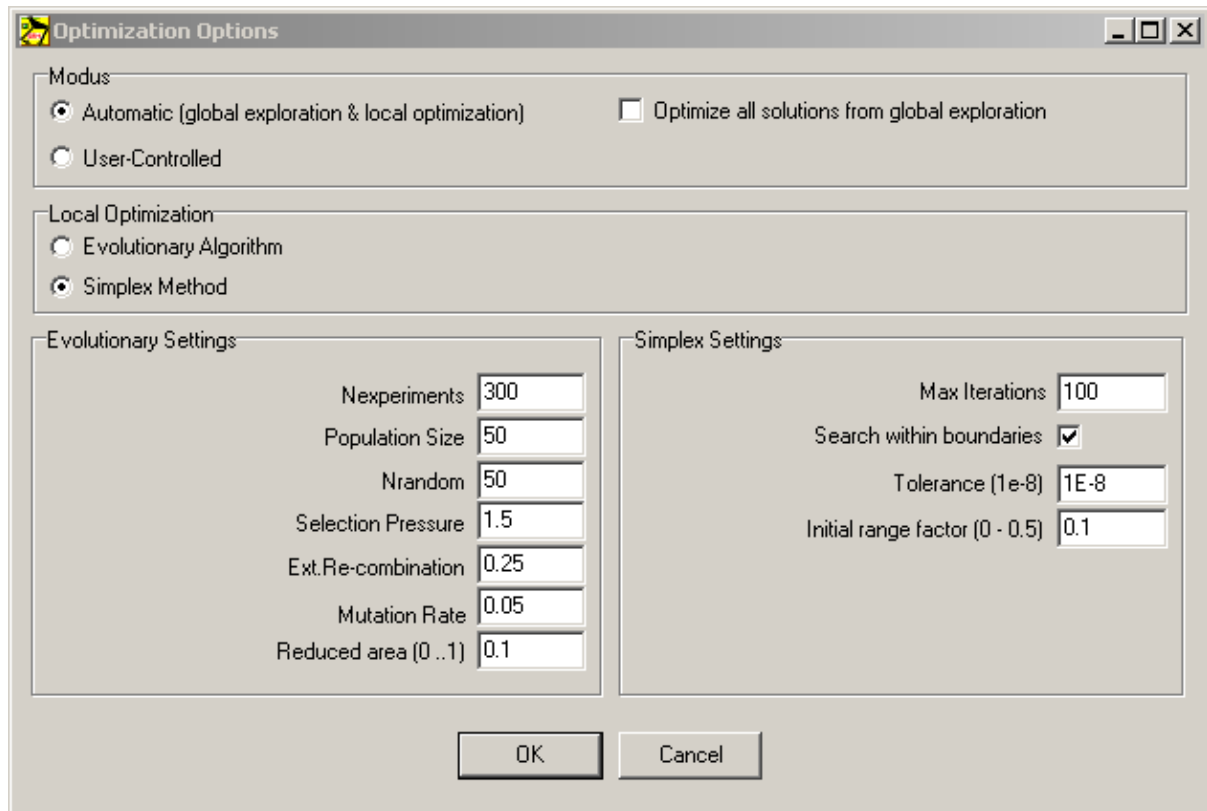
Dialog of standard optimization options

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

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

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

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



Dialog showing the full range of optimization options

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.

### 7.5.1 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 continues while replacing individuals in the population by better offsprings to maintain the size of the population.

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

**Ext. Re-combination:** In the chosen implementation of the evolutionary algorithm, the generation of

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

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

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

## 7.5.2 Settings Simplex Algorithm

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

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

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

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



# Chapter

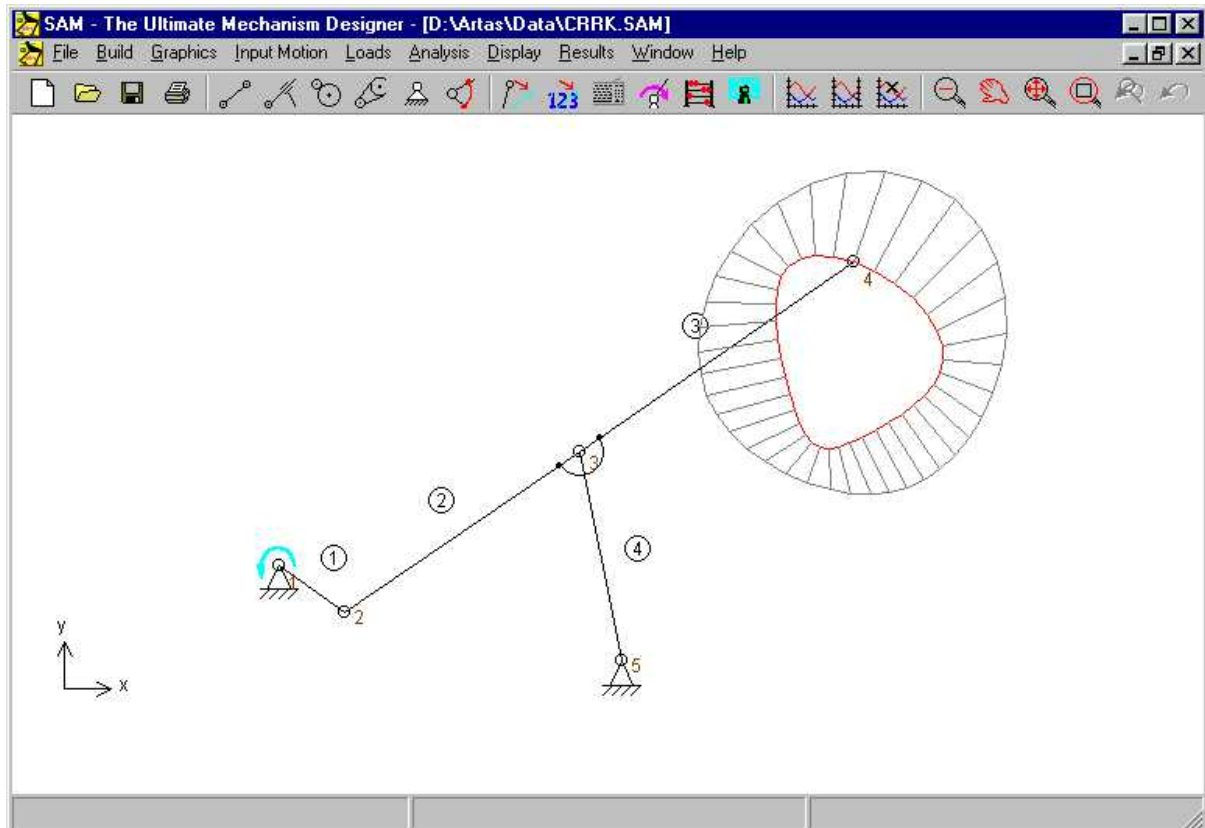
8

## 8 Examples

### 8.1 Motion Analysis

#### 8.1.1 Crank Rocker Mechanism

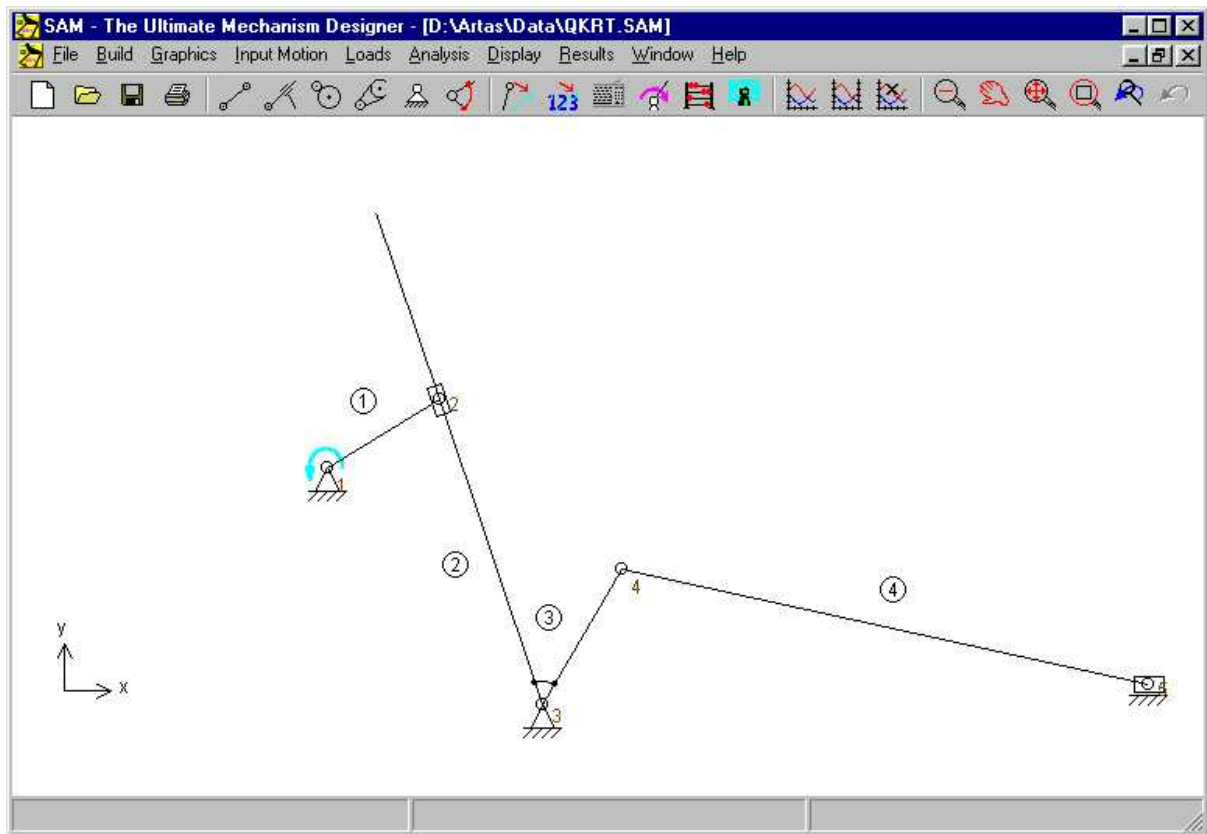
Example file CRRK.SAM defines a crank rocker mechanism, the crank of which is driven with constant speed. The path and velocity of node 4 is analyzed.



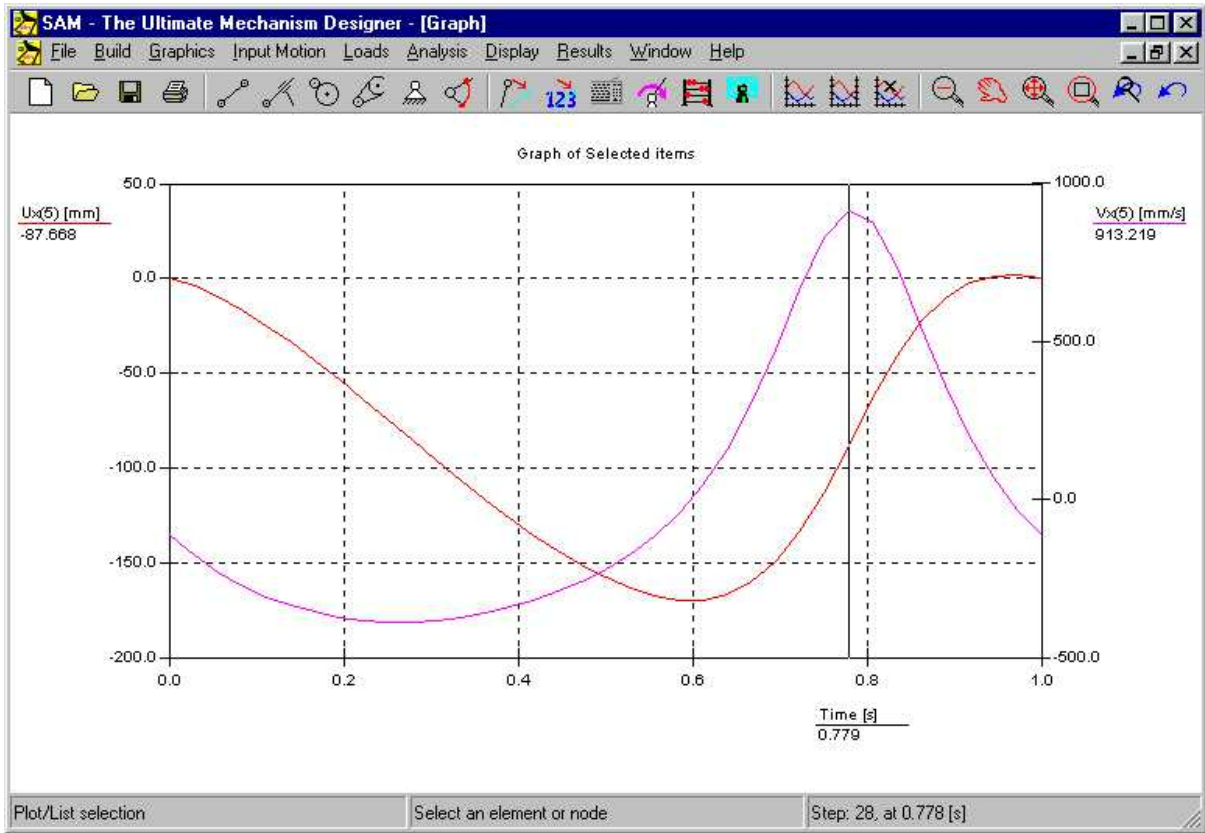
Crank Rocker Mechanism (CRRK.SAM)

#### 8.1.2 Quick-Return Mechanism

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



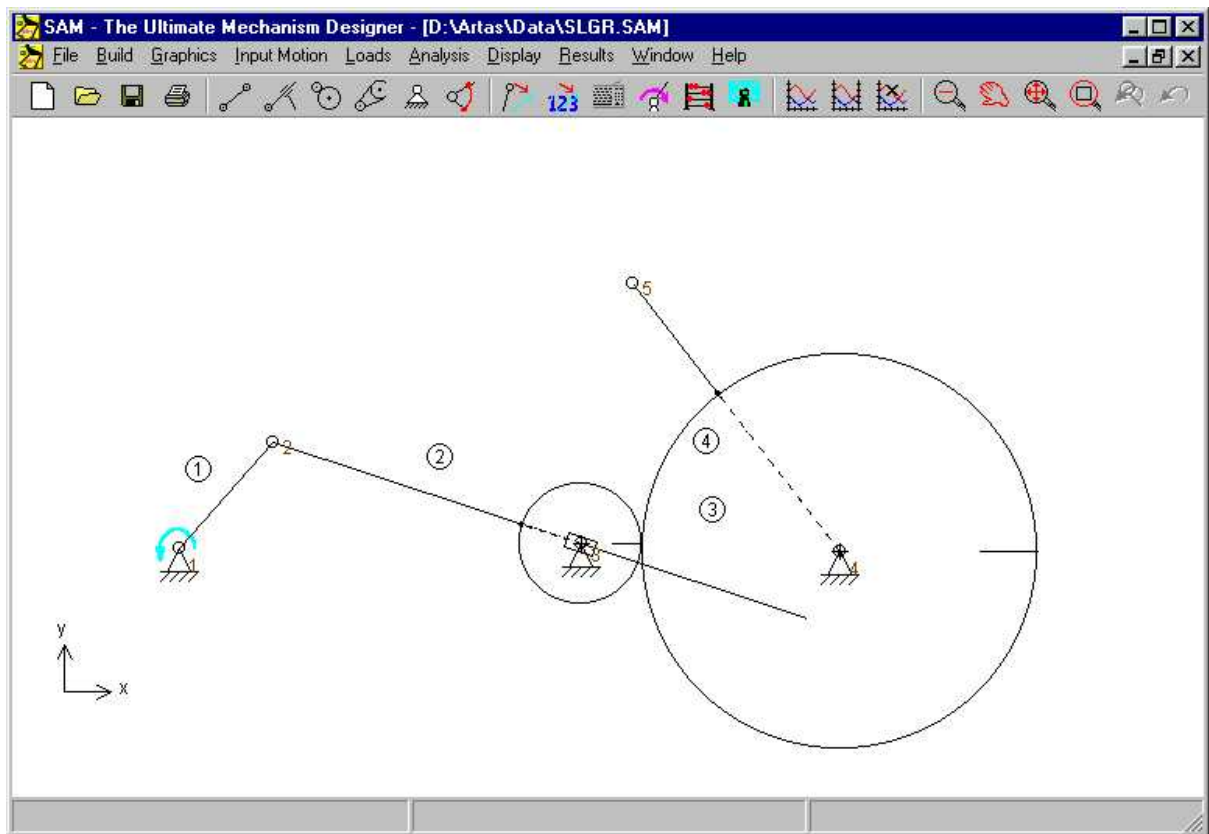
Quick-Return Mechanism (QKRT.SAM)



Displacement and velocity of node 5 as function of time

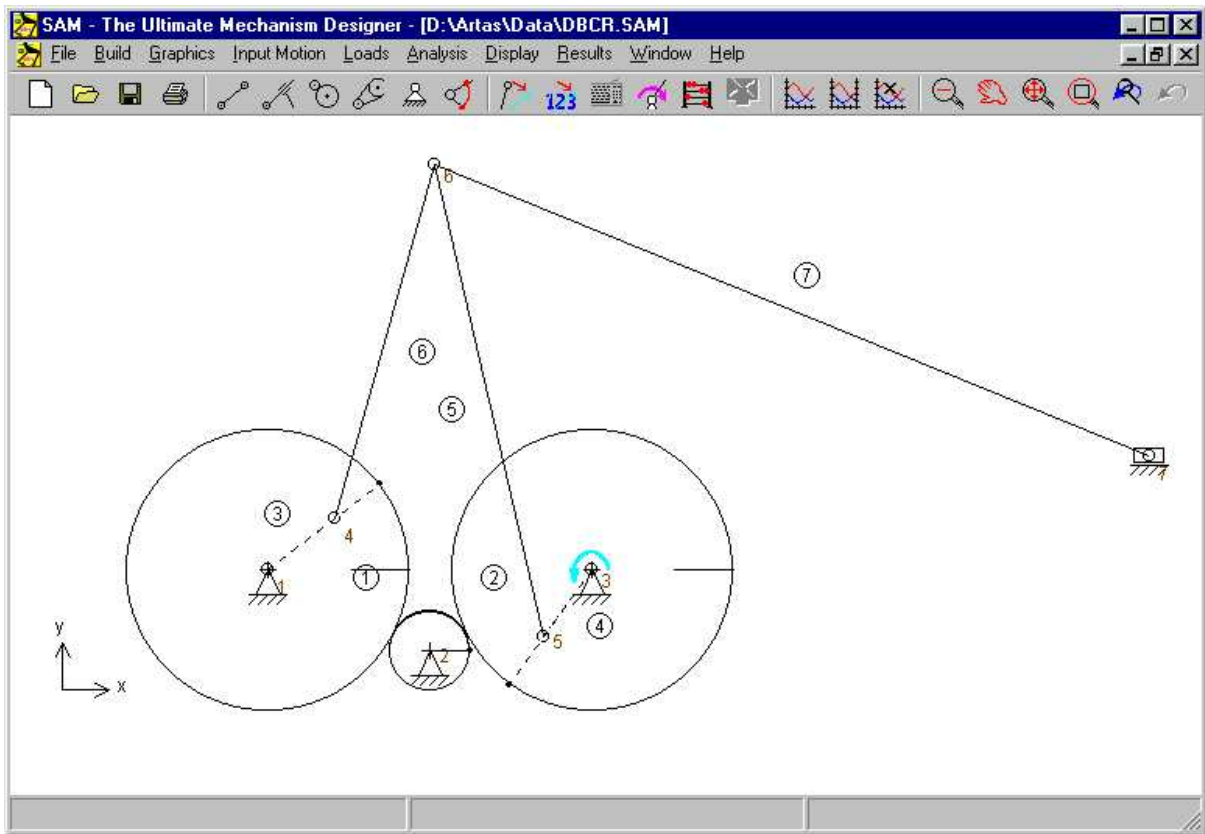


### 8.1.3 Slider Driven Gear Transmission



Slider Driven Gear Transmission (SLGR.SAM)

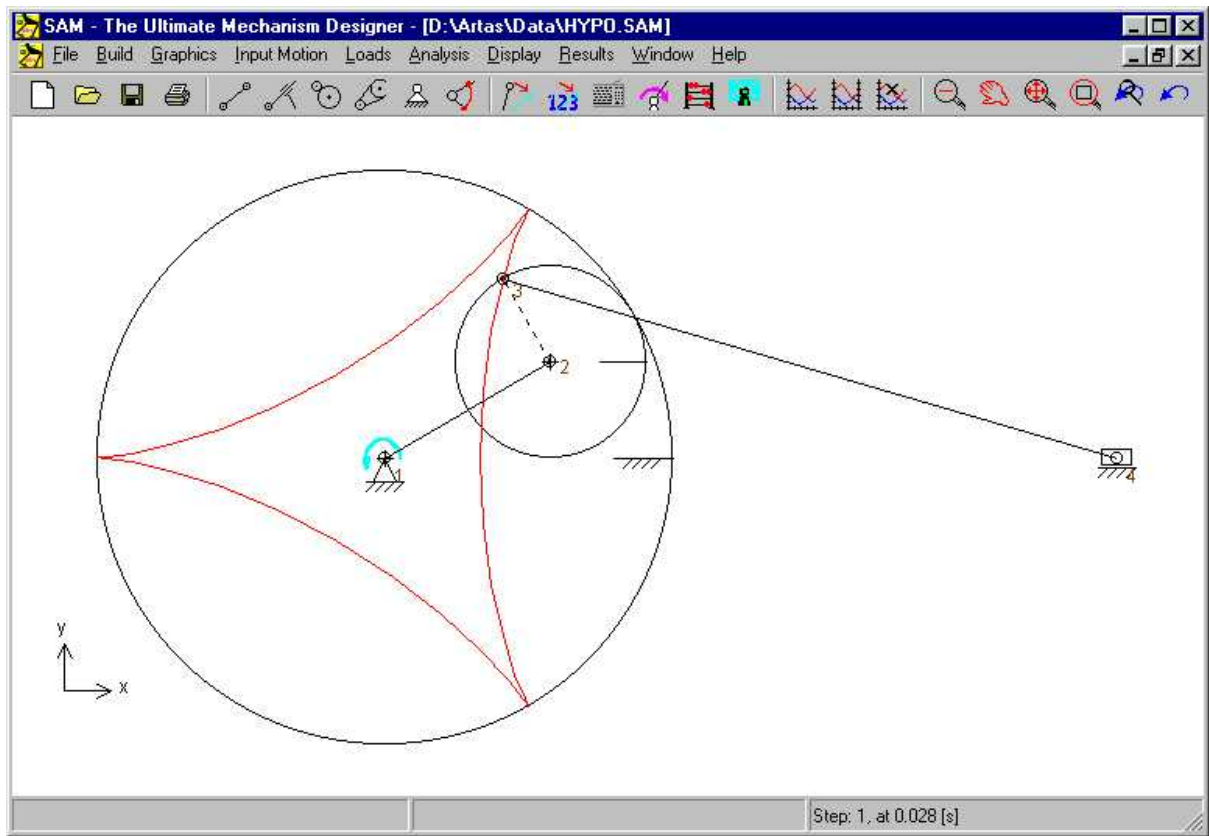
### 8.1.4 Double Crank Mechanism



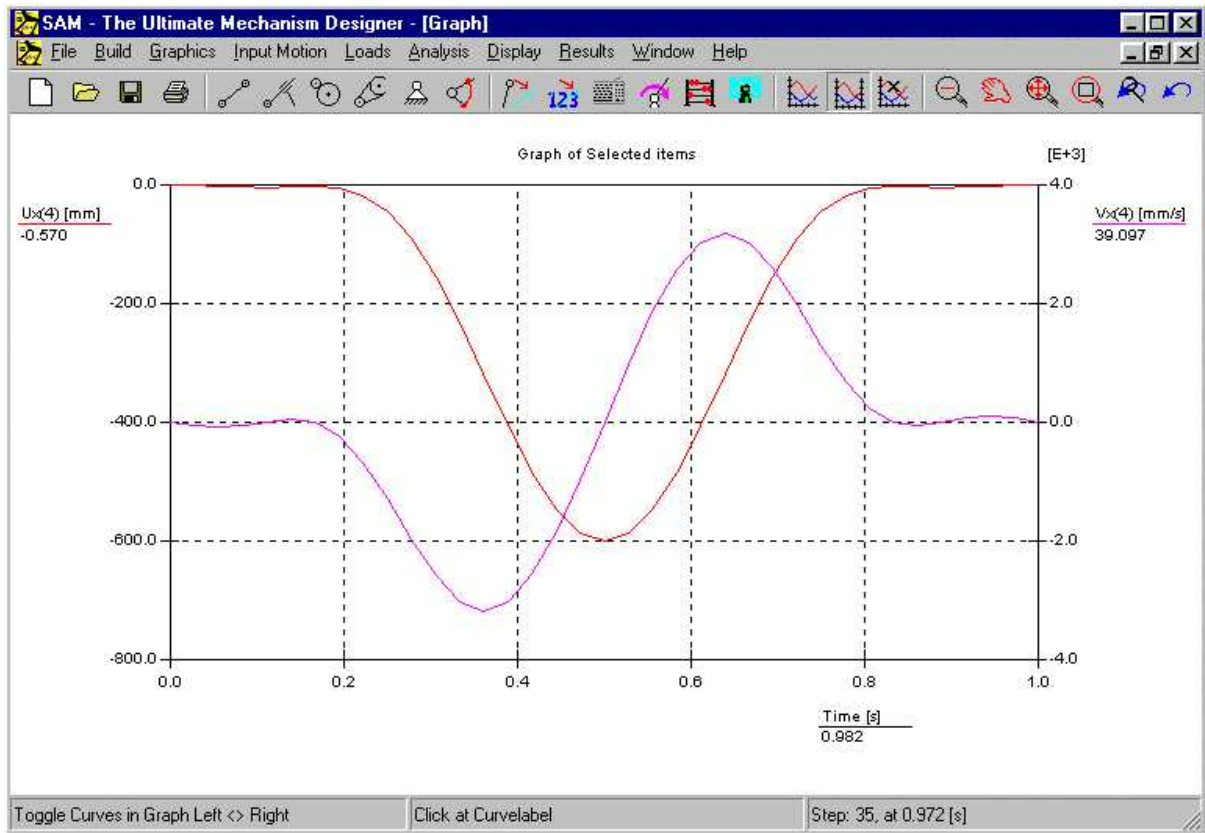
Double Crank Mechanism (DBCR.SAM)

### 8.1.5 Planetary Mechanism (Hypo-Cyclic)

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



Hypocyclic Mechanism (HYPO.SAM)

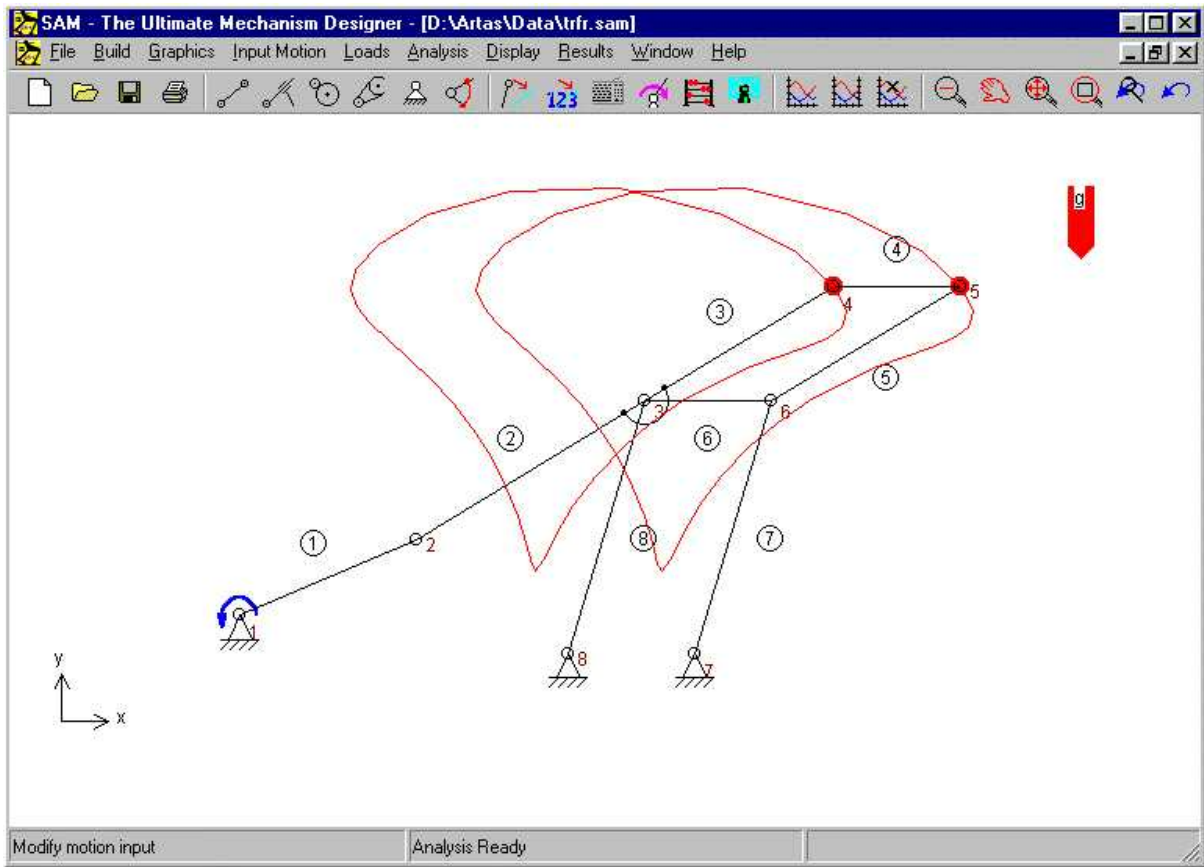


Output displacement and velocity of hypo cyclic mechanism

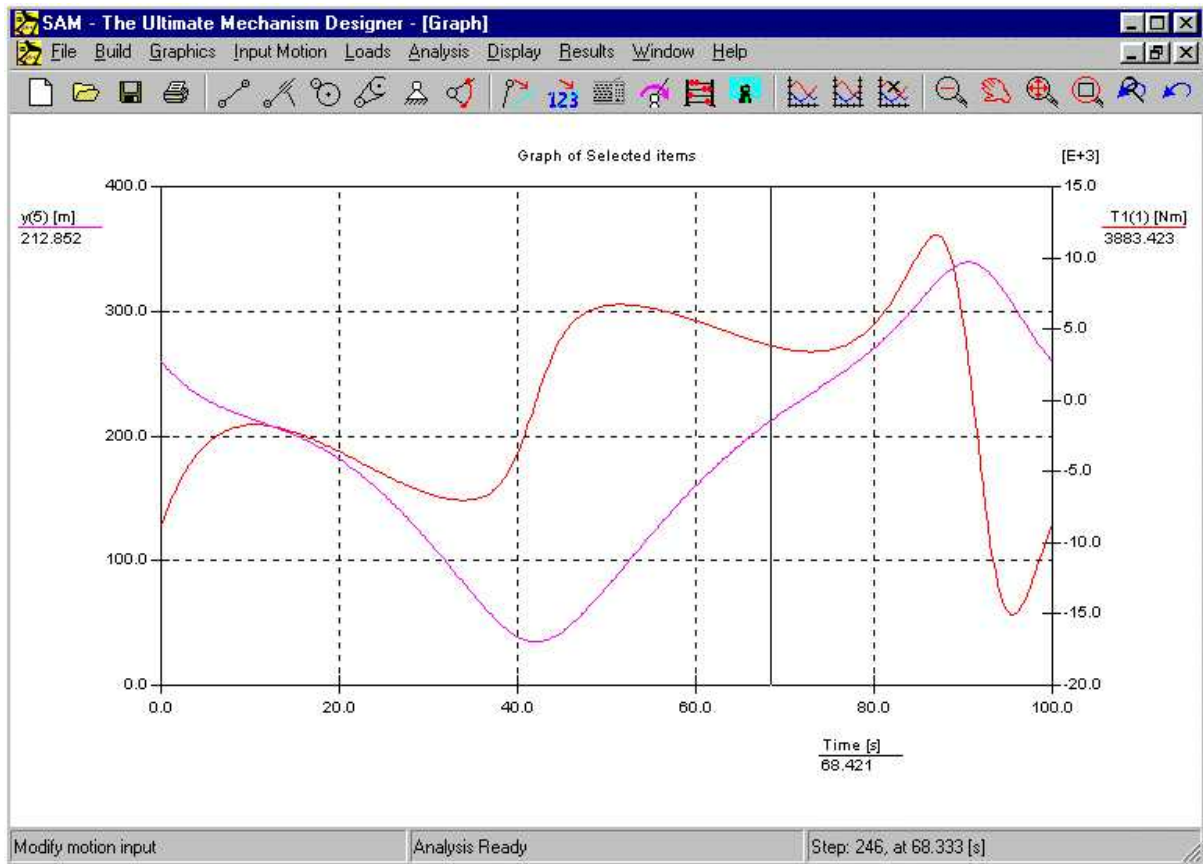
## 8.2 Force Analysis

### 8.2.1 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.



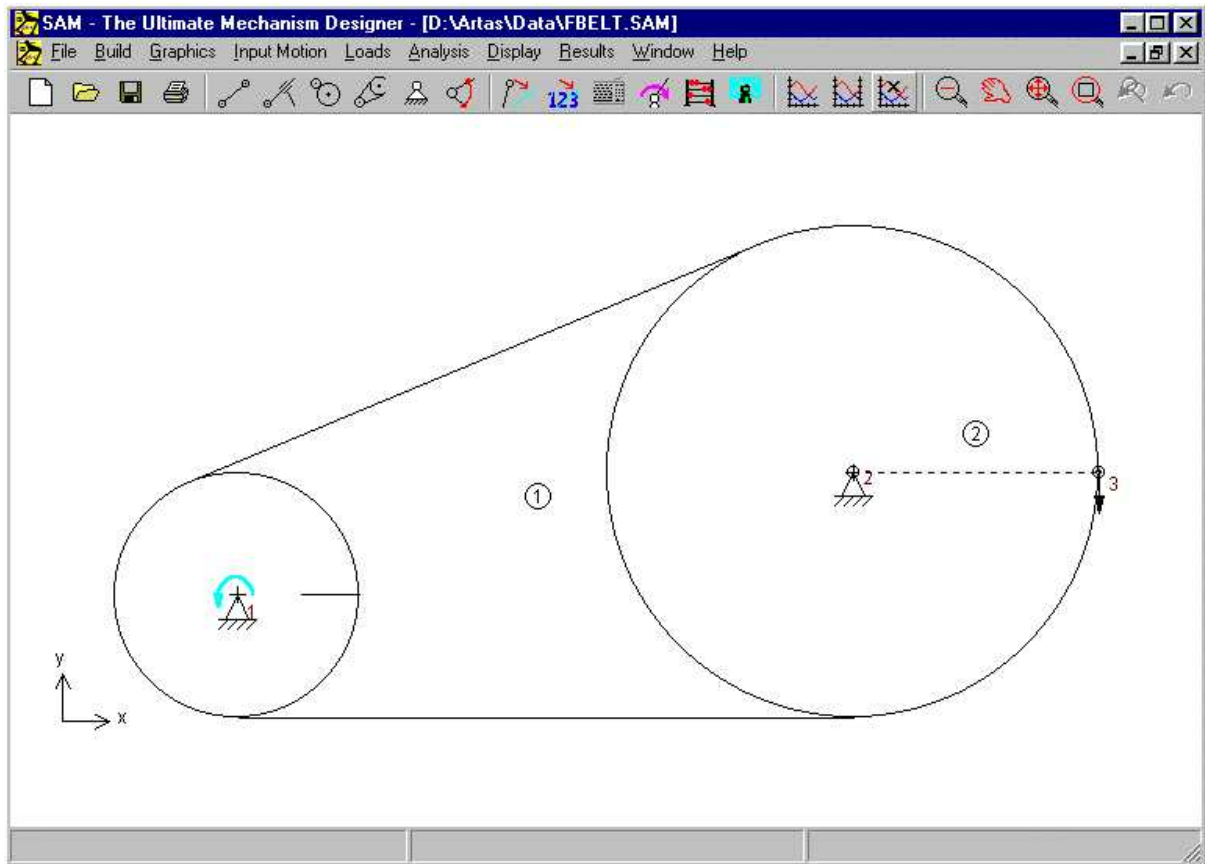
Transfer mechanism incl. path of transfer linkage (TRFR.SAM)



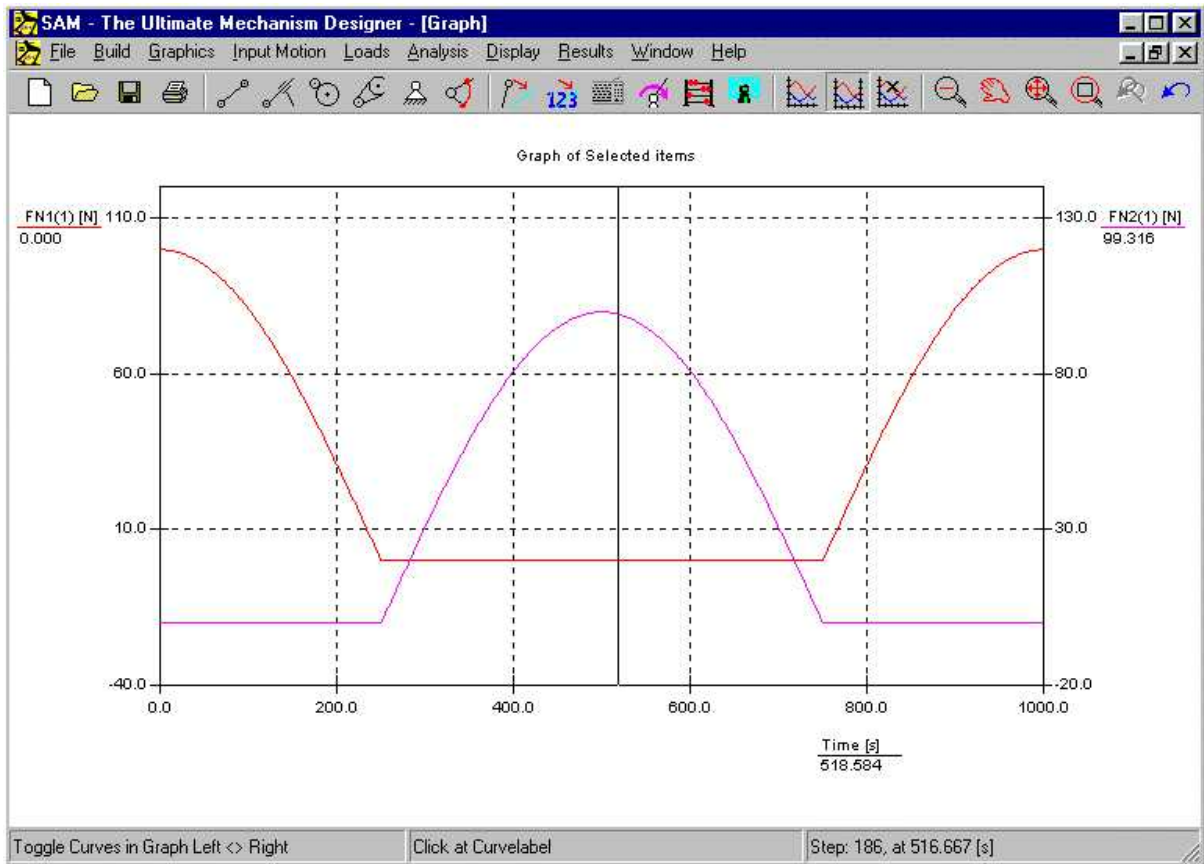
Vertical position transfer linkage & static driving torque

## 8.2.2 Belt drive with external force

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



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



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

## 8.3 Optimization

### 8.3.1 Driving Torque Minimization

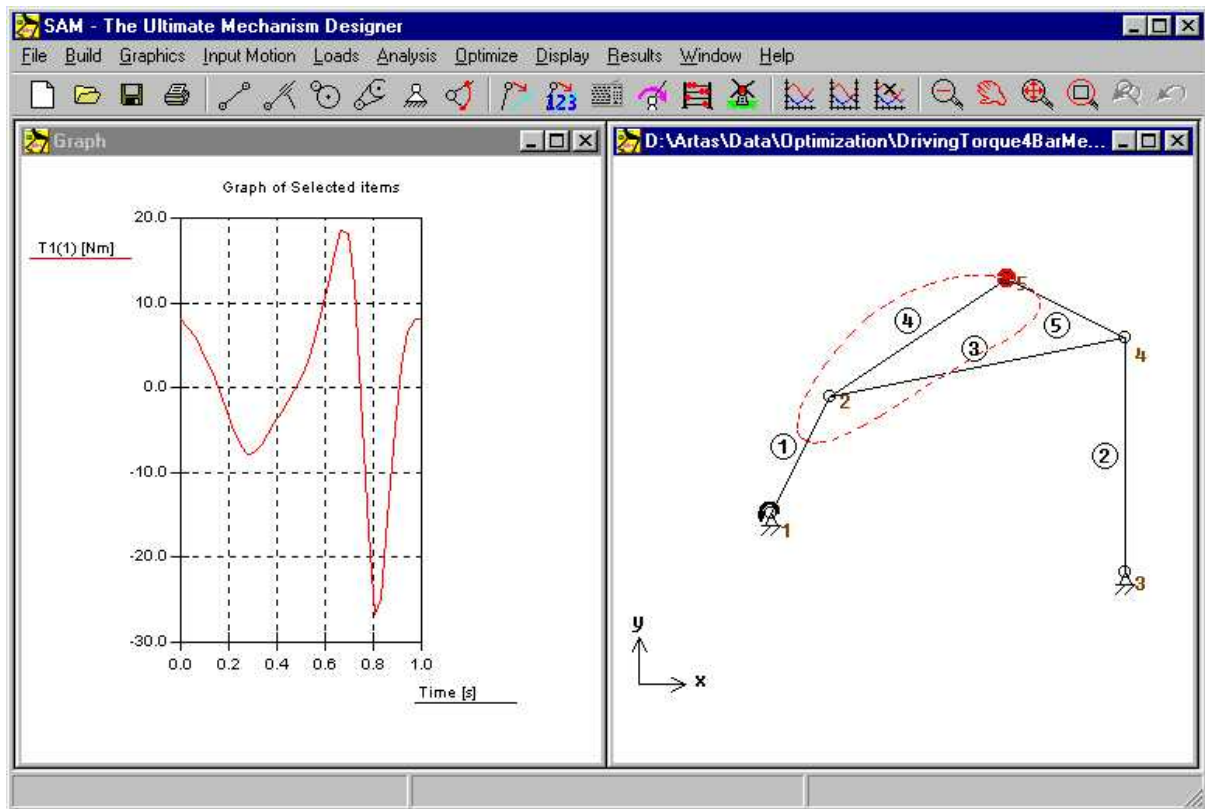
As a first example, the reduction of the driving torque of a 4-bar mechanism with mass in the coupler point will be demonstrated. This is achieved by the addition of one single mass in the coupler plane. The mechanism is driven by a constant angular velocity at the crank and gravity effects are excluded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

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

#### Reference Mechanism

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





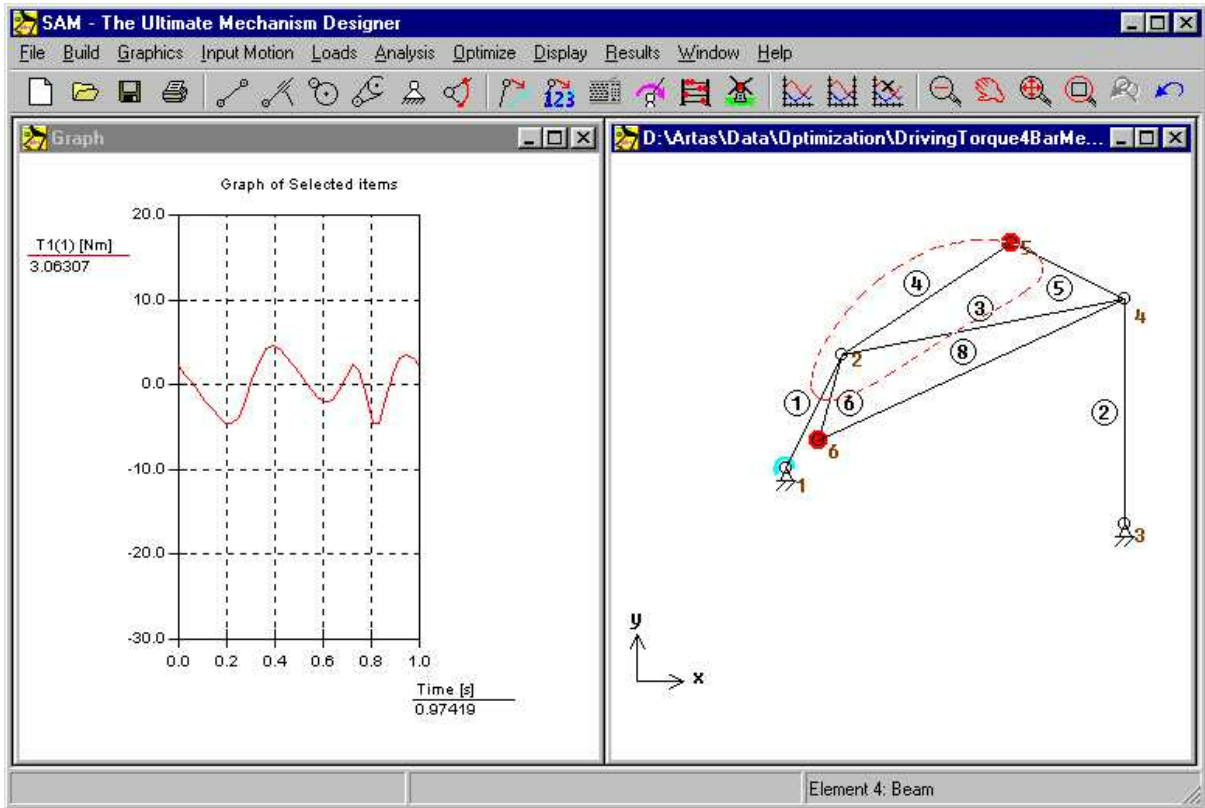
Reference Mechanism (a. Driving Torque b. Mechanism and path of coupler point)

### Optimization Parameter Range

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

### Optimized Mechanism

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



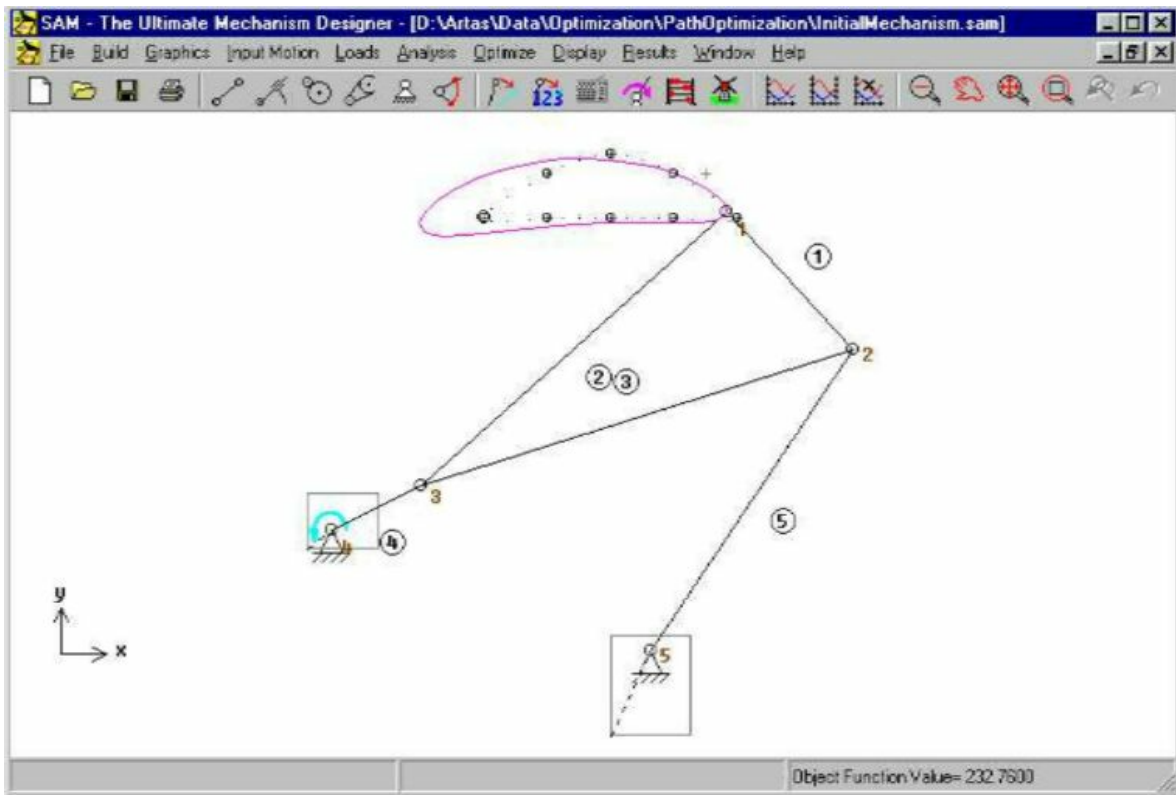
**Optimized Mechanism (a. Driving Torque b. Mechanism + compensating mass)**

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

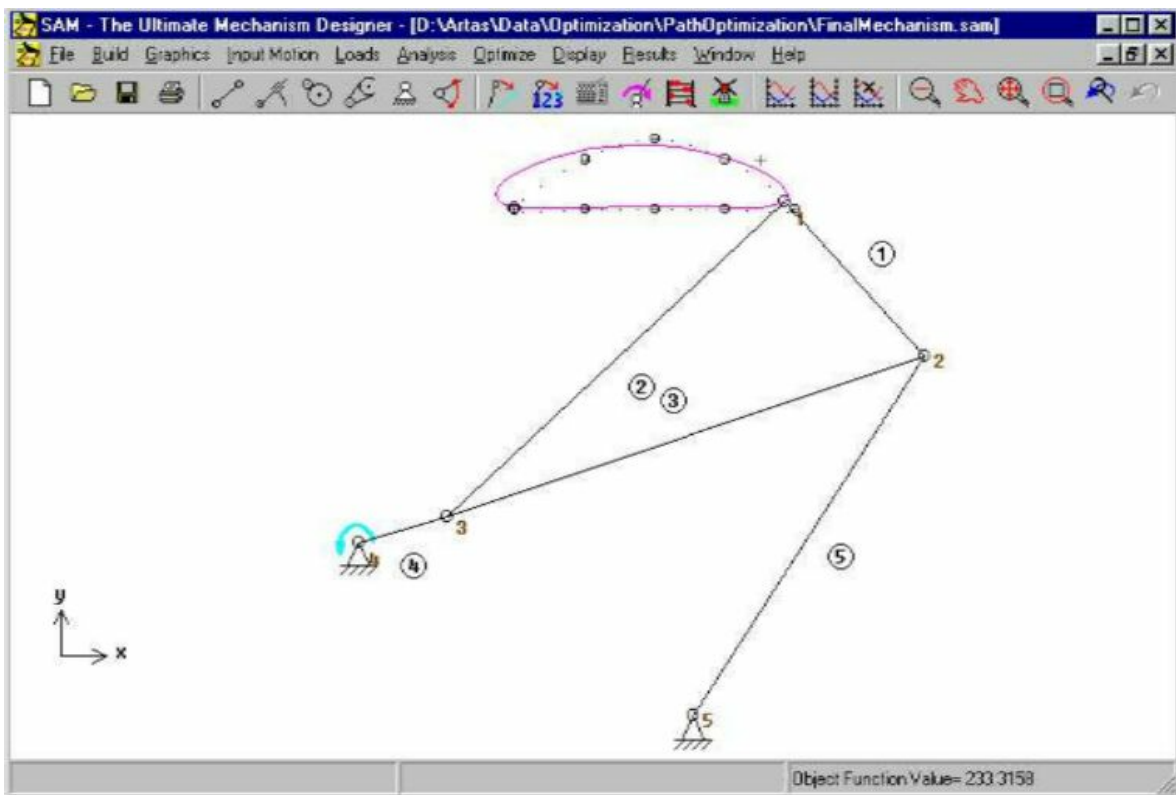
### 8.3.2 Optimization of trajectory

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

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



Manually chosen 4-bar mechanism



### Optimized 4-bar mechanism

## 8.3.3 Constrained Optimization

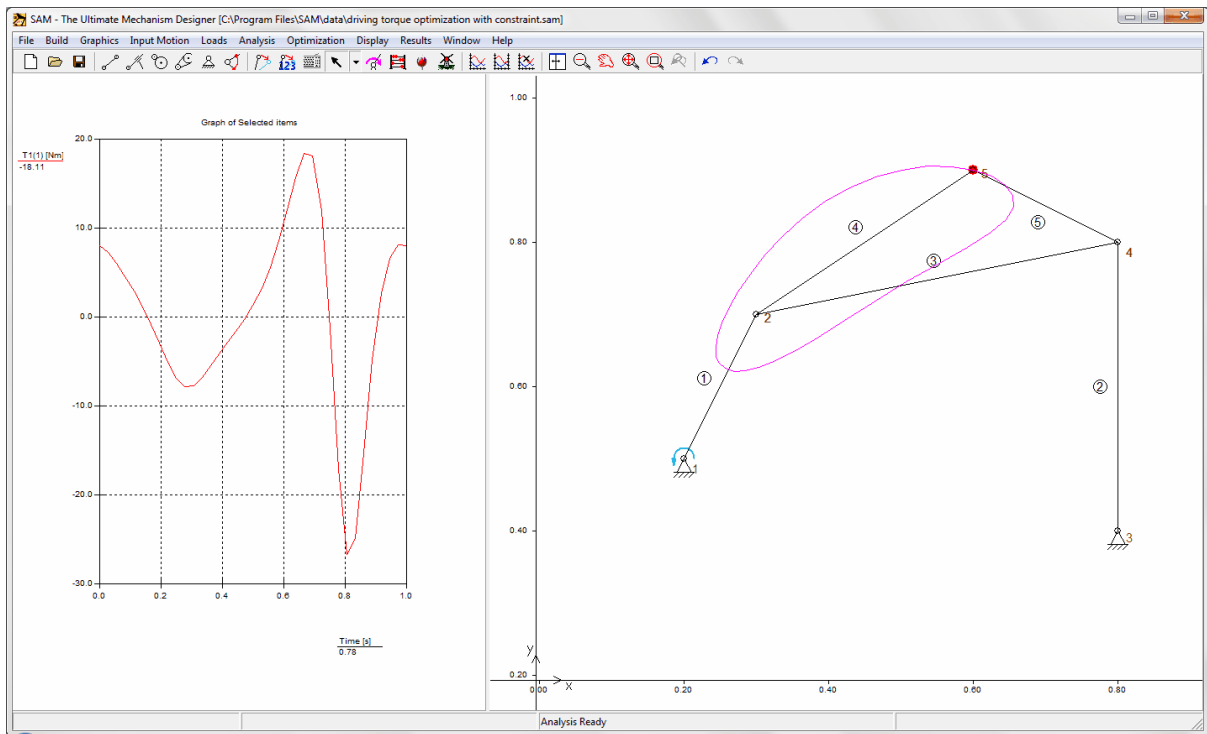
In this example we will show constrained optimization by adding a penalty function to the original cost function in order to incorporate the constraint.

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.

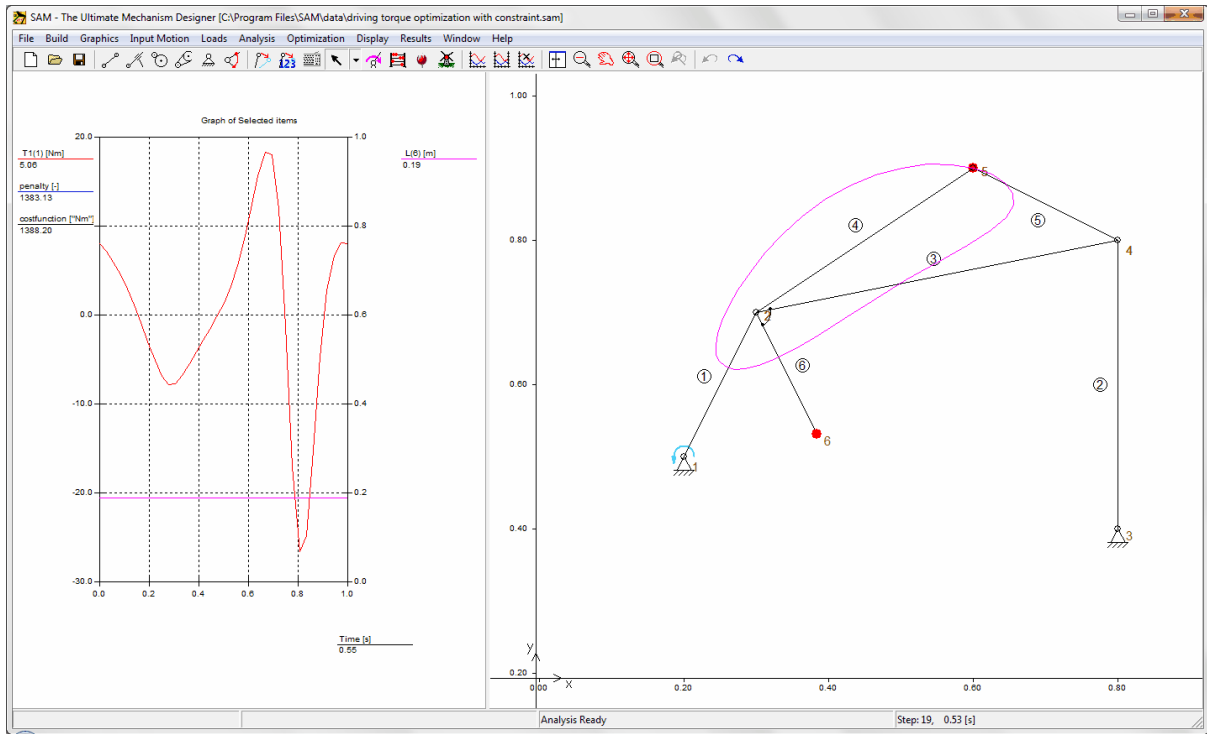
The optimization algorithm must search for the optimal value and location of the compensating mass. In this example, we want to add the extra constraint, that the length of the extra element, that is added to support the compensating mass, should not be more than 0.05m.

### 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.

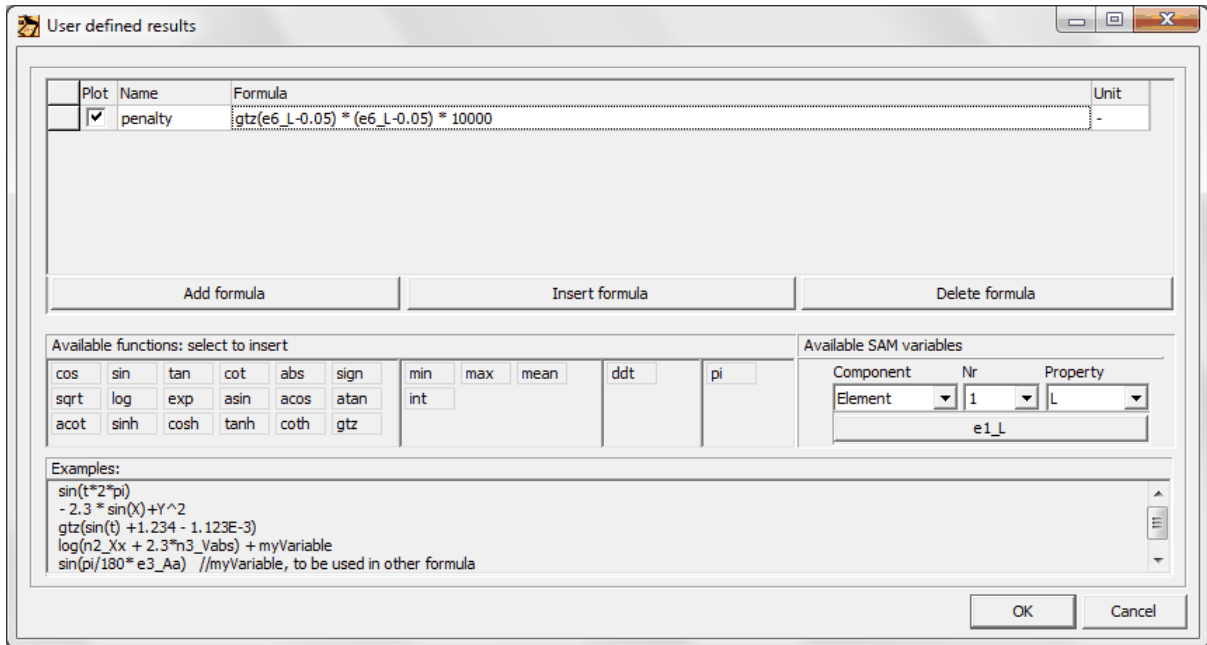


The next step is to add an extra element (nr. 6) at the end of which the compensation mass is located via element properties. The optimization module can't handle optimization of an individual mass and therefore the mass has to be added via element properties.



**Optimization**

In order to incorporate the constraint a user defined variable needs to be created via RESULTS / USER DEFINED ...

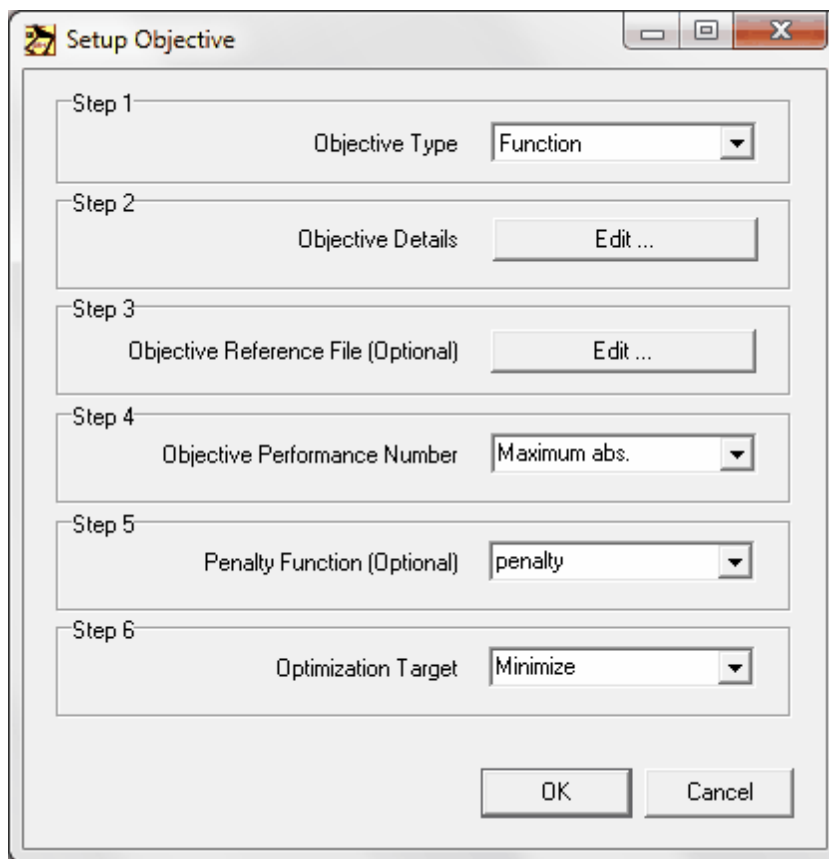


A penalty is defined in case the length of element 6 (SAM variable: e6\_L) is larger than 0.05m. The

penalty is based on the Heaviside or *Greater Than Zero* function  $GTZ(x)$ , which yields an output of 1 if  $x \geq 0$  and an output of 0 if  $x < 0$ . In order to achieve a larger penalty for larger deviations from the target the outcome is multiplied by the actual deviation. Finally a weighting function of 10000 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.

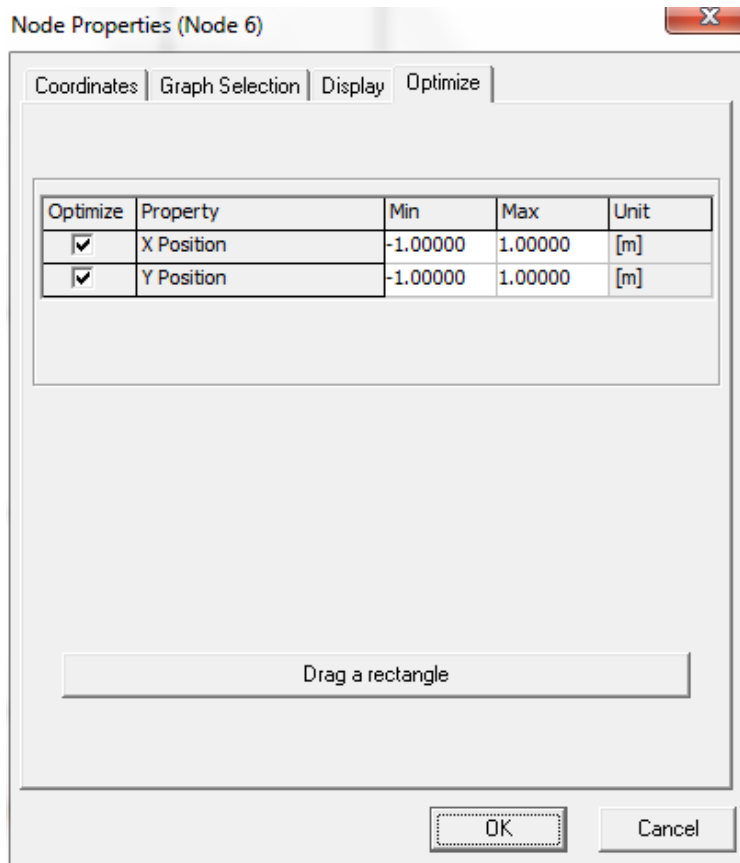
### Optimization Objective

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.

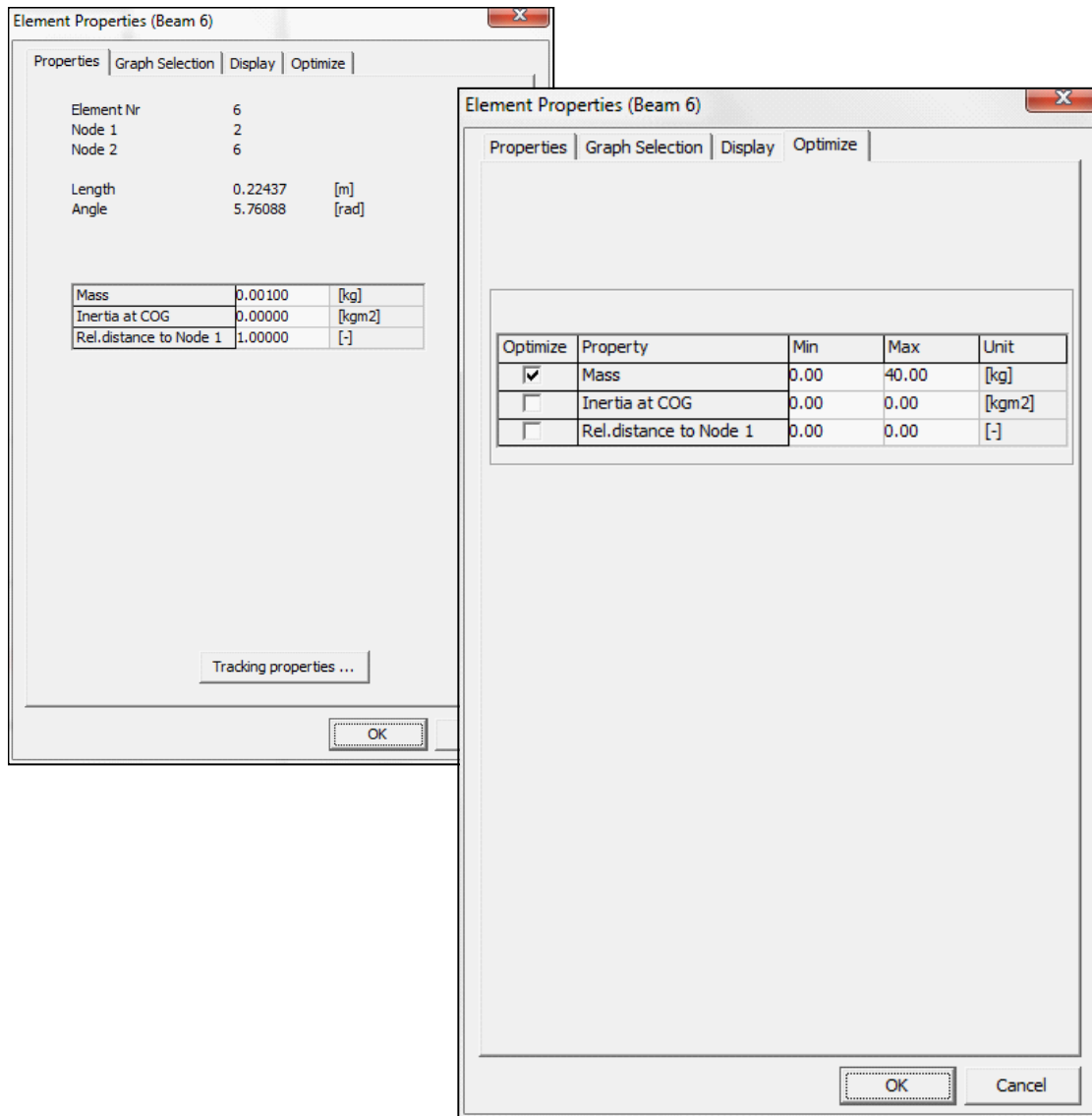


### Optimization Parameter Range

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).



The search region for the mass of element 6 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 6 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).



### Optimization Options

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

#### Modus

- Choosing AUTOMATIC is the easiest way to do an optimization.
- The checkbox "Optimize all solutions from global exploration" indicates that after the global exploration of the solution space (always done via evolutionary algorithm), ALL best results of the exploration are further improved via a local optimization. After the global exploration a pool of best results exists and the size of that pool of results is equal to the Population Size defined in the Evolutionary Settings. So in this case 300 evolutionary (or genetic) experiments are done based on a population size of 30 "members". At the end a pool of 30 best members or solutions exist. If the checkbox is checked,



all these 30 best results are further optimized via local optimization. If the checkbox is NOT checked, only the best solution found in the global exploration is further improved via local optimization.

#### Local Optimization

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

#### Evolutionary Setting

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

#### Simplex Setting

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

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

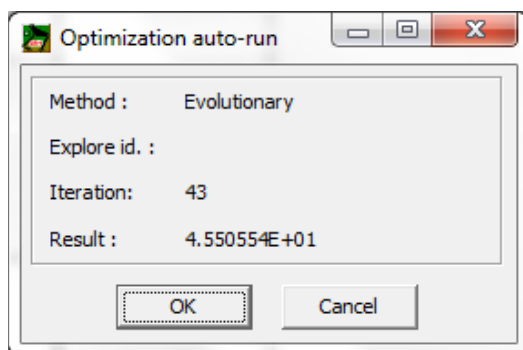
If the process takes too long one can decide to do less experiments, choose a smaller population size (though it should always be at least 20-30 to give the evolutionary algorithm a chance), do less iterations in the local optimization process or decide NOT to further optimize all results from the global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

### Optimization Run

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

#### Global Exploration

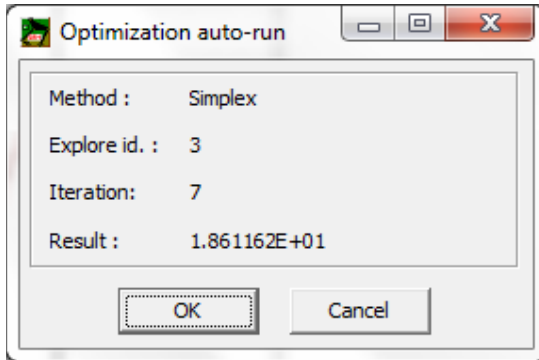
The status window shows the process of the global exploration.



Method: The method for global exploration is always the Evolutionary Algorithm.  
 Explore id: Empty (has only meaning when doing the local optimization).  
 Iteration: Current iteration step in the evolutionary process  
 Result: Current value of the experiment.

### Local Optimization

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

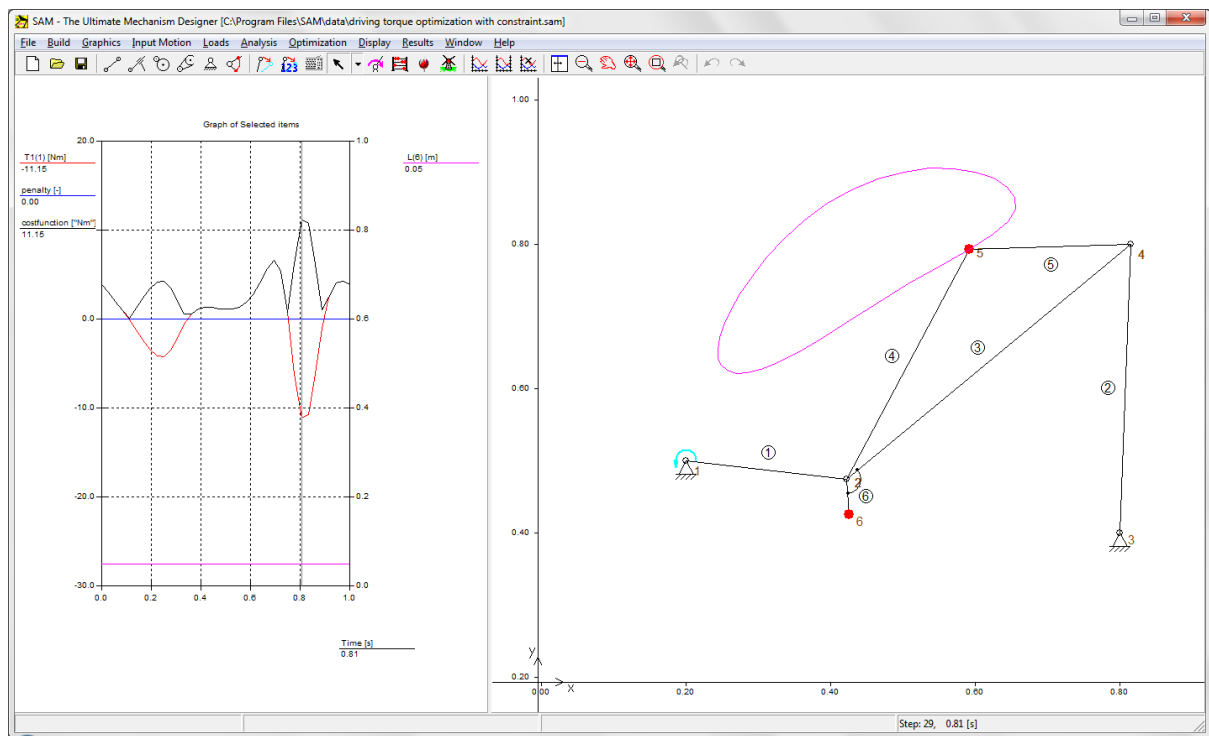


Method: Chosen method for local optimization  
Explore id: ID of the solution of the global exploration that is currently locally optimized.  
Iteration: Current iteration step in the local optimization process  
Result: Current value of the optimization goal.

### **Optimized Mechanism**

The value ( $m=38,73\text{kg}$ ) and location ( $x=0.28\text{m}$  and  $y=0.65\text{m}$ ) is derived by the optimization module. The corresponding maximum absolute value of the driving force in the optimized situation equals  $11.15\text{Nm}$ .

This is the best solution that could be found given the constraint that the length of the extra element 6 should be less than  $0.05\text{m}$ . As described in the application note "Driving Torque Optimization" the result without this extra constraint was significantly better ( $4.6\text{Nm}$ ).



### 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 11.15Nm.

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 application note "Driving Torque Optimization" the result without this extra constraint was significantly better (4.6Nm).



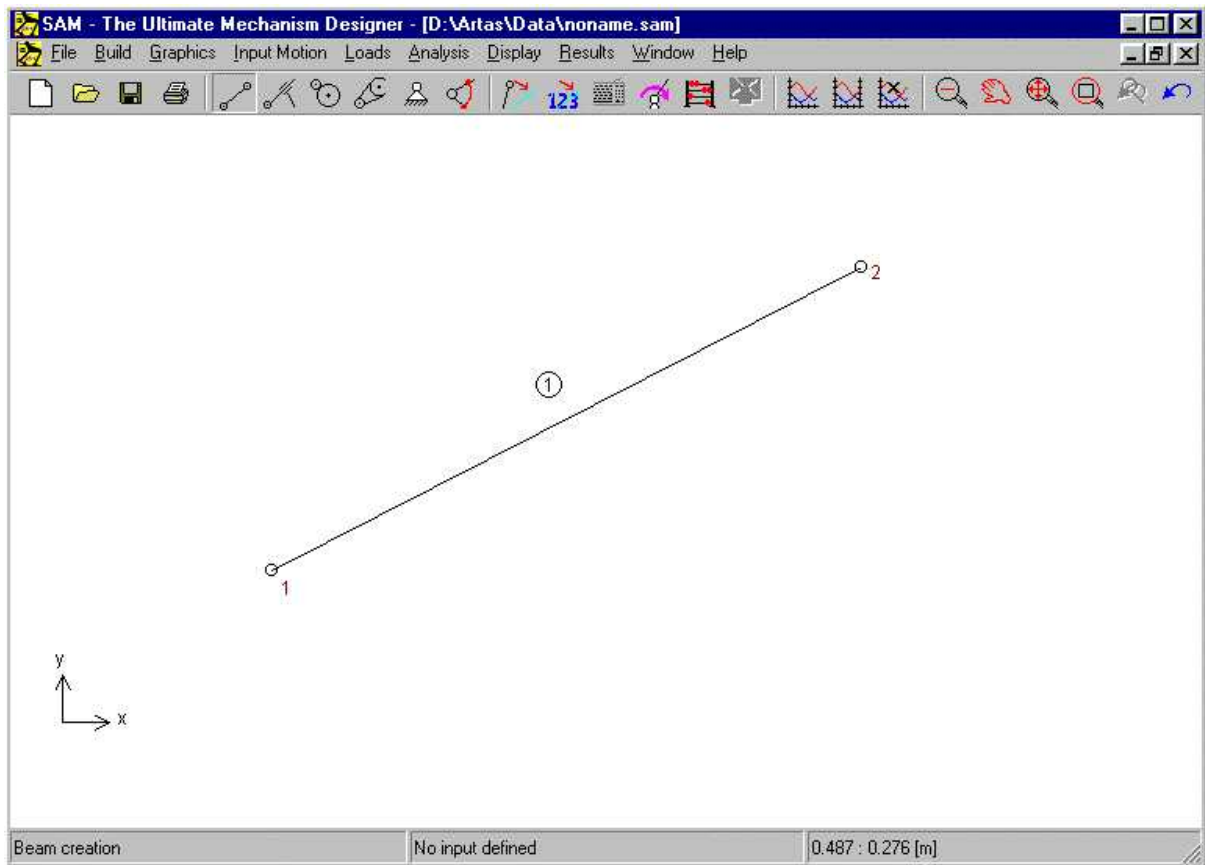
# Chapter

9

## 9 Elements

### 9.1 Beam

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

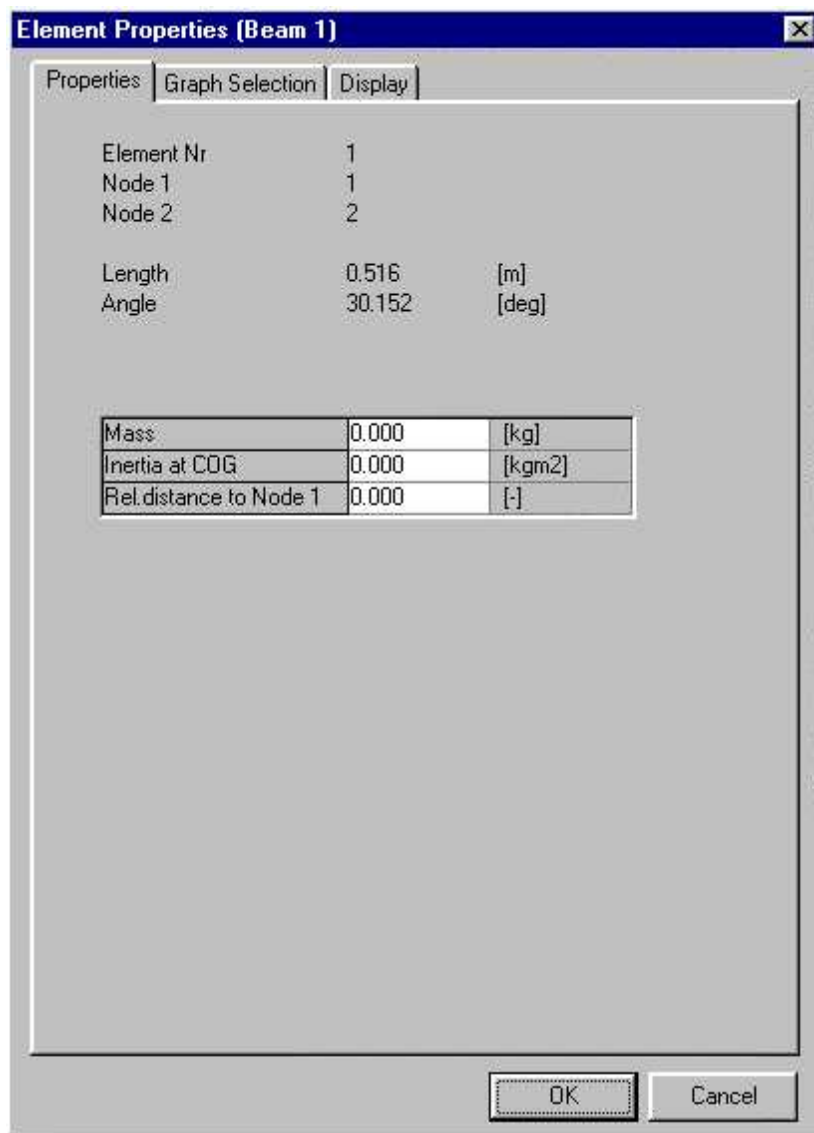


#### Beam element

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 w.r.t. center of gravity

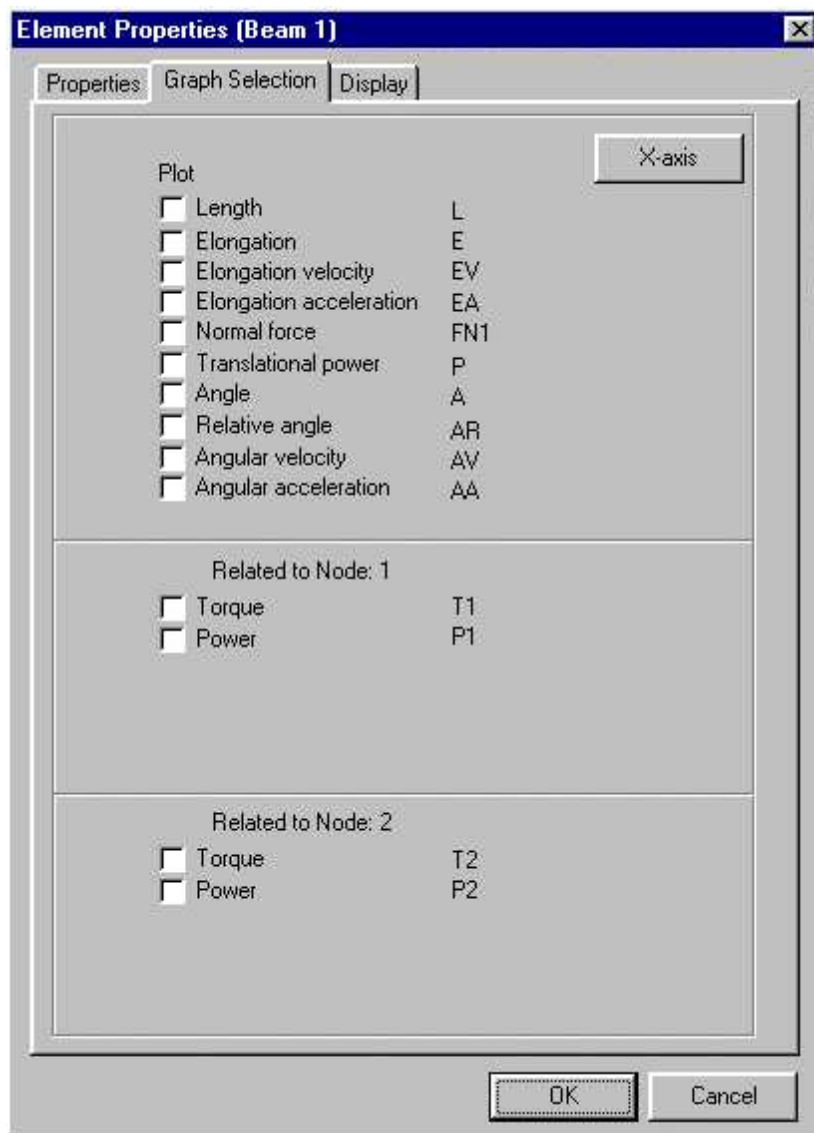
The center of gravity is defined as (distance with respect to first node)/ (length beam) and it ranges from 0-1. The location is therefore restricted to a position on a line that connects the two nodes and must be between these nodes.



**Properties of beam element**

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

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



**Calculable properties of beam element**

---

A positive force FN1 indicates a tension force, which - in case of flexibility - would result in an elongation of the beam.

---

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

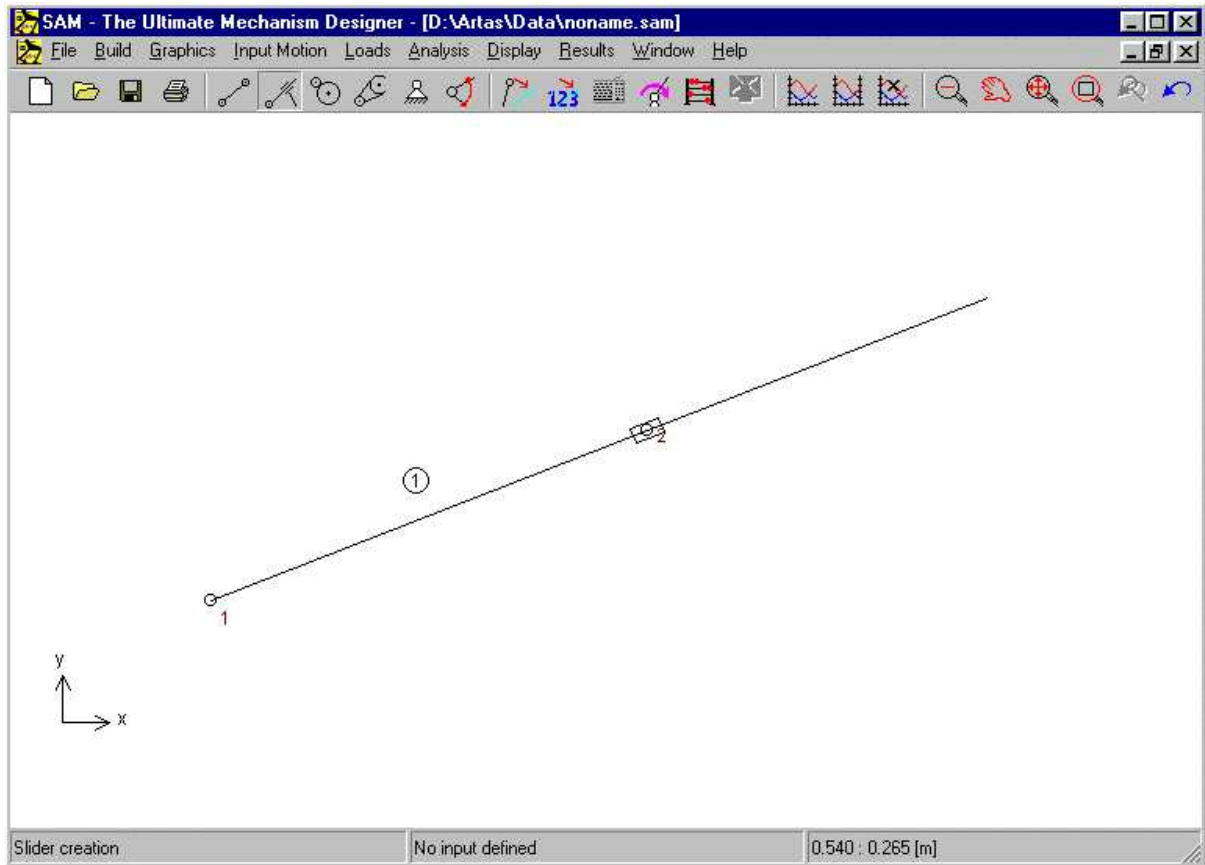
In the case of a linear actuator, which can be modeled as a beam element with prescribed elongation, these quantities have the anticipated physical meaning.

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



## 9.2 Slider

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

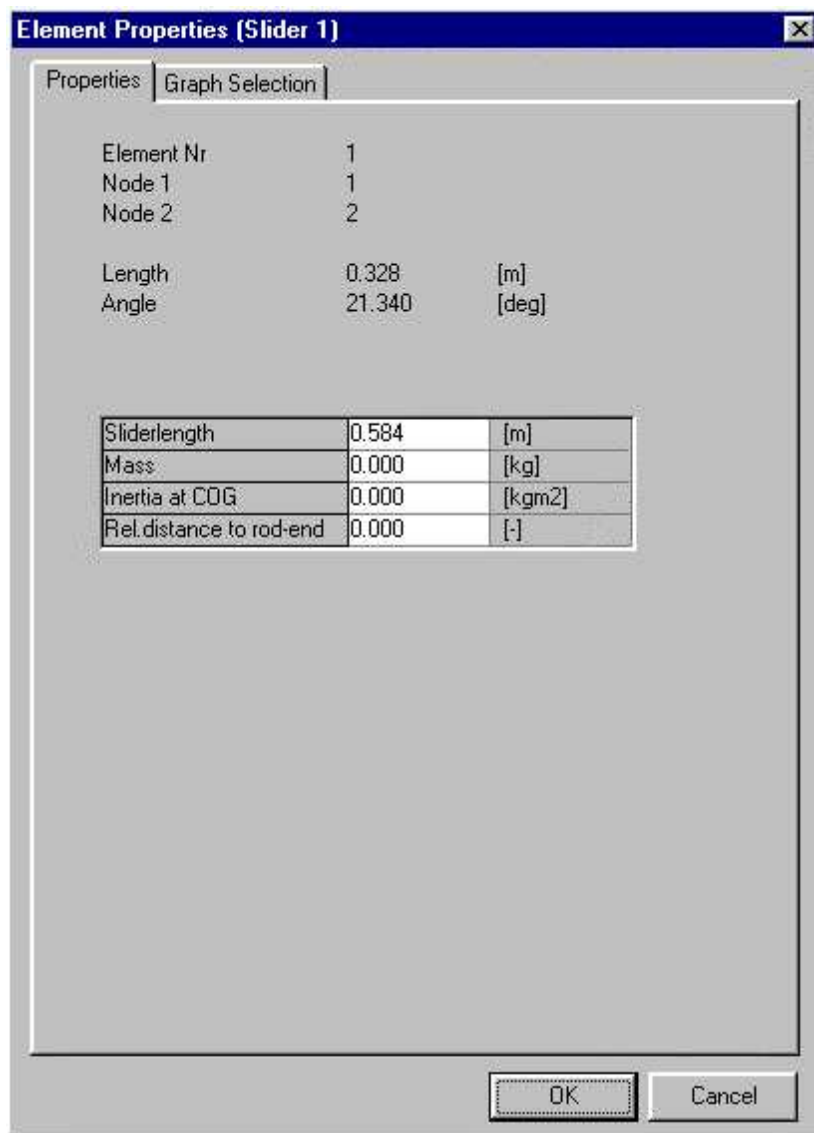


### Slider element

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

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

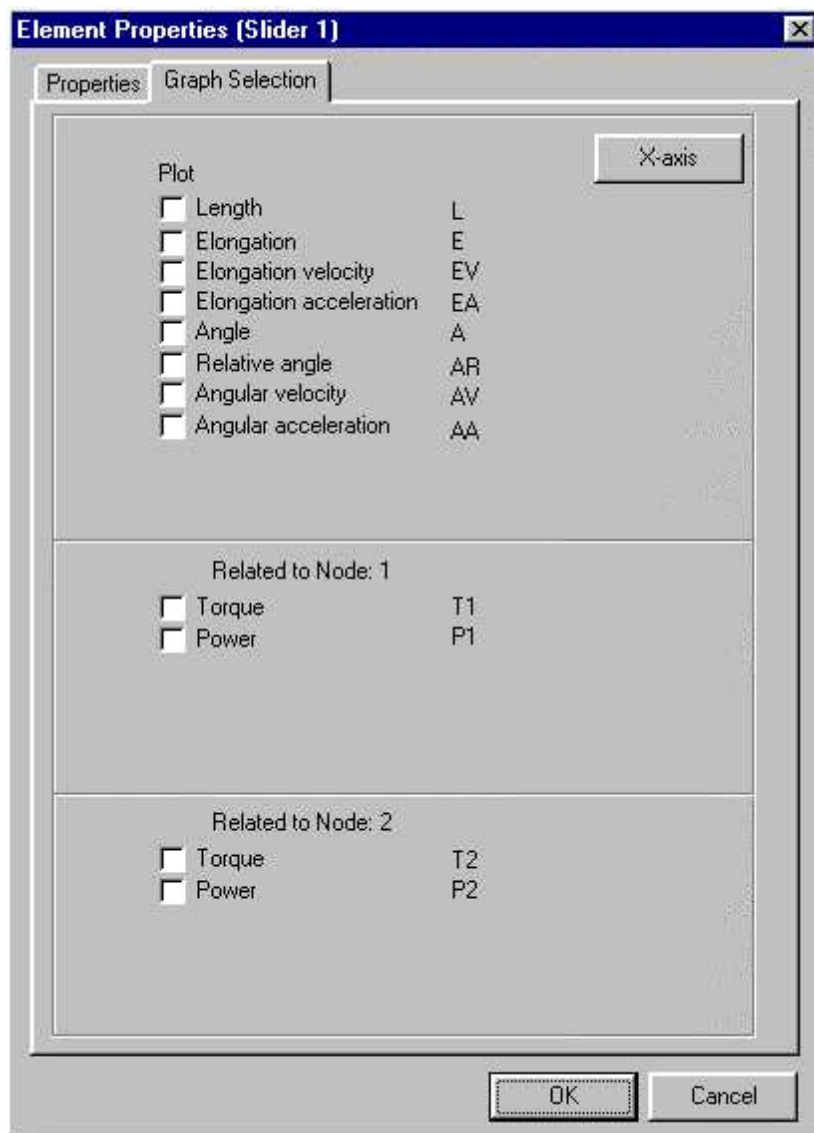
The center of gravity is defined as (distance w.r.t. first node) / (length element) and it ranges from 0-1. The location is therefore restricted to a position on the slider.



**Properties of slider element**

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

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



Calculable properties of slider element

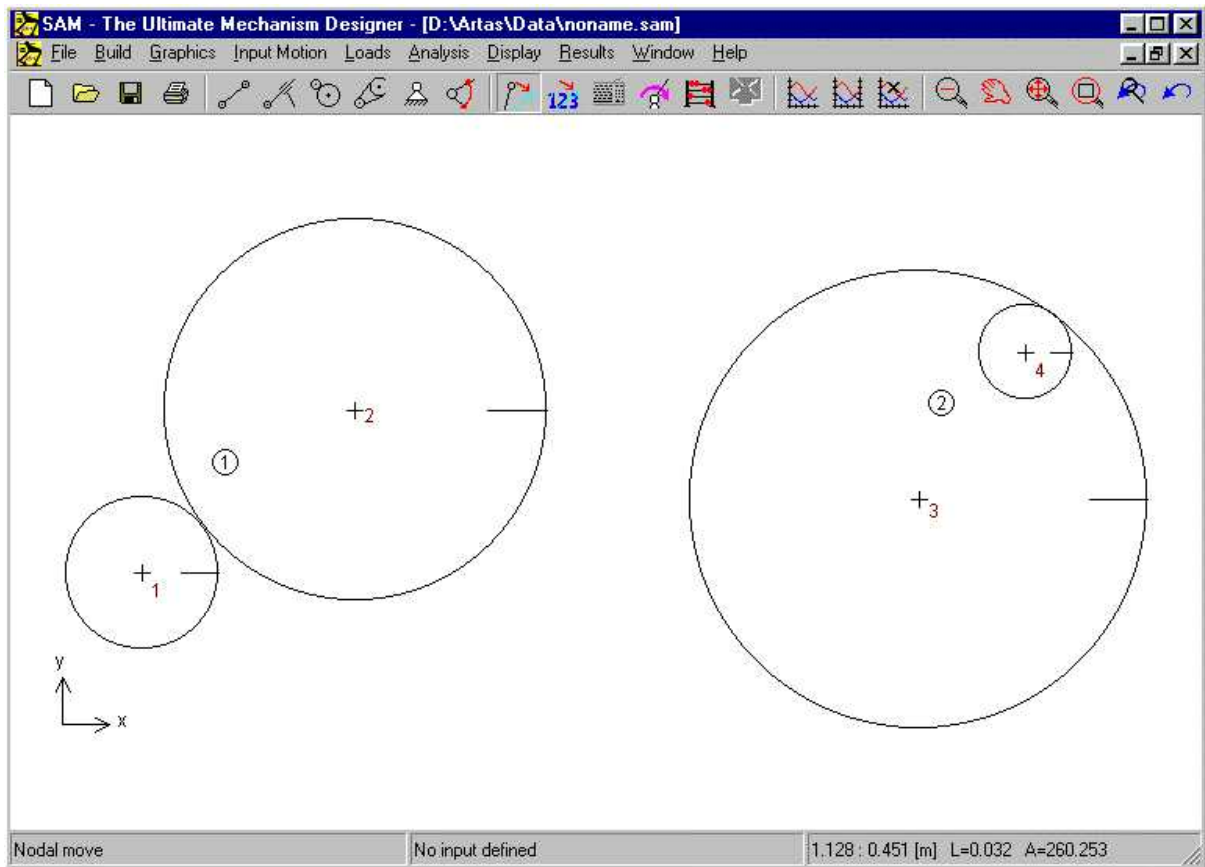
The output selection dialog of the slider is equal to that of a beam element, except that the item "normal force" and "normal power" is greyed. Since the elongation of a slider is not restricted, normal forces can not be transmitted and the corresponding power is naturally also equal to zero. The item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

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

### 9.3 Gear

The gear element is used to describe various gear configurations, ranging from standard gear configurations to gears with inner toothing and even planetary gear trains. It is important to note, that the

distance between the nodes of a gear element needs to be fixed, either by constraining the nodes or - in case of a planetary gear train - by defining a beam element between these nodes.



#### Gear element

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

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

**Element Properties (Gear 1)** [X]

Properties | Graph Selection

Element Nr 1  
 Node 1 1  
 Node 2 2

Nodal Distance 0.325 [m]

|                | Node 1 | Node 2 |                     |
|----------------|--------|--------|---------------------|
| Radius         | 0.093  | 0.232  | [m]                 |
| Mass           | 0.000  | 0.000  | [kg]                |
| Inertia at COG | 0.000  | 0.000  | [kgm <sup>2</sup> ] |

Internal Tooth

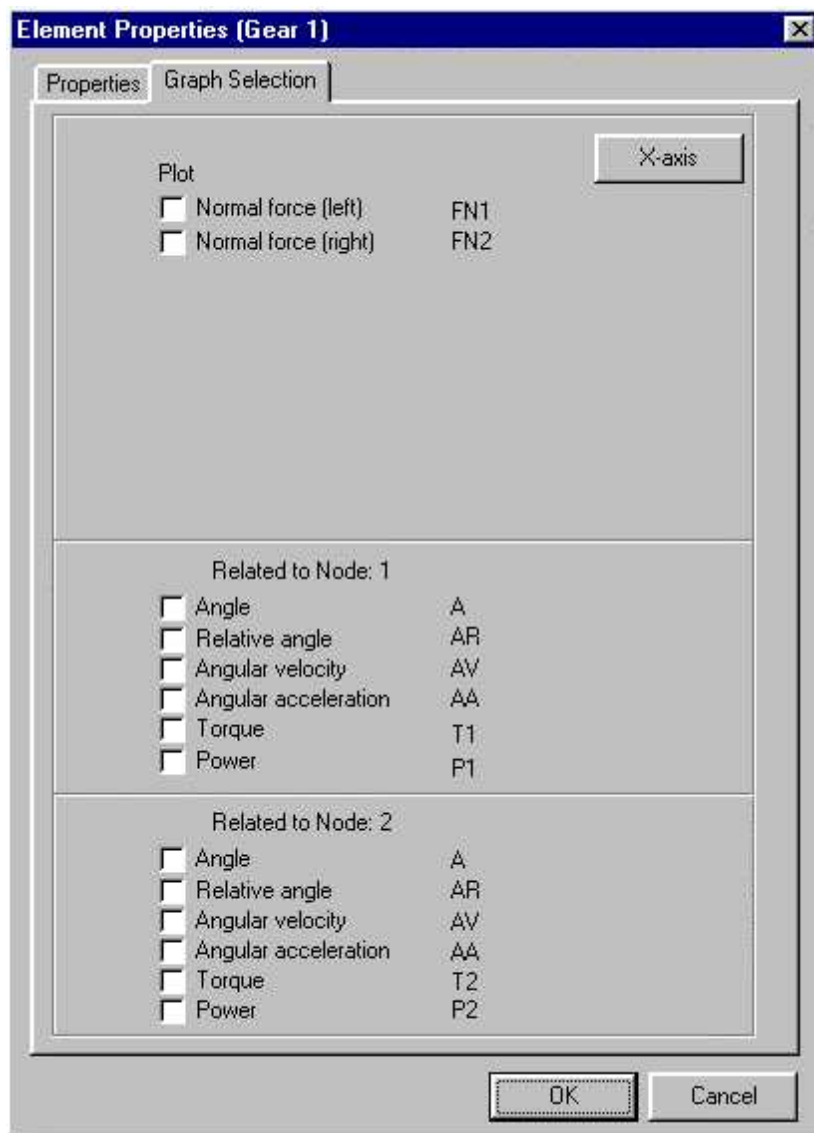
Pressure angle  [deg]

OK Cancel

#### Properties of gear element

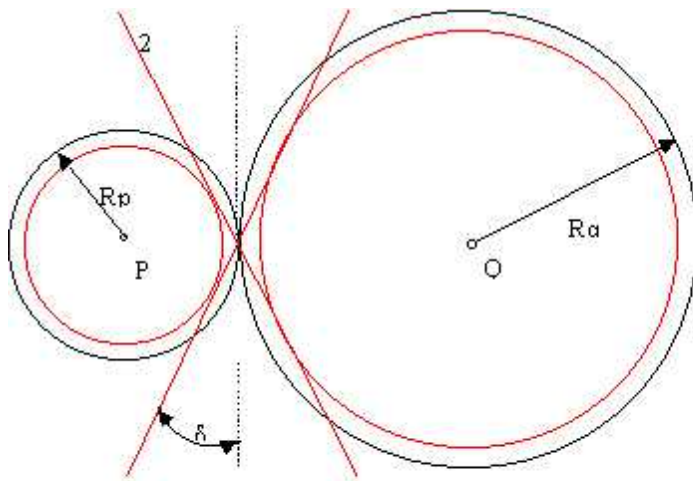
The following analysis results can be selected:

- 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.

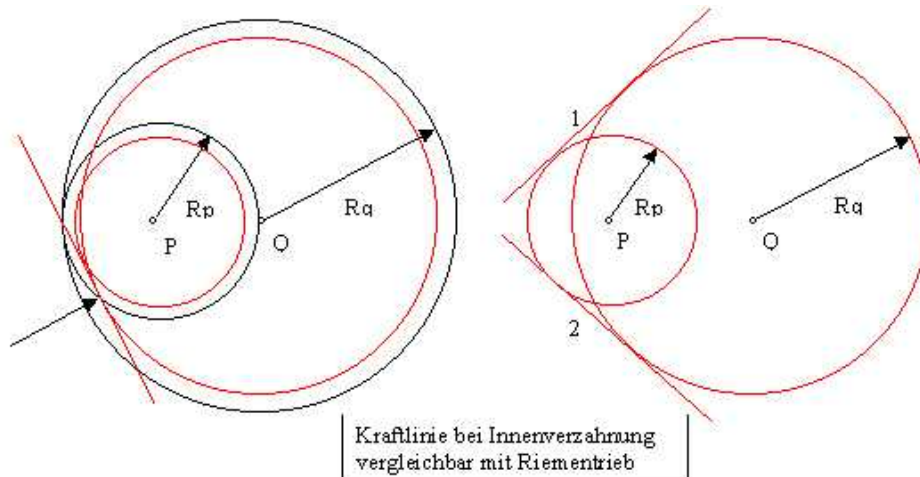


**Calculable properties of gear element**

In any gear transmission one can distinguish two possible force transmission directions. In the absence of pre-tension one of these two forces is equal to zero, whereas the other one is NEGATIVE (= pressure force). In order to understand the results of the force analysis (internal element forces FN1 and FN2) it is essential to have a clear definition of these two contactlines. Contactline 1 of a gear element which is defined from node P to node Q always comes into contact with wheel Q on the left-hand side of the line PQ (see example).



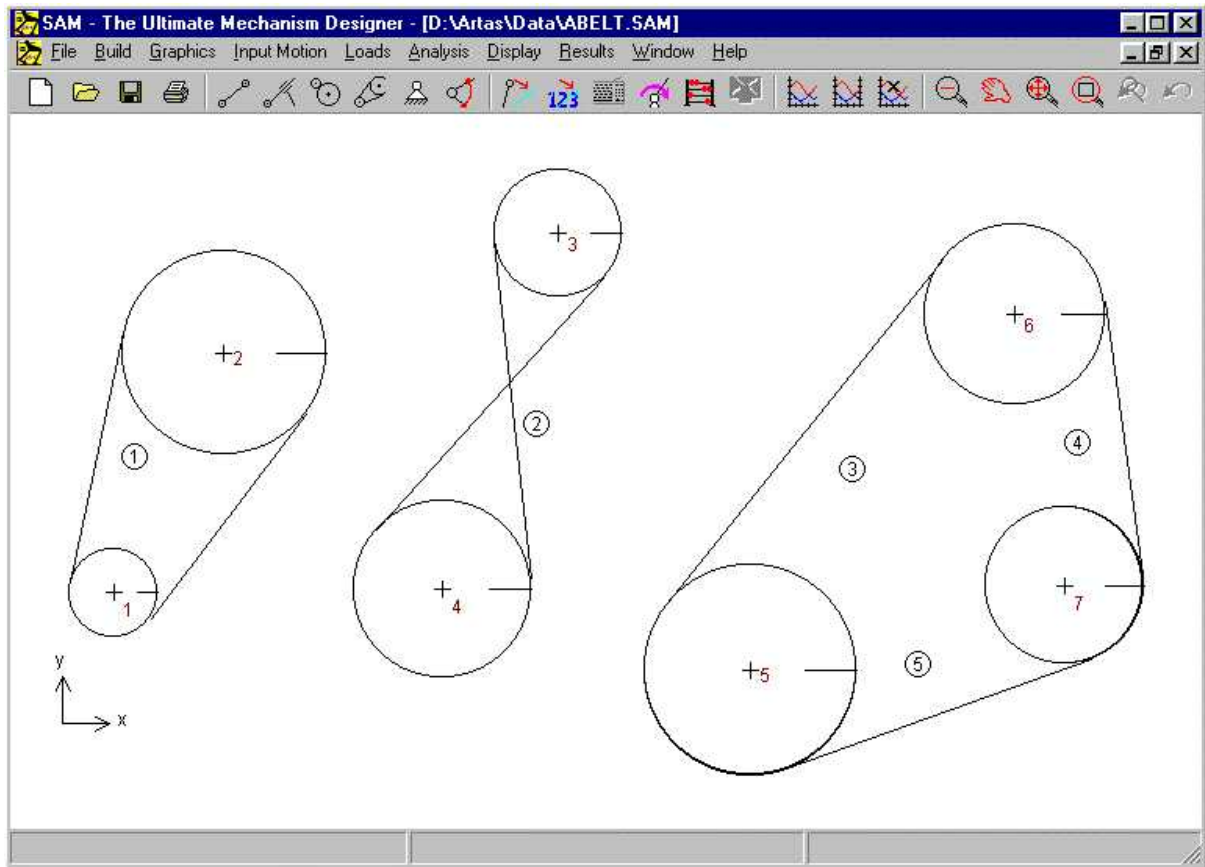
Definition of element forces FN1 and FN2 in a standard gear transmission



Definition of element forces FN1 and FN2 in a gear transmission with internal tooth

## 9.4 Belt

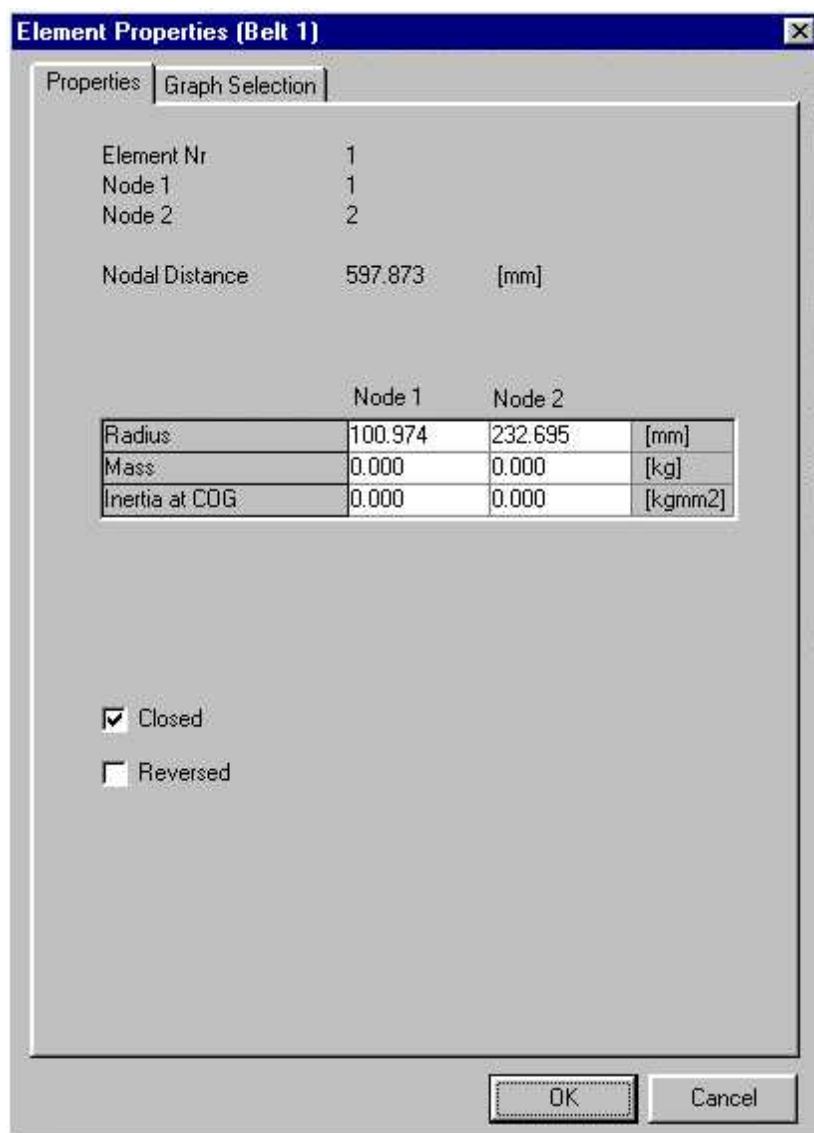
The belt element is used to describe various sort of belt configurations, ranging from standard belt to crossed belts or a belt that passes several wheels. It is important to note, that the distance between the nodes of a belt element needs to be fixed, either by constraining the nodes or - in case of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

**Belt element**

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

- radius
- mass
- mass moment of inertia

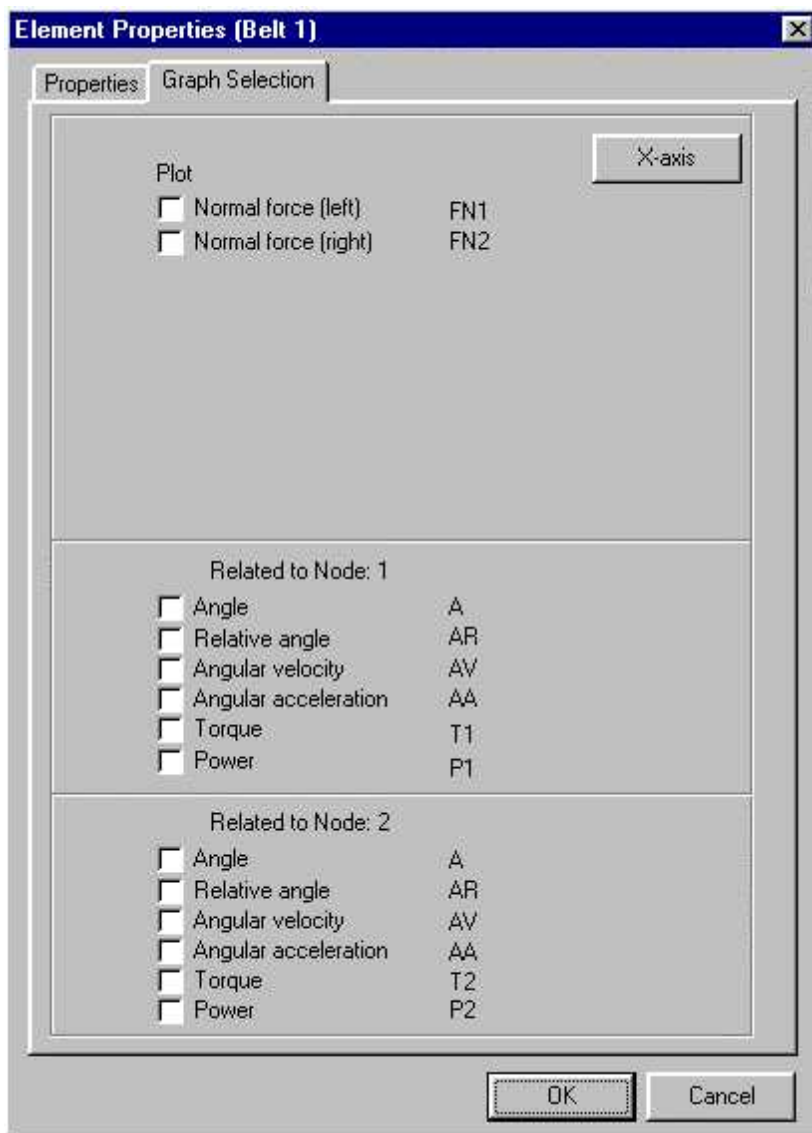




#### Properties of belt element

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

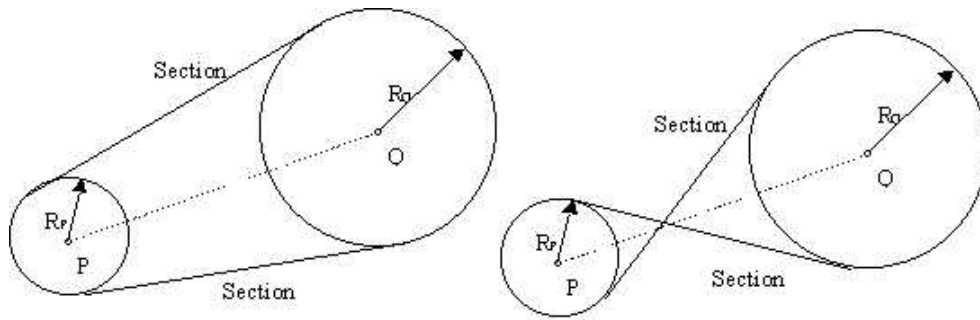
- absolute and relative angle, angular velocity, angular acceleration
- belt force that is transmitted between the two wheels (the two possible force transmission directions are indicated by the force FN1 and FN2, one of which is POSITIVE (= tension) and the other one is equal to zero).
- torque and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



**Calculable properties of belt element**

In any closed belt transmission one can distinguish two sections of the belt. It is assumed that there can only be tension forces in each section (pressure can not be present). In order to understand the results of the force analysis (internal element forces FN1 and FN2) it is essential to have a clear definition of these two sections. Section 1 of a belt element which is defined from node P to node Q always comes into contact with wheel Q on the left-hand side of the line PQ (see example).

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

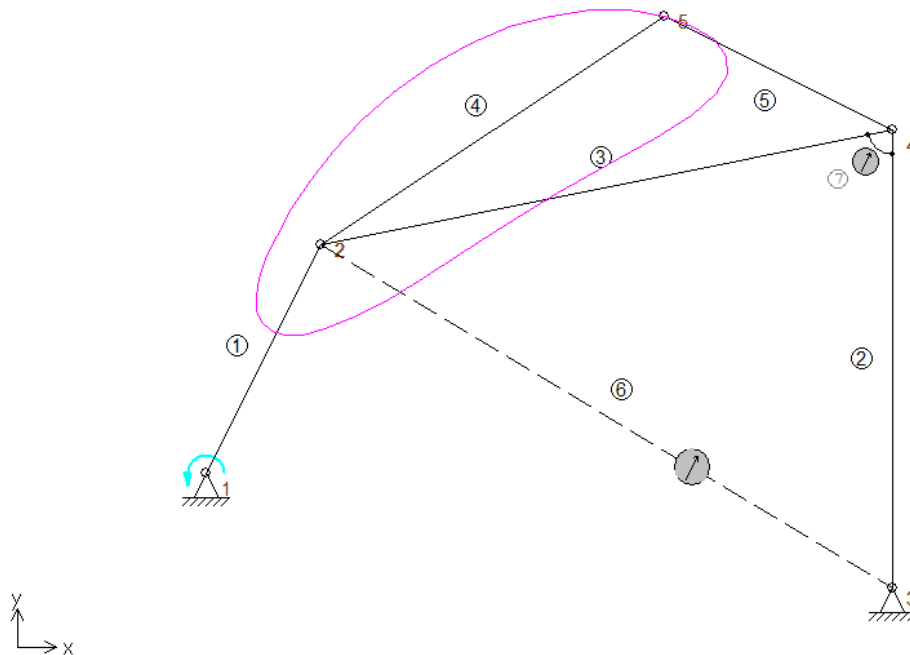


Definition of element forces FN1 and FN2 in a belt transmission

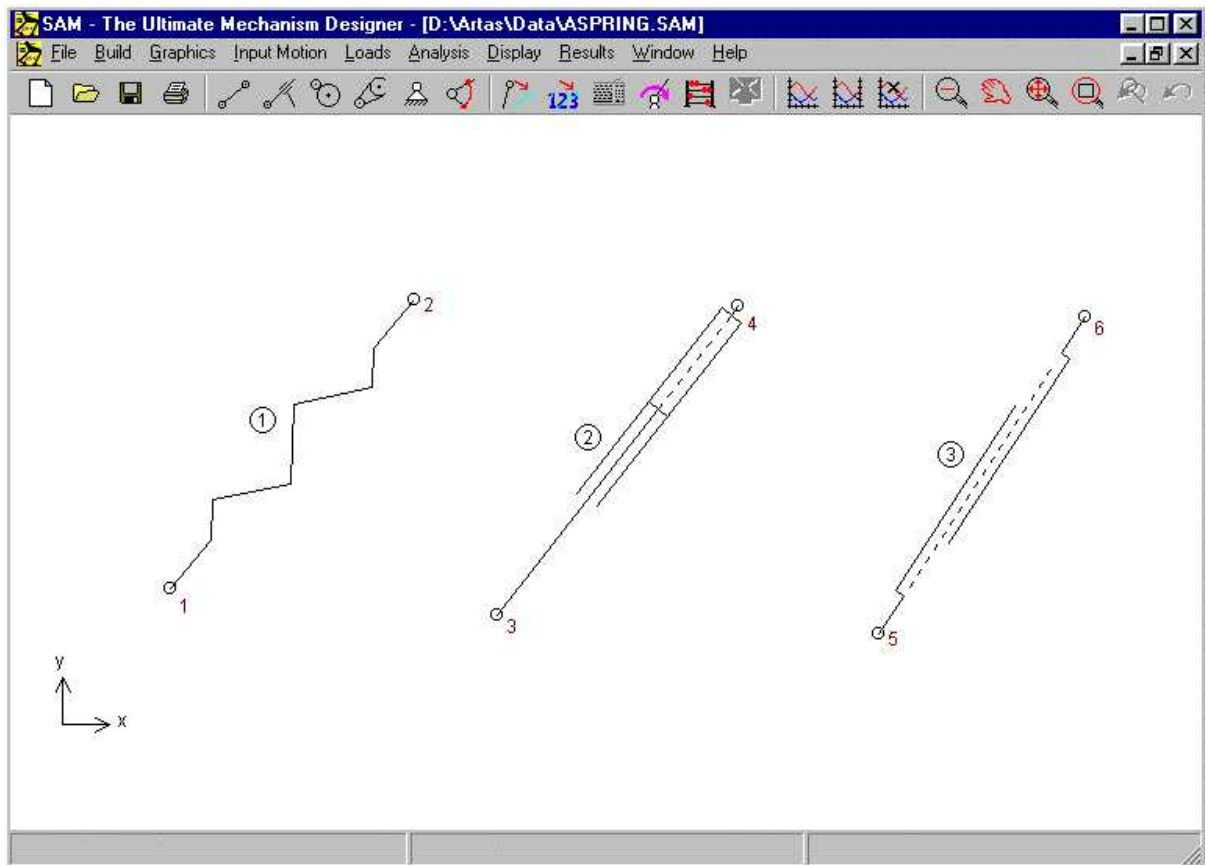
## 9.5 Sensor

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.

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.



## 9.6 Spring, Damper, Friction



**Translational spring, damper and friction element**

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

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

The spring element has two mechanical input properties :

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

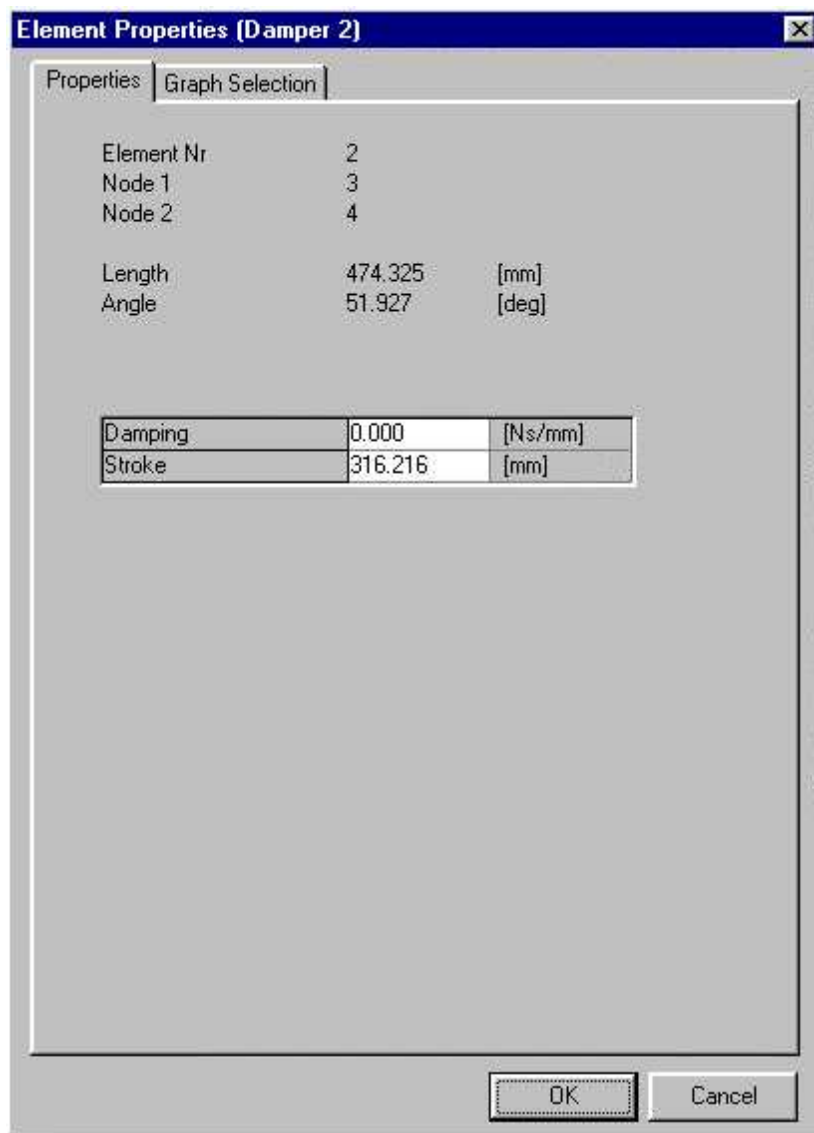
of the mechanism is preferred).

| Element Properties (Spring 1)                    |          |        |
|--------------------------------------------------|----------|--------|
| Element Nr                                       | 1        |        |
| Node 1                                           | 1        |        |
| Node 2                                           | 2        |        |
| Length                                           | 457.9405 | [mm]   |
| Angle                                            | 49.9605  | [deg]  |
| <input type="radio"/> Stiffness                  | 5.5      | [N/mm] |
| <input type="radio"/> Preload                    |          | [N]    |
| <input checked="" type="radio"/> Unloaded Length | 400      | [mm]   |

#### Properties of spring element

The damper element possesses two input properties :

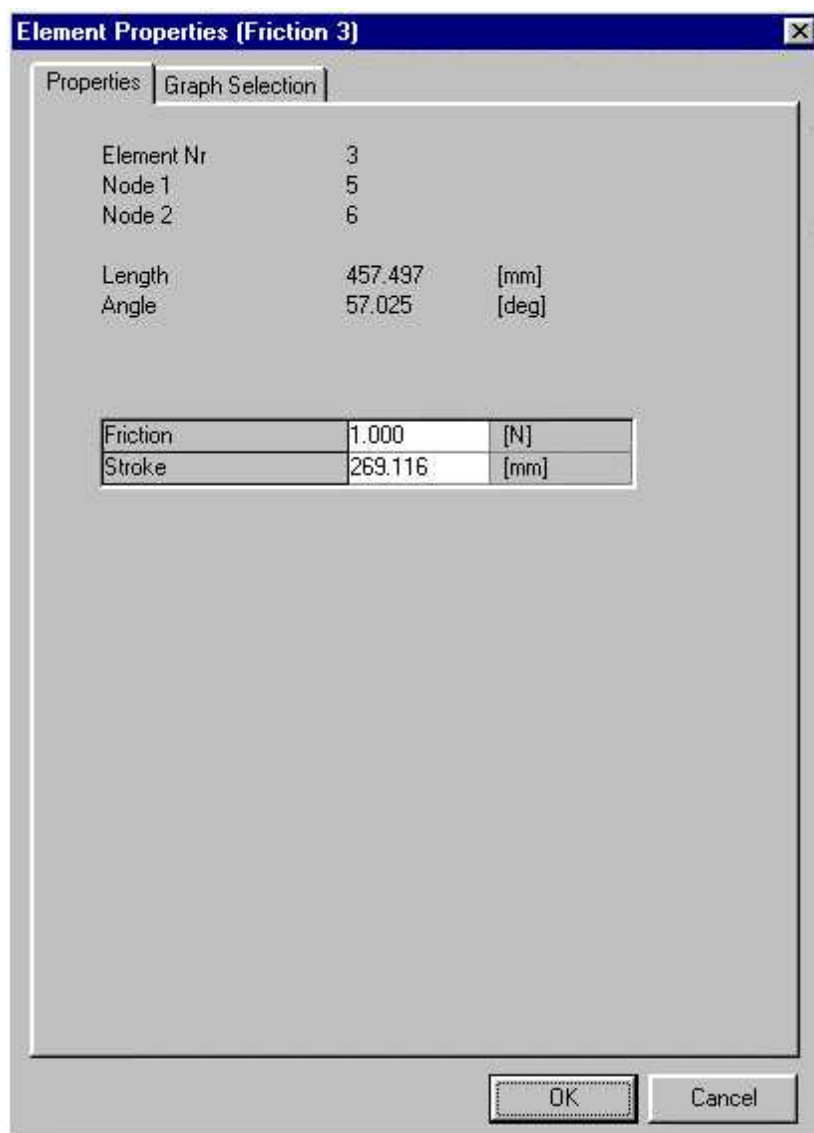
- damping coefficient
- stroke of the damper



Properties of damper element

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

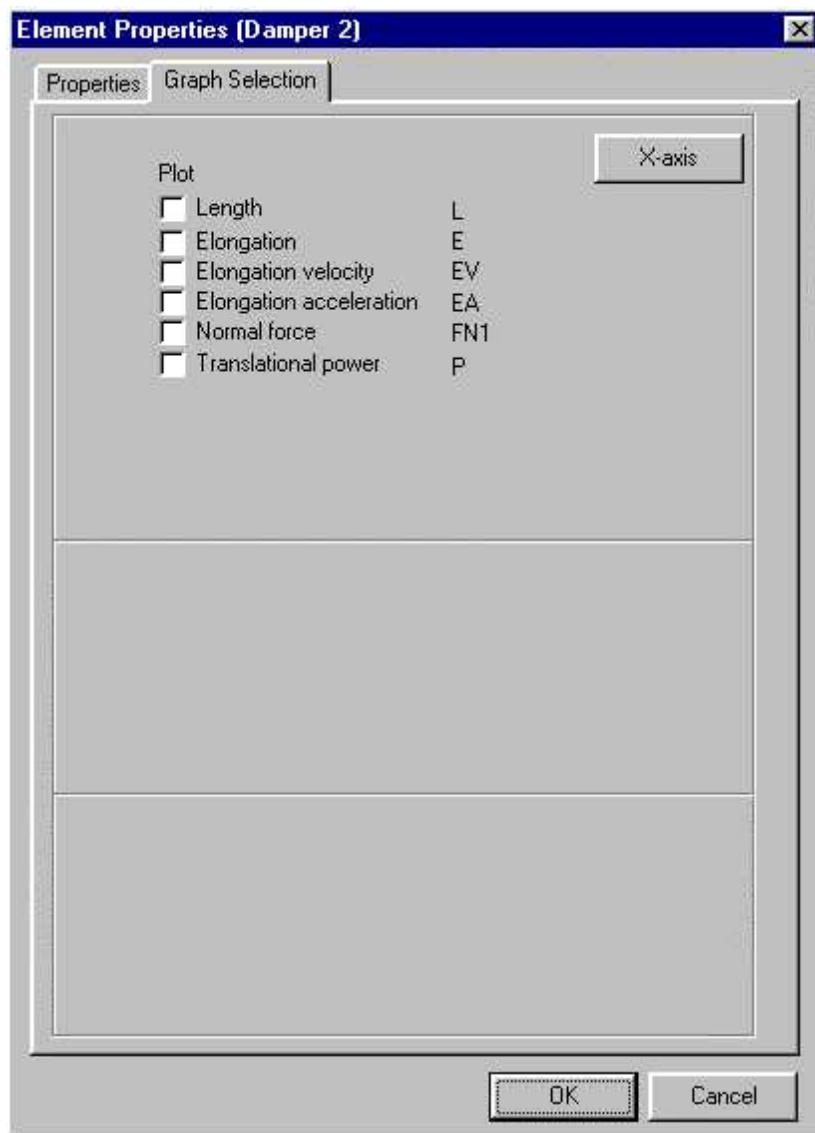
- constant friction force
- length of the stroke (visualization only)



Properties of friction element

The output choices of each of these elements is identical :

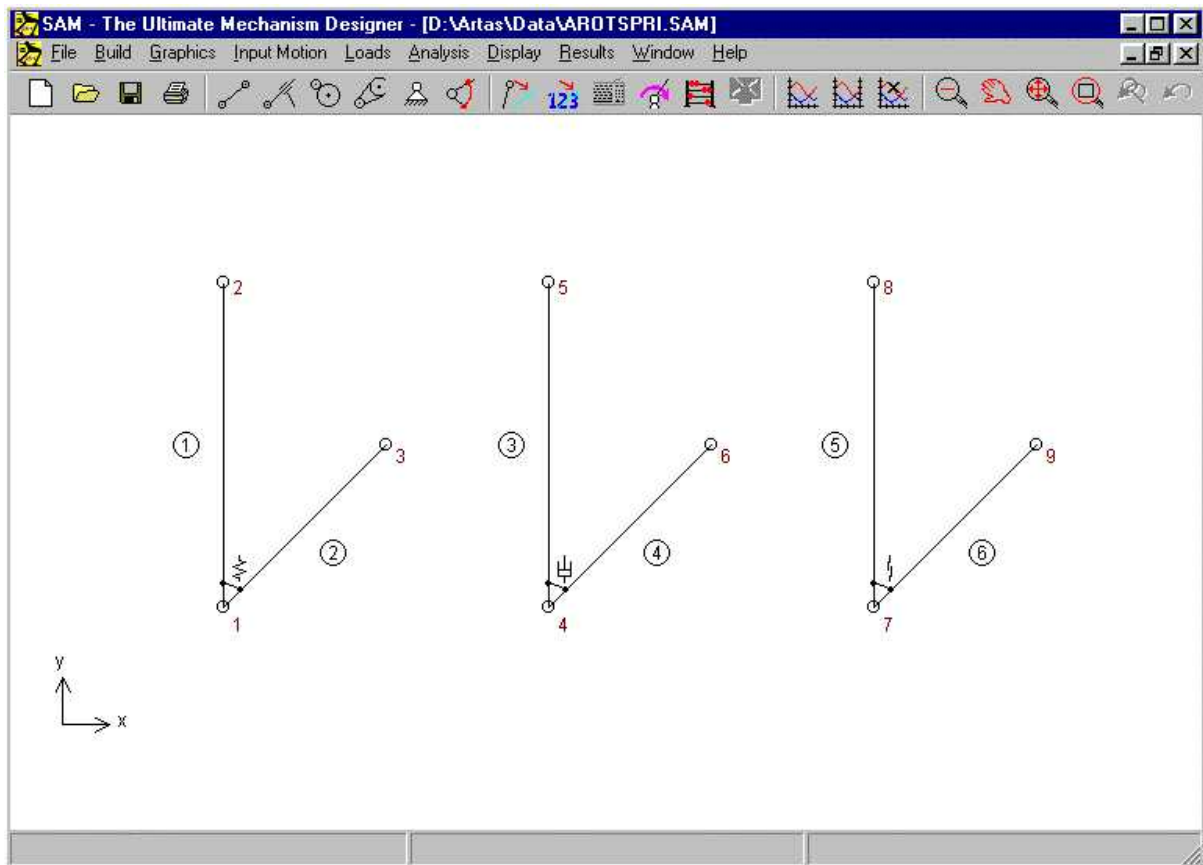
- length and its derivatives
- normal force (a positive value of the force FN1 indicates a tension force) and corresponding power



Calculable properties of spring, damper and friction element



## 9.7 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 exist on their own. They can only be attached to already existing elements.

---

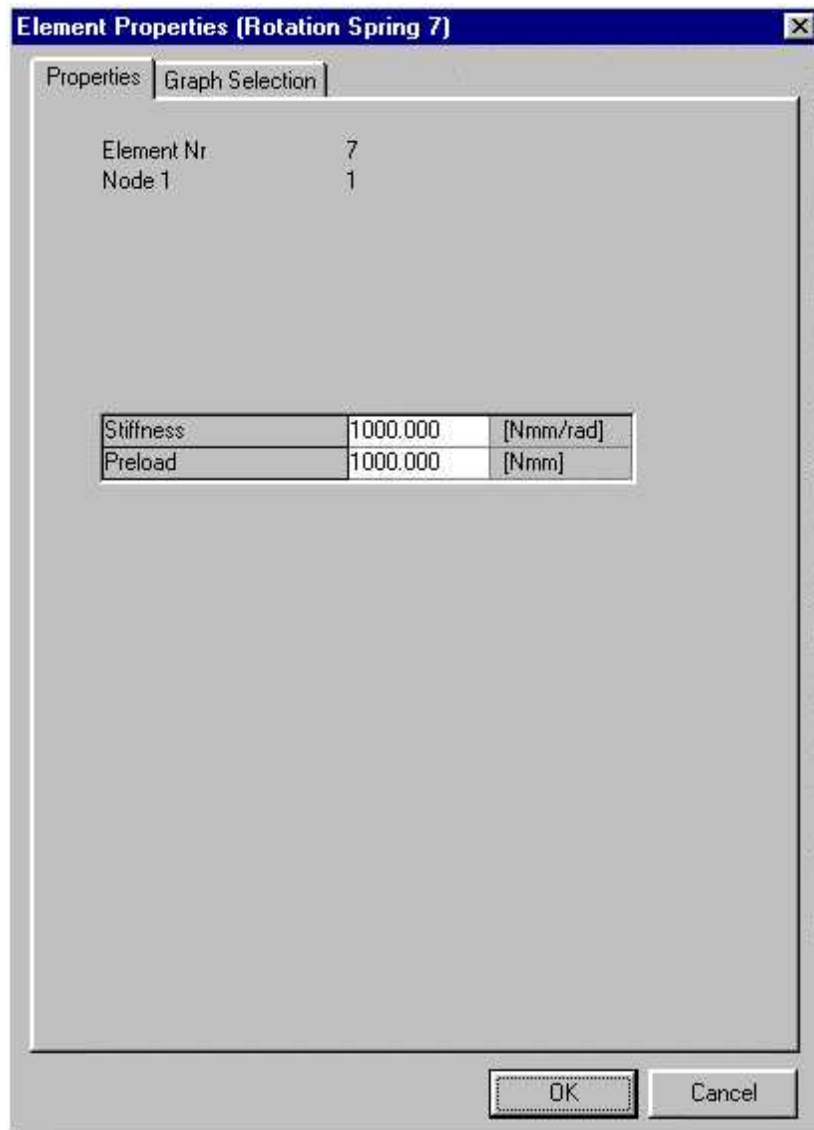
Creating a rotational spring, damper or friction between an existing element and the "world" is done by clicking somewhere on the workspace instead of clicking the second element.

---

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

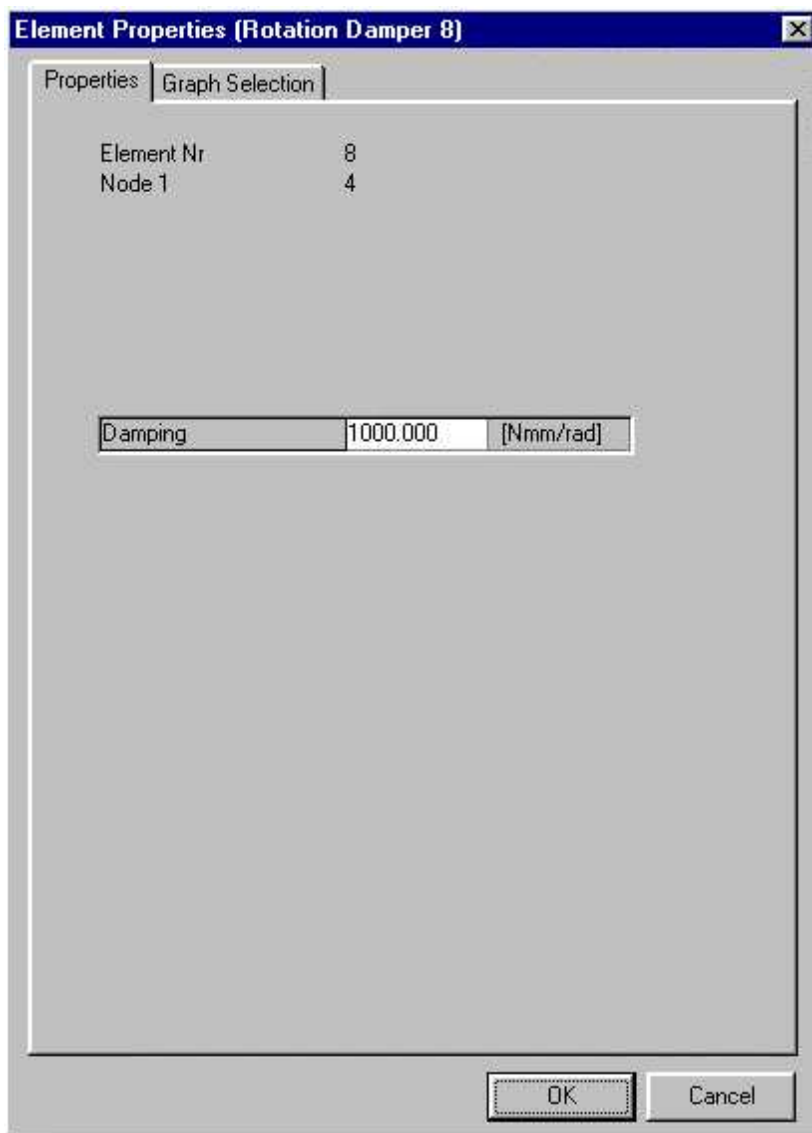
The rotational spring element has two mechanical properties :

- rotational stiffness
- preload in the initial position of the mechanism.



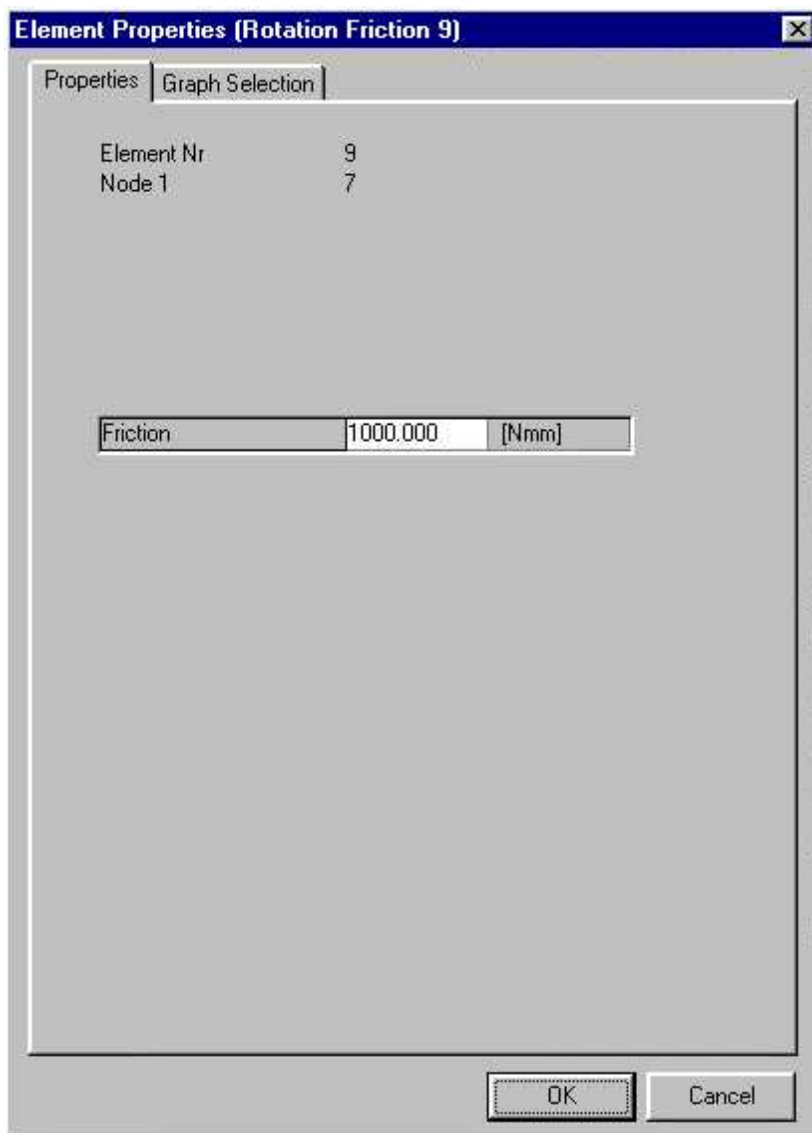
Properties of rotational spring element

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



**Properties of rotational damper element**

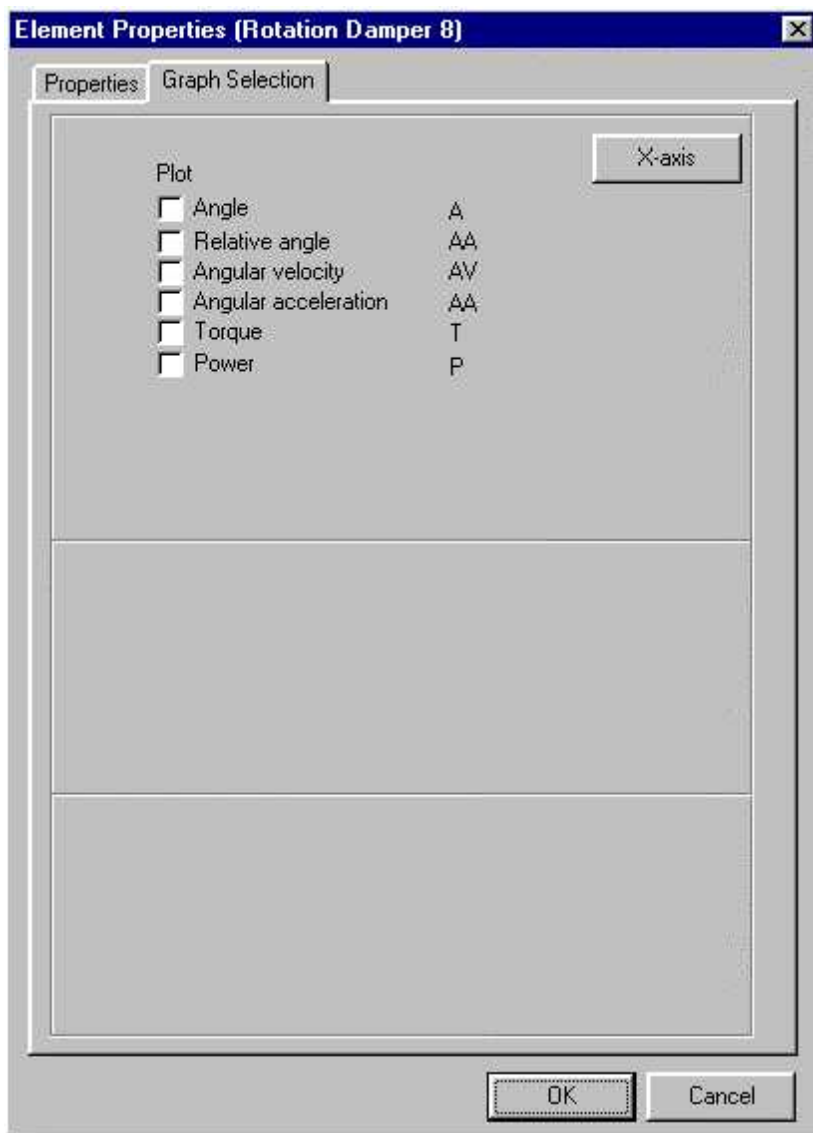
The rotational friction element is used to model pure coulomb friction, which is defined by the value of the friction torque.



**Properties of rotational friction element**

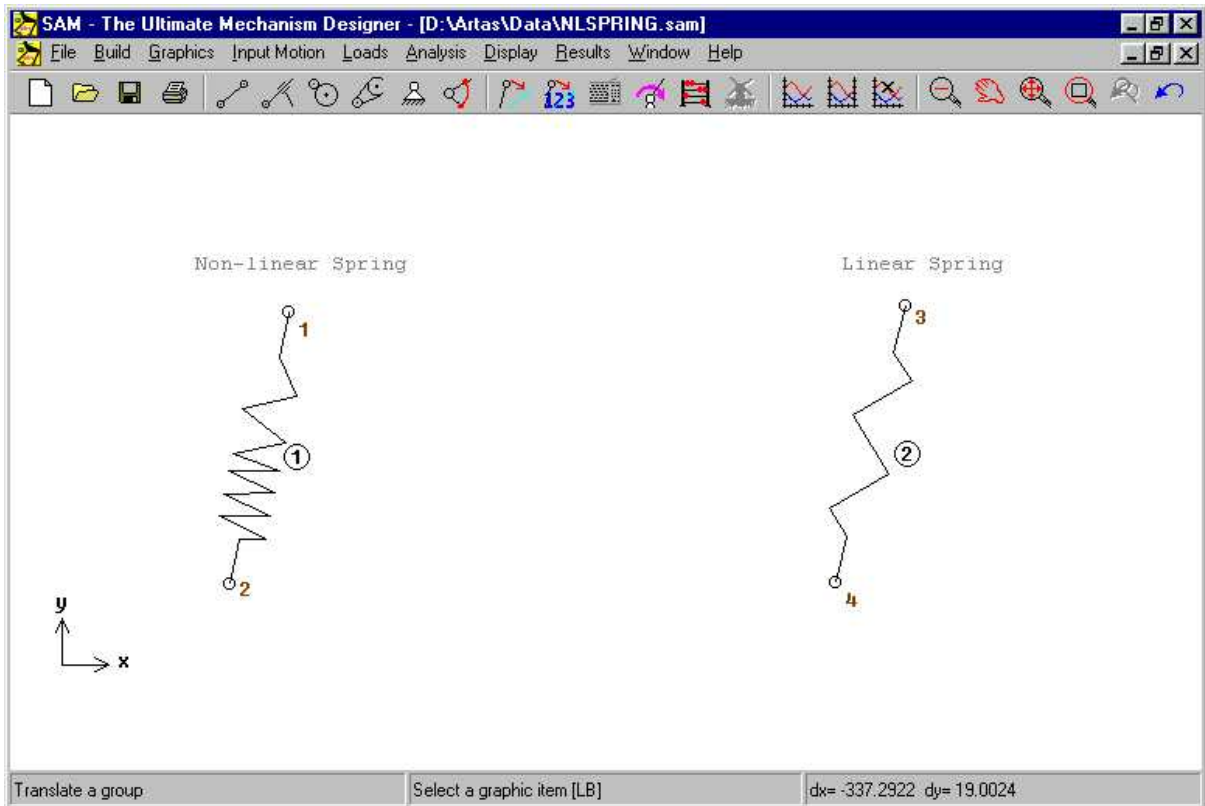
The output choices of each of these elements is identical :

- relative angle and its derivatives
- torque and corresponding power



Calculable properties of rotational spring, damper and friction element

## 9.8 Non-linear spring

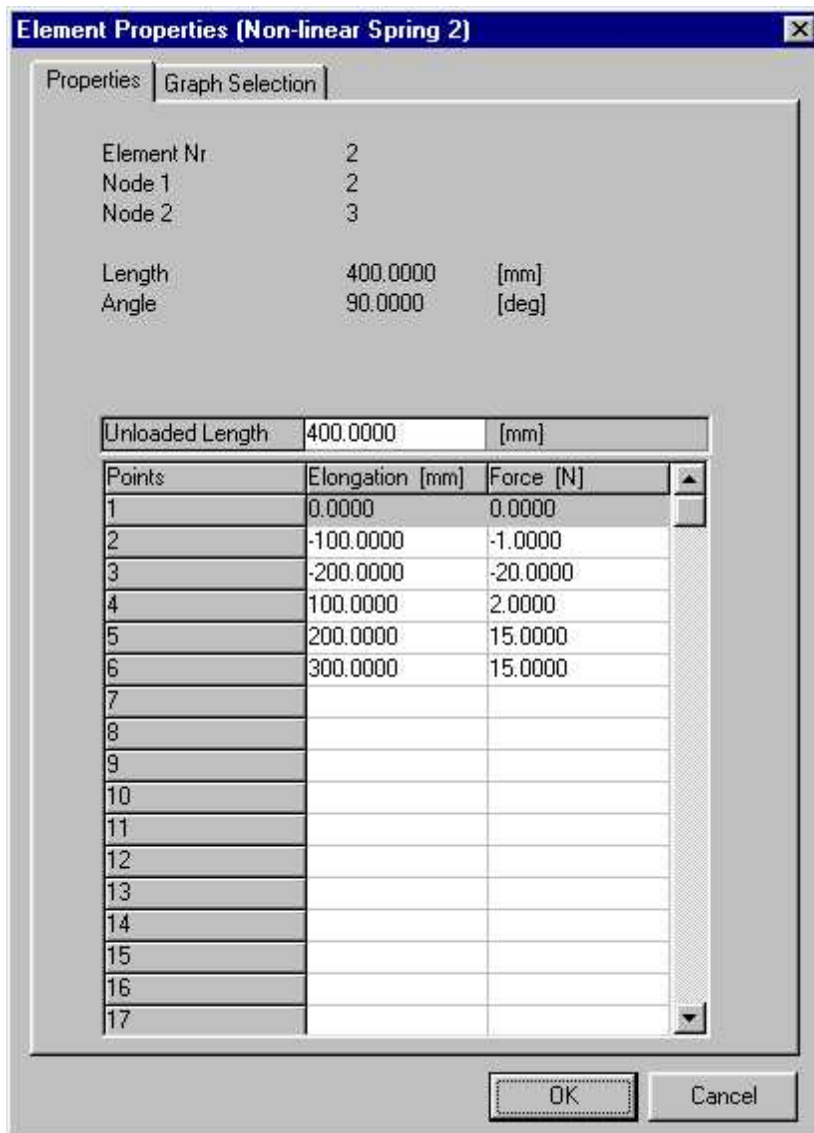


### Non-linear spring (and a standard linear spring)

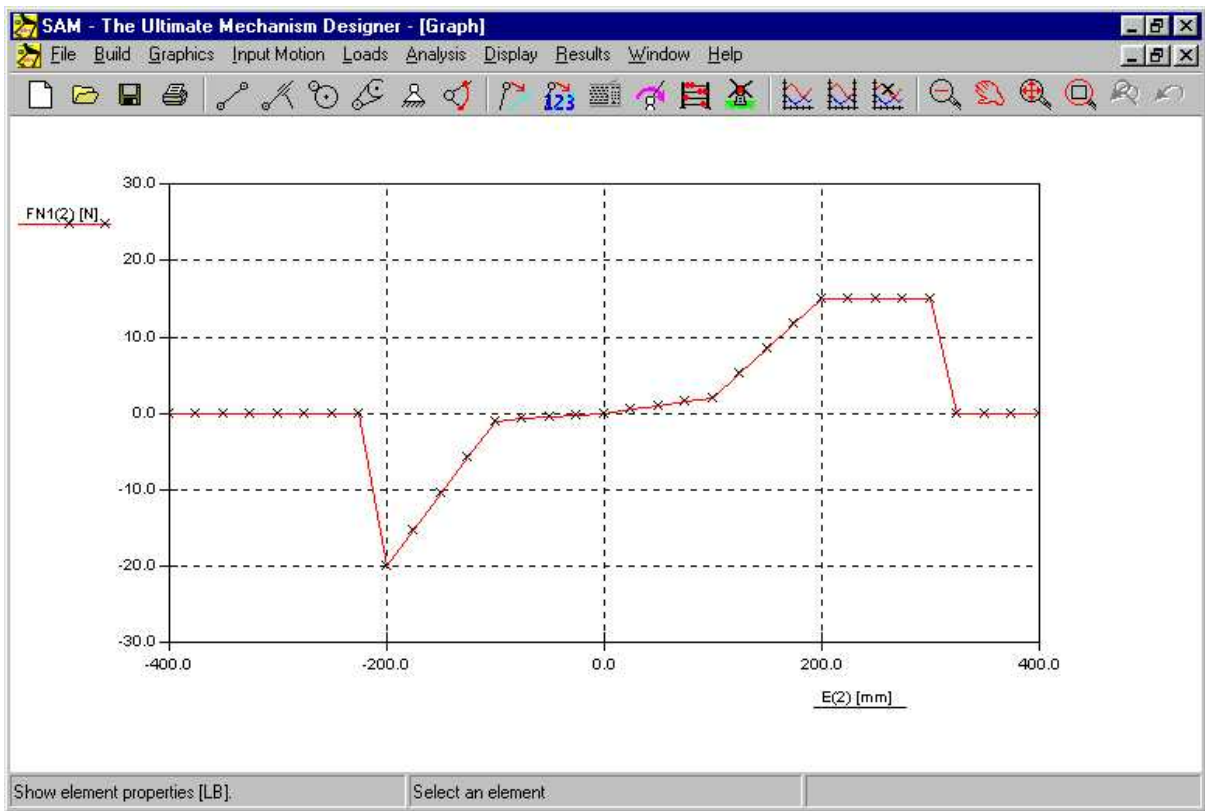
The non-linear spring is very similar to the linear spring in terms of functionality, however one can specify a non-linear relation between spring force and elongation of the spring. This non-linear characteristics is defined in the following way :

- 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.**



Definition of the non-linear elongation-force relation



Spring force as function of elongation



# Chapter

10

## 10 Integrated Analysis Environment

### 10.1 Keyboard and Mouse

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

#### **Left Mouse Button (single click)**

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

#### **Left Mouse Button (double click)**

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

#### **Right Mouse Button (single click)**

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

Copies the active SAM window to the clipboard.

#### **<Ctrl>+M**

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

**<Ctrl>+P**

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

**<Ctrl>+mouse move**

Horizontal or vertical alignment of element.

**<F2>**

Shortcut for Display/Animation. Start the animation.

**<F3>**

Shortcut for Build/Element Properties.

**<F4>**

Shortcut for File/Preferences

**<F5>**

Shortcut for Optimization/Run.

**<F8>**

Shortcut to return to Design Wizard

**<F9>**

Shortcut for Analysis.

**<F10>**

Shortcut for Display/Options

## 10.2 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.



**Gear** Enter a gear element.



**Belt** Enter a belt element.



**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.



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



**Analysis** Start the analysis of a mechanism.



**Optimization** Start Optimization.



**Animation** Start Animation.



**Graph/Select** Select items to be included in graph.



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



**Graph/Deselect** Remove item from graph.



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



**Zoom out** Zoom out.



**Zoom max** Zoom such that mechanism just fits into window.



**Zoom** Zoom (user can define zoom window).



**Pan** Move the viewing window



**Zoom previous** Return to previous zoom settings.



**Undo** Reverses the last action.



**Redo** Redo the last (undone) action.

## 10.3 Menu Commands

### 10.3.1 File Menu

**New**

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

**Wizard**

Lets you choose from a number of mechanism design wizards.

**Open**

Opens an existing mechanism project.

**Save**

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

**Save As**

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

**Import DXF**

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

**Export DXF**

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

**Project Notes**

Starts text editor in order to include project related notes.

**Project Documentation**

Generates a complete project documentation in ASCII-format.

**Statistics**

Displays some characteristics of the current project.

**Preferences**

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

**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.

### 10.3.2 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, Gear, Belt

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

#### Fix Node (X or Y)

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

#### Fix Angle

Fix an angle of an element relative to the world.

#### Fix relative Angle

Create a rigid fixation between the angles of two elements.

#### Unfix

Remove fixation of any kind

#### Move Node

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

#### Delete Element

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

#### Transform Mechanism

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

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

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

#### Element Properties

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

#### Node Properties

If you want to specify exact nodal coordinates select this menu item first, then click a node. A dialog will

be opened which lets you specify exact nodal coordinates in various manners.

### 10.3.3 Graphics Menu

The Graphics Menu lets you manipulate graphical data that is added to the mechanism 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).

#### **Circle**

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

#### **Centerline**

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

#### **Intersection**

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

#### **Text**

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

#### **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.

#### **Unattach**

Unattach 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".

### 10.3.4 Input Menu

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

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

**X-displacement**

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

**Y-displacement**

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

**Angle**

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

**Relative Angle**

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

**Elongation**



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

**Modify**

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

**Delete**

Lets you delete an existing input motion.

**More:**

[Input Dialog Box](#)

**10.3.4.1 Input Dialog Box**

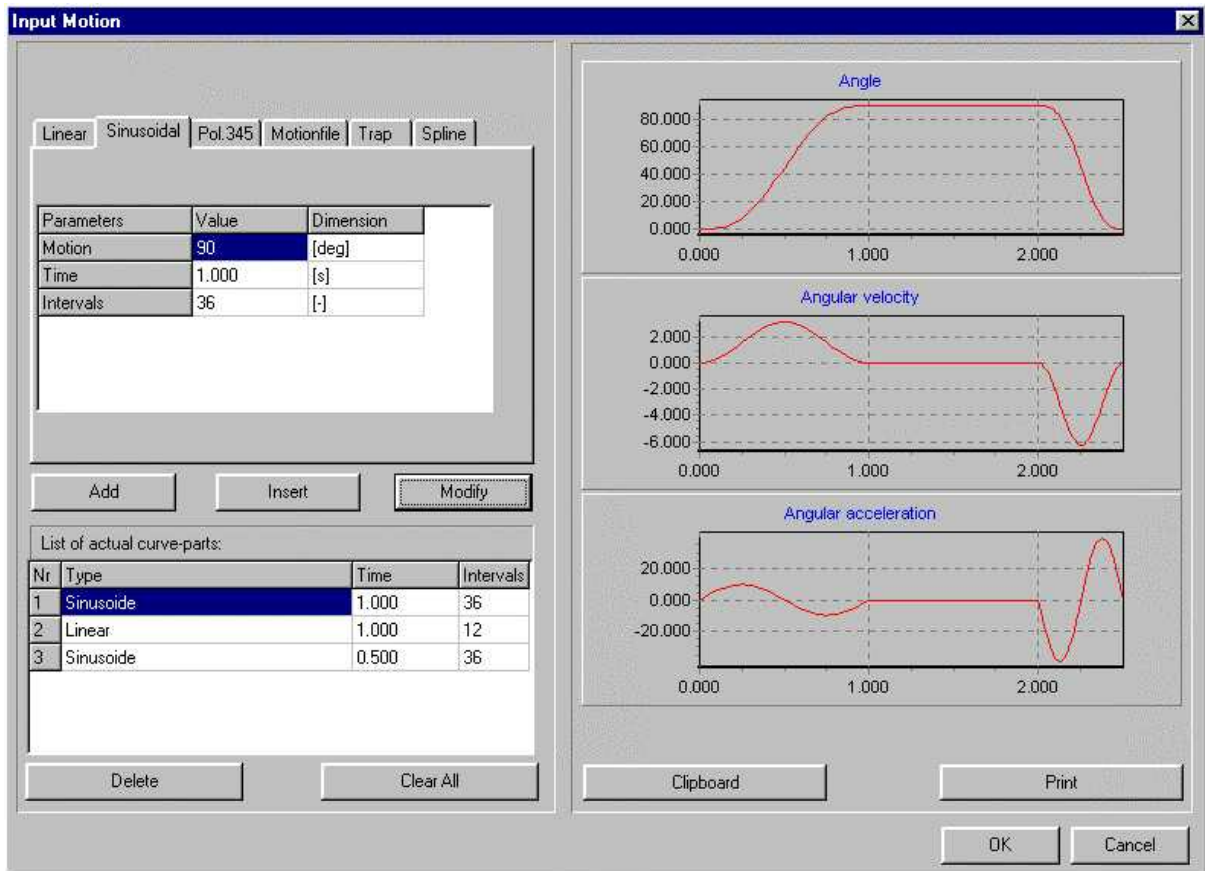
The heart of the input dialog box is formed by a list box, which contains a sequential list of selected basic motion blocks (the graph shows the equivalent of these data). Initially, this list is empty. It can be filled in the following manner :

**1. Activate one of the basic motion law tabs :**

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

**2. Enter the necessary input data****3. Add motion block to list by clicking the "Add" button**

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



Input Dialog Box

#### 4. Modify a motion block

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

#### 5. Insert an additional block in between

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

---

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

---

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

### 10.3.5 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 magnitude and direction of the forces. Forces can either be constant or a combination of linear pieces and constant pieces.

---

Please note that the created force function is related to the steps defined in the input motion. It is therefore not a separate function of time. Furthermore, load data must be present at every motion step.

---

**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 magnitude of the torque. Again, a torque can either be constant or a combination of linear pieces (as function of motion step NOT time) and constant pieces.

---

Please note that the created torque function is related to the steps defined in the input motion. It is therefore not a separate function of time. Furthermore, load data must be present at every motion step.

---

**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](#)

**10.3.5.1 Force/Torque Dialog Box**

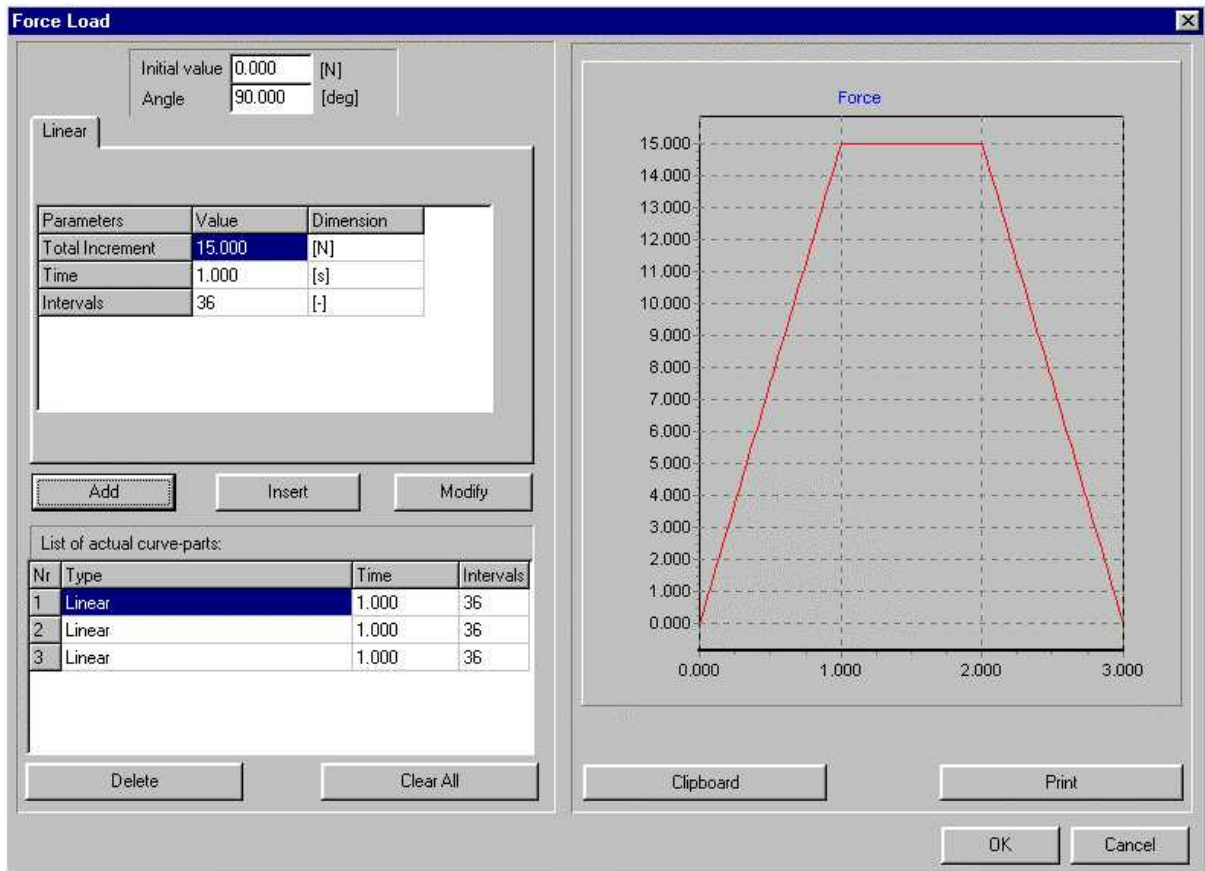
The heart of the force/torque dialog box is formed by a list box, which contains a sequential list of selected linear load blocks (the graph shows the equivalent of these data). Initially, this list is empty. It can be filled in the following manner :

**1. Enter the necessary data**

- direction of force (the specified direction is valid for the entire loading diagram)
- starting value of the force/torque in the first position
- force/torque increment of decrement value
- number of motion steps during which force/torque is adapted

**2. Add current block to list by clicking the "Add" button**

By repeating this process, it is very easy to define all sort of profiles. If you want to delete or modify an item in the list, simply activate it by clicking it once and press the "Delete" button or "Modify" button (see also the "Input Motion" instructions) .



**Force/Torque Dialog Box**

Please note that the created force/torque function is related to the steps defined in the input motion. It is therefore not a separate function of time. Furthermore, load data must be present at every motion step.

### 10.3.6 Analysis Menu

This menu command starts the analysis process. However, 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.

### 10.3.7 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.

**10.3.8 Display Menu****Animation**

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

**Create Movie**

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

**Screen to Clipboard Ctrl+C**

Sends the contents of the currently active 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.

**10.3.9 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.

**Copy to Clipboard Ctrl+C**

Copies the graph windows to the clipboard.

**Remove Curve**

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

**Toggle Curve**

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

**Graph Options**

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

**10.3.10 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 default values which are defined under File > Preferences > Windows.

**10.3.11 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.

**Check Updates**

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

**About SAM**

Displays information about SAM.

# Chapter

11

## 11 Literature

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

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

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

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

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

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

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

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

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



**Chapter**

**12**

## 12 Appendix

### 12.1 Analysis Problems

Various problems can be encountered while SAM is trying to perform an analysis. The most important errors and the most probable causes are listed here :

#### **Mechanism Underdetermined**

Some sort of constraint is missing :

- You forgot to define a support.
- You did not connect the angle between two beams.
- You intended to connect a beam to a belt/gear but forgot to connect the angles.
- Missing input motion.
- Two nodes were intended to coincide (= one node) but are created as separate nodes.

#### **Mechanism Overdetermined**

Too many constraints such that mechanism can not move:

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

#### **Convergence Problem --> Analysis stopped !**

Mechanism can not reach the desired position :

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

#### **No input defined**

Missing input motion :

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

### 12.2 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 SAM50.INI. If you do not want to change the SAM50.INI you can simply save the line drawing as DXFTEMPL.DXF.

If the template file, that is specified in SAM50.INI, can not be found, SAM prompts you to select one.

---

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 !)

---

## 12.3 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.

## 12.4 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.

## 12.5 What was new in SAM 6.0 ?

### Optimization (Professional version of SAM only)

Taking the initial design/topology as a starting point one can for example further improve the quality in which the trajectory of a coupler point equals the target trajectory by changing the geometry of the

mechanism within pre-defined ranges. Or one can minimize the peak or RMS value of the driving torque of a mechanism by adding a compensating mass and let SAM determine the optimal value of the mass and its position within the allowable range. Just as in the case of the trajectory optimization one can also specify a reference function and minimize the difference between the actual and the reference function. When designing for example fitness equipment one is generally seeking a predefined force as function of displacement.

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

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

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

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

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

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

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

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

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

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

### **Context-Sensitive Right Mouse Button**

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

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

### **News Alert**

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

**License Levels: Professional, Standard and Light**

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

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

## 12.6 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).

## 12.7 What was new in SAM 5.0 ?

**Optimization of the analysis-kernel:**

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

**"Real Time" analysis**

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

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

- cubic spline

- 2.order velocity profile
- table

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

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

**"Design Wizard" for approximated straight line coupler motion:**

- Watt
- Chebyshev
- Roberts
- Evans
- Hoecken

**"Design Wizard" for exact straight line coupler motion:**

Various alternatives incl. planetary gears, belt drives are available

**Generation of video file (AVI-format)**

## 12.8 What was new in SAM 4.2 ?

**Force analysis extended to mechanisms containing belts and gear**

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

**Simple CAD construction tools required for graphical mechanism synthesis**

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

**Display of the fixed/moving centrode and of the evolute**

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

**Various display options**

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

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

## 12.9 What was new in SAM 4.1 ?

### **Project Documentation**

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

### **Selectable Engineering Units and Number Representations**

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

### **Screenshots with filename, date/time and your company name**

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

### **Easy screenshot generation of motion/force profile**

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

### **Start of Analysis without Options Dialog**

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

### **New Display Options**

The menu Display/Options now contains additional items :

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

### **Export Results**

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

### **Dynamic link between animation and graph**

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

## 12.10 What was new in SAM 4.0 ?

### **CAD-Interface via Import/Export of DXF-data**

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

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

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

### **Extended display options**

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

**Zoom facilities**

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

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

**Export of results**

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

**Transformations**

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

**Expanded UNDO functionality**

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

**Input motions and loads dialog**

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

**Motion file**

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

**Manual animation**

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

**Keyboard entry of nodal coordinates**

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

**Norm or vector-length**

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



**Chapter**

**13**

## 13 Glossary of Terms

velocity hodograph

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

path

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

kinetostatics

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

Group

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

Evolute

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

Centrode

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

Moving Centrode

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

Fixed Centrode

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

# Index

## - 3 -

3-Position Synthesis (I) 57, 59  
3-Position Synthesis (II) 59

## - 4 -

4-Bar Mechanism 52, 54

## - A -

A quick example 22, 78, 127, 146, 149, 151, 167  
About this User's Guide 9, 10, 15, 16, 18, 22, 68,  
104, 146, 147, 151, 165, 167  
Analysis Menu 156  
Analysis Problems 162  
Analysis Results 13  
Approximated Straight Line Mechanism 62, 63, 64,  
65, 165

## - B -

Beam 118  
Bearing Forces 75  
Belt 127  
Belt drive with external force 102  
Belt-Coupled Dyad 62  
Building a new mechanism (without Design Wizard)  
38, 39, 150, 152, 153  
Building Multi-Node Linkages 72

## - C -

CAD Interface 13, 22, 68, 78, 149, 151, 162, 167  
Constrained Optimization 108  
Crank Rocker Mechanism 94

## - D -

Design 12, 54, 62, 165  
Display Menu 157, 166  
Double Crank Mechanism 98

Driving Torque Minimization 100, 104  
DXF Template File 162

## - E -

European Academic Software Award 16  
Exact Nodal Coordinates 68, 146, 147, 150  
Exact Straight Line Mechanism 59  
Example of a 4-bar mechanism (motion analysis)  
39, 52, 68, 73, 84, 85, 86, 89, 118, 146, 147, 150, 167  
Exporting a motion file 70

## - F -

Force/Torque Dialog Box 155  
Forces in Moving Nodes 75  
Format 69

## - G -

Gear 123  
Geared Double-Crank Mechanism 60  
General (nodes) 52  
Graphics Menu 151

## - H -

Help Menu 158  
History  
    Present and Future 15, 68, 69, 73, 86, 94, 167  
Hypo-Cyclic Planetary Gear Mechanism 61

## - I -

Importing a motion file 70  
Inclined Linear Guiding 71  
Input Dialog Box 153, 165  
Input Motion 13, 33, 38, 39, 57, 59, 69, 70, 71, 104,  
146, 152, 153, 154, 155, 157, 162, 165, 167  
Installing SAM 18  
Introduction 84

## - K -

Keyboard and Mouse 146

**- L -**

LB 146  
Linear Actuator 71, 118, 152  
Loads Menu 154

**- M -**

Modelling 13, 15, 39, 54, 62, 71, 72, 73, 102, 121,  
123, 127, 132, 137, 147, 150, 151, 152, 162, 165, 166  
Modelling a rack/pinion 73  
Multiple Input Motions 71

**- N -**

Node-Locked License 18  
Non-linear spring 142

**- O -**

Optimization Menu 156  
Optimization Objective 85, 106  
Optimization of trajectory 106  
Optimization Options 89, 156  
Overview - Capabilities - Optimization 14, 15

**- P -**

Planetary Gear-Coupled Dyad 61  
Planetary Mechanism (Hypo-Cyclic) 98  
Post-Processing 14, 22, 33, 38, 39, 85, 86, 94,  
100, 104, 106, 157, 165, 166, 167  
Purchase Agreement 10, 22, 78, 137, 146, 151

**- Q -**

Quick-Return Mechanism 94

**- R -**

RB 146  
Reference or Target Definition 86  
Running a sample project 22, 38, 39, 78, 149, 156,  
157

**- S -**

Settings Evolutionary Algorithm 90  
Settings Simplex Algorithm 91  
Slider 121  
Slider Driven Gear Transmission 97  
Symmetric Slider-Crank Mechanism 60

**- T -**

The Design Process 12, 13, 22, 33, 39, 52, 53, 54,  
57, 59, 60, 71, 72, 97, 104, 106, 147, 150, 151  
Transfer Mechanism (incl. gravity effects) 100  
trial license 18

**- U -**

Units 68, 69, 70, 152, 153, 157, 167  
User Defined Results 79  
Using Graphics 78  
Using the Design Wizard to create a mechanism  
33, 52, 57

**- W -**

What is SAM ? 8, 9, 13, 14, 15, 33, 38, 39, 61, 68,  
72, 73, 75, 78, 84, 85, 86, 89, 90, 97, 98, 104, 106,  
123, 132, 137, 146, 147, 149, 150, 151, 152, 156, 157,  
162, 163, 165, 166  
What was new in SAM 4.0 ? 167  
What was new in SAM 4.1 ? 167  
What was new in SAM 4.2 ? 166  
What was new in SAM 5.0 ? 165  
What was new in SAM 5.1 ? 165  
What's new in SAM 6.0 ? 9, 13, 14, 15, 22, 38, 84,  
86, 89, 90, 91, 100, 102, 104, 123, 127, 132, 137, 142,  
147, 150, 158, 163, 165, 166  
Window Menu 158